

ENERGYPLUS™

EnergyPlus Engineering Reference

The Reference to EnergyPlus Calculations

(in case you want or need to know)

Date: April 5, 2009

COPYRIGHT © 1996-2009 THE BOARD OF TRUSTEES OF THE UNIVERSITY OF ILLINOIS AND THE REGENTS OF THE UNIVERSITY OF CALIFORNIA THROUGH THE ERNEST ORLANDO LAWRENCE BERKELEY NATIONAL LABORATORY.
ALL RIGHTS RESERVED. NO PART OF THIS MATERIAL MAY BE REPRODUCED OR TRANSMITTED IN ANY FORM OR BY ANY MEANS WITHOUT THE PRIOR WRITTEN PERMISSION OF THE UNIVERSITY OF ILLINOIS OR THE ERNEST ORLANDO LAWRENCE BERKELEY NATIONAL LABORATORY.
ENERGYPLUS IS A TRADEMARK OF THE US DEPARTMENT OF ENERGY.

TABLE OF CONTENTS

Overview	1
Document Overview	1
General Modeling Overview	1
Figure 1. EnergyPlus Program Schematic	1
Simulation Manager	1
Table 1. Simulation Flags	2
Integrated Solution Manager	3
Figure 2. Sequential Simulation Supply/Demand Relationship.	3
Figure 3. Schematic of Simultaneous Solution Scheme	4
Basis for the Zone and System Integration	4
Zone Capacitance Multiplier	7
Summary of Predictor-Corrector Procedure	7
System Control	7
Figure 4. Simplified Single Zone Draw Through System	9
Figure 5. Simplified Variable Volume System.	9
Figure 6. Idealized Variable Volume System Operation	10
Moisture Predictor-Corrector	12
Moisture Prediction	13
Moisture Correction	14
Summary of Time Marching Solution	15
Summary of Timestep Model Formulation	15
Figure 7. Diagram of TimeStep Formulation	16
Zone Update Method	16
Variable Timestep	17
Simultaneous Solution of Plant/System Water Loop	18
References	18

TABLE OF CONTENTS

Surface Heat Balance Manager / Processes	20
Conduction Through The Walls.....	20
Conduction Transfer Function Module	20
Calculation of Conduction Transfer Functions	21
Figure 8. Two Node State Space Example.	23
EnergyPlus CTF Calculations	23
Figure 9. Multiple, staggered time history scheme	25
Figure 10. Sequential interpolation of new histories.....	25
Figure 11. Master history with interpolation.....	26
References	26
Conduction Finite Difference Solution Algorithm.....	26
Figure 12. Effects of Large PCM on Outside Zone Surface	28
Combined Heat and Moisture Transfer (HAMT) Model.....	28
Overview	28
HAMT Nomenclature.....	28
Table 2. Combined Heat and Moisture Transfer Model Nomenclature	29
HAMT Model Description	30
References	33
Effective Moisture Penetration Depth (EMPD) Model.....	34
Overview	34
EMPD Model Description	34
EMPD Value Determination	35
Figure 13. Limit of Effective Penetration Depth Values for Various Vapor Diffusivities at Different Ambient Excitations.....	36
EMPD Nomenclature	36
References	37
Outside Surface Heat Balance.....	37

TABLE OF CONTENTS

Figure 14. Outside Heat Balance Control Volume Diagram	37
External Shortwave Radiation	38
External Longwave Radiation	38
Table 3. Nomenclature List of Variables.....	38
References	40
Atmospheric Variation	41
Table 4. Atmospheric Variables at Two Different Altitudes above Ground Level.....	41
Table 5. Wind Speed Profile Coefficients (ASHRAE Fundamentals 2005).	43
Outdoor/Exterior Convection	43
Table 6. Roughness Coefficients D, E, and F.	44
Table 7. Nomenclature List of Variables.....	45
Table 8. Surface Roughness Multipliers (Walton 1981).	46
Table 9. Nomenclature List of Variables.....	47
Table 10. MoWiTT Coefficients (Yazdanian and Klems 1994).....	48
Table 11. Nomenclature List of Variables.....	48
Exterior/External Conduction	49
References	49
Inside Heat Balance	50
Figure 15. Inside Heat Balance Control Volume Diagram.....	50
Internal Long-Wave Radiation Exchange.....	50
Internal Short-Wave Radiation	52
Interior Conduction.....	52
Interior Convection	52
Figure 16. Ceiling Diffuser Correlation for Floors	54
Figure 17. Ceiling Diffuser Correlation for Ceilings	55

TABLE OF CONTENTS

Figure 18. Ceiling Diffuser Correlation for Walls	55
Infrared Radiation Transfer Material	57
Figure 19. Schematic View of Stacked Zones.....	58
Figure 20. Schematic Model of a UFAD Zone.....	59
Radiation Exchange Basics	59
Figure 21. Radiation Exchange Model	59
Radiation Transfer Surface Details	60
Behavior Checks	60
Figure 22. Single Zone Model	61
Figure 23. Stacked Three Zone Model	62
Figure 24. Comparison of Single and Three Stacked Zones.....	63
Transparent Insulation Material (TIM)	63
Introduction	63
Comparison of Opaque and Transparent Insulation	64
Figure 25. Energy Flows of Opaquely and Transparently Insulated Walls (Wood and Jesch 1993).....	64
Types of Transparent Insulation Materials	64
Figure 26. Geometrical Categories of Classification for Transparent Insulation Material (Wood and Jesch 1993).	65
TIM- Basic Mathematical Model	65
Figure 27. Cross Section of TIM and wall, showing energy flow	65
Sample Test Run Cases: – Comparison	67
Table 12. TIM with Summer Conditions	68
Table 13. TIM with Winter Conditions.....	68
References	69
Surface Heat Balance With Moveable Insulation	69
Basic Heat Balance Cases.....	69

TABLE OF CONTENTS

Figure 28. Thermal Network for Simple Outside Surface Coefficient	71
Figure 29. Thermal Network for Detailed Outside Surface Coefficient.....	71
Figure 30. Thermal Network for Outside Moveable Insulation	71
Heat Balance Cases	71
FORTTRAN Algorithm Examples	73
Fortran Variable Descriptions.....	74
Table 14. Fortran Variables and Descriptions	74
References.....	77
Advanced Surface Concepts.....	78
Exterior Naturally Vented Cavity	78
Baffle Heat Balance	78
Figure 31. Baffle Surface Heat Balance	78
Cavity Heat Balance.....	80
Figure 32. Cavity Air Heat Balance	80
Underlying Heat Transfer Surface.....	82
Solar and Shading Calculations	82
Local Wind Speed Calculations	82
Table 15. Terrain-Dependent Coefficients (ASHRAE 2001).	83
Convection Coefficients	83
Radiation Coefficients	83
References.....	84
Green Roof Model (EcoRoof).....	84
Overview	84
Green Roof Model Description.....	85
Figure 33. The Energy Balance for a Green Roof.	85
Linearization.....	90

TABLE OF CONTENTS

Final Equations	91
Green Roof Nomenclature	92
References	93
Climate, Sky and Solar/Shading Calculations.....	95
Climate Calculations	95
Figure 34. Default Daily Temperature Range Profile.....	96
ASHRAE Clear Sky Solar Model.....	96
Table 16. Extraterrestrial Solar Irradiance and Related Data Note: Data are for 21st day of each month during the base year of 1964.	96
Zhang-Huang Solar Model	97
Design Week Specification	97
Sky Radiance Model	98
Figure 35. Schematic view of sky showing solar radiance distribution as a superposition of three components: dome with isotropic radiance, circumsolar brightening represented as a point source at the sun, and horizon brightening represented as a line source at the horizon.....	98
Table 17. Variables in Anisotropic Sky Model and Shadowing of Sky Diffuse Radiation	99
Sky Diffuse Solar Radiation on a Tilted Surface	100
Table 18. F_{ij} Factors as a Function of Sky Clearness Range.....	101
Shadowing of Sky Diffuse Solar Radiation.....	101
Shadowing of Sky Long-Wave Radiation.....	102
Shading Module	103
Shading and Sunlit Area Calculations	103
Figure 36. Overall Shadowing Scheme Depiction	103
Solar Position	103
Table 19. Relationship of Angles (degrees) to Time	104
Figure 37. Solar Position Illustration.....	104
Surface Geometry	104

TABLE OF CONTENTS

Figure 38. EnergyPlus Coordinate System	105
Shadow Projection	106
Figure 39. Basic shadowing concept structure.....	107
Figure 40. Illustration of Shadow Clipping	108
Homogeneous Coordinates	108
Overlapping Shadows	109
Figure 41. Point a – Vertex of A Enclosed by B	110
Figure 42. Surface A Totally Overlaps Surface B.....	110
Figure 43. Figure Formed from Intercept Overlaps Between A and B	111
Figure 44. Complex Overlapping Condition.....	111
Table 20. Surface / Area Characteristic / Convention	112
Figure 45. Multiple Shadow Overlaps.....	112
Solar Gains	112
Solar Distribution.....	113
Figure 46. Illustration of Convex and Non-convex Zones.....	114
Details of the Interior Solar Distribution Calculation	114
Figure 47. Vertical section through a two-zone building showing where transmitted beam solar falls. Some of the beam solar from exterior window EW is absorbed by the floor, D, interior wall, B, and interior window, IW. Some is transmitted by IW to the adjacent zone, Z2. Aoverlap is the irradiated area of a surface projected back onto the plane of EW. Beam reflected by D, B and IW contributes to the interior short-wave radiation flux in Z1.....	117
Ground Reflectances	120
Ground Reflectances (Snow)	120
References.....	120
Solar Radiation Reflected from Exterior Surfaces	121
Figure 48. Examples of solar reflection from shadowing surfaces in the Shading series of input objects. Solid arrows are beam solar radiation; dashed arrows are diffuse solar radiation. (a) Diffuse reflection of beam solar radiation from the top of an overhang. (b)	

TABLE OF CONTENTS

Diffuse reflection of sky solar radiation from the top of an overhang. (c) Beam-to-beam (specular) reflection from the façade of an adjacent highly-glazed building represented by a vertical shadowing surface.....	121
Figure 49. Solar reflection from building surfaces onto other building surfaces. In this example beam solar reflects from a vertical section of the building onto a roof section. The reflection from the window is specular. The reflection from the wall is diffuse.....	122
Figure 50. Shadowing by the building itself affects beam solar reflection from the ground. Beam-to-diffuse reflection from the ground onto the building occurs only for sunlit areas, A and C, not for shaded area, B. Shadowing by the building also affects sky solar reflection from ground (not shown).	122
Diffuse Reflection of Beam Solar and Sky Solar Radiation.....	122
<i>Receiving points</i>	123
Figure 51. Vertical rectangular exterior heat transfer surface showing location of receiving points for calculating incident solar radiation reflected from obstructions.	123
<i>Rays</i>	123
Figure 52. Two-dimensional schematic showing rays going outward from a point on a receiving surface. Rays 1-6 hit the ground, rays 7-11 hit an obstruction, and rays 12-15 hit the sky.	124
Sky Solar Radiation Diffusely Reflected from Obstructions	124
Sky Solar Radiation Diffusely Reflected from the Ground.....	125
Figure 53. Two-dimensional schematic showing rays going upward from a ground hit point.....	125
Beam Solar Radiation Diffusely Reflected from Obstructions	126
Beam Solar Radiation Diffusely Reflected from the Ground	126
Beam Solar Radiation Specularly Reflected from Obstructions	127
Figure 54. Two-dimensional schematic showing specular reflection from an obstruction such as the glazed façade of a neighboring building. The receiving point receives specularly reflected beam solar radiation if (1) DB passes through specularly reflecting surface EF, (2) CD does not hit any obstructions (such as RS), and (3) AC does not hit any obstructions (such as PQ).	127
Daylighting and Window Calculations	129
Daylighting Calculations.....	129

TABLE OF CONTENTS

Daylight Factor Calculation	129
Table 21. Variables in Daylighting Calculations	129
Overview	133
Interior Illuminance Components	133
Daylight Factors	133
Sky Luminance Distributions	134
Figure 55. Angles appearing in the expression for the clear-sky luminance distribution.....	136
Direct Normal Solar Illuminance	136
Exterior Horizontal Illuminance	136
Direct Component of Interior Daylight Illuminance	137
Figure 56. Geometry for calculation of direct component of daylight illuminance at a reference point. Vectors R_{ref} , W_1 , W_2 , W_3 and R_{win} are in the building coordinate system.	138
Internally-Reflected Component of Interior Daylight Illuminance	139
Transmitted Flux from Sky and Ground	140
Transmitted Flux from Direct Sun	141
Luminance of Shaded Window	142
Daylight Discomfort Glare	142
Figure 57. Geometry for calculation of displacement ratios used in the glare formula.....	143
Table 22. Position factor for glare calculation.....	143
Time-Step Daylighting Calculation	144
Overview	144
Table 23. Variables in Time-Step Calculations.....	144
Time-Step Sky Luminance	146
Interior Illuminance	147
Glare Index.....	148

TABLE OF CONTENTS

Lighting Control System Simulation	149
Figure 58. Control action for a continuous dimming system.....	150
Figure 59. Stepped lighting control with three steps.....	151
References	151
DElight Daylighting Calculations	152
DElight Daylight Factor Calculation Differences from EnergyPlus Detailed Methods	153
Figure 60. Bi-Directional Transmittance Data.....	154
DElight Time-Step Interior Daylighting Calculation Differences from EnergyPlus Detailed Methods.....	154
References	155
Daylighting Devices.....	155
Tubular Daylighting Devices	155
Figure 61. Tubular Daylighting Devices.....	155
Figure 62. Dome and Diffuser Surfaces	156
Figure 63. Discrete Ray in a Pipe.....	157
Figure 64. Pipe Transmittance Comparison	158
Daylighting Shelves.....	163
Figure 65. Daylighting Shelf Diagram.....	163
Figure 66. Window and Outside Shelf as Adjacent Perpendicular Rectangles.....	165
Window Calculation Module.....	166
Optical Properties of Glazing	167
Table 24. Variables in Window Calculations	167
Glass Layer Properties.....	169
Glass Optical Properties Conversion	169
Simple Window Model.....	170
Glazing System Properties.....	173

TABLE OF CONTENTS

Figure 67. Schematic of transmission, reflection and absorption of solar radiation within a multi-layer glazing system.	174
Table 25: Solar spectral irradiance function.	175
Table 26: Photopic response function.	175
Calculation of Angular Properties.....	176
Table 27: Polynomial coefficients used to determine angular properties of coated glass.	178
Calculation of Hemispherical Values.....	179
Optical Properties of Window Shading Devices.....	179
Figure 68. Variations of Window Properties with the Temperature of the Thermochromic Glazing Layer	182
Figure 69. Configurations of Thermochromic Windows.....	183
Table 28. Slat Optical Properties.....	184
Figure 70. (a) Side view of a cell formed by adjacent slats showing how the cell is divided into segments, s_i , for the calculation of direct solar transmittance; (b) side view of a cell showing case where some of the direct solar passes between adjacent slats without touching either of them. In this figure ϕ_s is the profile angle and ϕ_b is the slat angle.....	185
Figure 72. Slat cell showing arrangement of segments and location of source for calculation of diffuse-to-diffuse optical properties.....	189
Figure 73. Side view of horizontal slats in a vertical blind showing geometry for calculating blind transmission, reflection and absorption properties for sky and ground diffuse radiation.	191
Figure 74. Side view of slats showing geometry for calculation of slat edge correction factor for incident direct radiation.....	193
Table 29. Comparison of blind optical properties calculated with the EnergyPlus and ISO 15099 methods. EnergyPlus values that differ by more than 12% from ISO 15099 values are shown in bold italics.	194
Figure 75. Screen model rendering of intersecting orthogonal crossed cylinders	200
Figure 76. Physical screen material properties	201
Figure 77. Schematic of a vertical window screen facing due South	201
Figure 78. Sun Angles in Screen Calculations.	206

TABLE OF CONTENTS

Figure 79. Screen/Glass System Transmittance Equation Schematic.....	209
Window Heat Balance Calculation.....	212
Table 30. Fortran Variables used in Window Heat Balance Calculations.....	212
The Glazing Heat Balance Equations.....	213
Figure 80. Glazing system with two glass layers showing variables used in heat balance equations.....	214
Room-Side Convection.....	215
Solving the Glazing Heat Balance Equations.....	216
Edge-Of-Glass Effects.....	217
Table 31. Fortran Variables used in Edge of Glass calculations.....	217
Figure 81: Different types of glass regions.....	218
Apportioning of Absorbed Short-Wave Radiation in Shading Device Layers.....	220
Window Frame and Divider Calculation.....	220
Table 32. Fortran Variables used in Window/Frame and Divider calculations.....	220
Figure 82. Cross section through a window showing frame and divider (exaggerated horizontally).....	223
Beam Solar Reflection from Window Reveal Surfaces.....	227
Figure 83. Example of shadowing of reveal surfaces by other reveal surfaces.....	227
Figure 84. Expression for area of shaded regions for different shadow patterns: (a) window without frame, (b) window with frame.....	229
Figure 85. Vertical section through a vertical window with outside and inside reveal showing calculation of the shadows cast by the top reveal onto the inside sill and by the frame onto the inside sill.....	231
Shading Device Thermal Model.....	235
Heat Balance Equations for Shading Device and Adjacent Glass.....	235
Figure 86. Glazing system with two glass layers and an interior shading layer showing variables used in heat balance equations.....	236

TABLE OF CONTENTS

Solving for Gap Airflow and Temperature	237
Figure 87. Vertical section (a) and perspective view (b) of glass layer and interior shading layer showing variables used in the gap airflow analysis. The opening areas A_{bot} , A_{top} , A_l , A_r and A_h are shown schematically.	238
Heat Balance Equations for Between-Glass Shading Device	243
Figure 90. Glazing system with two glass layers and a between-glass shading device showing variables used in the heat balance equations.	243
Airflow Windows	247
Figure 91. Gap airflow configurations for airflow windows. From “Active facades,” Version no. 1, Belgian Building Research Institute, June 2002.	248
Figure 92. Glazing system with forced airflow between two glass layers showing variables used in the heat balance equations.	249
Figure 93. Airflow window with between-glass shading device showing variables used in the heat balance equations.	251
References	251
Air Heat Balance Manager / Processes	253
Convection from Surfaces	253
Convection from Internal Sources	253
Infiltration/Ventilation	253
Infiltration Design Flow Rate	253
Infiltration by Effective Leakage Area	254
Infiltration by Flow Coefficient	254
Ventilation	255
Air Exchange	255
Calculation of Zone Air Temperature	256
Building System Simulation System Manager / Processes	257
Air Loops	257
Definition of Air Loop	257

TABLE OF CONTENTS

Simulation Method	257
Component Models	257
Iteration Scheme	258
Determination of Air Mass Flow Rates	258
Air Loop Simulation	258
Primary Air System Simulation.....	259
Table 33. ManageAirLoop Code.....	260
Input data	260
Initialization Calculations	260
Central air system simulation	262
Outdoor Air System.....	263
Simulation	263
Outdoor Air Mixer	263
Simulation	264
Zone Equipment Simulation	264
Input data	264
Initialization Calculations	265
Simulation	265
Air Path Components	266
Overview	266
Zone Supply Air Path	266
Zone Splitter	267
Zone Supply Plenum.....	267
Zone Return Air Path	267
Zone Mixer	267
Zone Return Plenum	268

TABLE OF CONTENTS

Plant Load Profile.....	268
Calculation Model.....	268
Plant/Condenser Loops.....	269
Integration of System and Plant.....	269
Current Primary System Modeling Methodology.....	269
Figure 94. Connections between the Main HVAC Simulation Loops and Sub-Loops.....	270
Figure 95. Branch Layout for Individual HVAC Sub-Loops.....	271
Plant Flow Resolver.....	271
Figure 96. Plant/Condenser Supply Side Solution Scheme.....	273
Figure 97. Demand and Supply Side Loops.....	276
Summary of Load Distribution Schemes.....	278
Figure 98. Load Distribution Scheme.....	278
Summary of Plant Loop Demand Calculation Schemes.....	279
Plant and Condenser Equipment Operation Schemes.....	280
Plant Operation Schemes.....	280
Condenser Operation Schemes.....	281
Primary-Secondary Loop Systems.....	282
Figure 99. Example of a Primary-Secondary Nested Loop Simulation.....	282
Figure 100. Schematic of a Two-Way Common Pipe used in Primary- Secondary System.....	286
Figure 101. Two-Way Common Pipe Interface in Detail.....	287
Heat Recovery Loop Systems.....	287
Figure 102. Example of a Heat Recovery Loop Simulation.....	288
Steam Systems and Component Models.....	288
Steam Loop Assumptions.....	289
Figure 103. Schematic of Temperature – Entropy Diagram for Steam loop.....	290

TABLE OF CONTENTS

Steam To Air Heat Exchanger	293
Condensate Pump	299
Figure 105. Schematic of Condensate Pump in Steam Loop.....	301
Steam Pipe.....	301
Loop and Equipment Sizing.....	303
Sizing Manager	303
Zone Design Loads and Air Flow Rates.....	304
Overview	304
Zone Design Data Arrays	304
Table 34. Zone Sizing Data	304
Zone Design Load Calculation	306
Updating and Adjusting the Zone Results	307
System Design Loads and Air Flow Rates.....	308
Overview	308
System Design Data Arrays	309
Table 35. System Sizing Data	309
System Design Flow Rate and Load Summation and Adjustment.....	310
Plant Loop Sizing	316
Introduction	316
Hot and Chilled Water Loop Sizing	316
Condenser Loop Sizing.....	317
Component Sizing.....	317
Introduction	317
Fan Sizing	317
Coil:Cooling:Water	318
Coil:Cooling:Water:DetailedGeometry Sizing	321

TABLE OF CONTENTS

Coil:Heating:Water Sizing	323
Sizing of Gas and Electric Heating Coils.....	324
DX Coil Sizing	325
DX MultiSpeed Coil Sizing	328
Pump Sizing	328
Electric Chiller Sizing	328
Boiler Sizing	329
Cooling Tower Sizing	329
Fan Coil Unit Sizing.....	331
Window Air Conditioner Sizing.....	332
Packaged Terminal Heat Pump Sizing	332
MultiSpeed Heat Pump Sizing	333
Single Duct Terminal Units.....	334
Indirect Evaporative Cooler Sizing	334
References	335
Demand Limiting.....	336
Algorithm	336
Table 36. Demand Manager Types and Resimulation.	337
Alternative Modeling Processes	338
RoomAir Models.....	338
Table 37. Summary of room air models available in EnergyPlus	338
User Defined RoomAir Temperatures.....	339
One-Node Displacement Ventilation RoomAir Model	341
Figure 106. Height versus temperature schematic for Mundt model.....	343
Three-Node Displacement Ventilation RoomAir Model.....	344
Under-Floor Air Distribution Interior Zone Model.....	353

TABLE OF CONTENTS

Figure 107. Data comparisons in the non-dimensional (a) regular $\Gamma-\phi$ plot.....	356
Figure 108. (b) log-log $\Gamma-\phi$ plot.}	356
Figure 109. Non-dimensional interface height of small-scale experiments.....	357
Under-Floor Air Distribtuion Exterior Zone Model	362
Cross Ventilation Room Air Model	369
Figure 110. Left: basic compartment geometry considered in the model. Right: top view of one half of the velocity field (at height H/2), result of a CFD simulation using the geometry on the left (taking advantage of the system symmetry to simulate only one half of the room volume).....	370
Figure 111. Top View of the possible airflow patterns in cross-ventilation.	371
Figure 112. Differences between flat plate heat transfer and cross ventilation flow.....	372
Figure 113. Average velocity in the recirculation region.....	375
Figure 114. Average velocity in the main jet region.....	376
Figure 115. Maximum flow rate in the recirculation region.	377
Figure 116. Results of the correlation for CV induced airflow velocity near the room surfaces.....	379
Figure 117. Top view of the flow structure in case R.....	379
Figure 118. Schematic View - Room airflow dominated by jet	384
Figure 119. Schematic - Room airflow -- a combination of jet and recirculation flow.....	384
Table 38. Nomenclature for Cross-Ventilation RoomAir Model.....	385
AirflowNetwork Model	387
Overview	387
Model Description	387
Pressure and Airflow Calculations	388
Figure 120. The general problem of gravitational flow through a vertical opening.....	390

TABLE OF CONTENTS

Figure 121. Schematic drawing of a horizontally-pivoted window	392
Figure 122. Schematic of large opening and associated three flow patterns.....	393
Node Temperature Calculations	396
Node Humidity Ratio Calculations.....	398
Sensible and Latent Load Calculations	399
Impacts of Supply Air Fan Control on Load: Cycling vs. Continuous	401
Integration of the AirflowNetwork Model	402
Model Output.....	402
References	402
Simulation Models – Encyclopedic Reference.....	404
Air System Distribution Terminals	404
Constant Volume Single Duct Uncontrolled Air Terminal.....	404
Constant Volume Single Duct Reheat Air Terminal	404
Figure 123. Schematic of AirTerminal:SingleDuct:ConstantVolume:Reheat Unit	405
Variable Air Volume Single Duct Reheat and No Reheat Air Terminals	405
Figure 124. Schematic of AirTerminal:SingleDuct:VAV:NoReheat Unit	405
Figure 125. Schematic of AirTerminal:SingleDuct:VAV:Reheat Unit.....	405
Variable Air Volume Heating and Cooling Single Duct Reheat and NoReheat Air Terminal.....	406
Figure 126. Schematic of AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat Unit.....	406
Figure 127. Schematic of AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat Unit	407
Constant Volume Single Duct Four Pipe Induction Air Terminal.....	408
Fan Powered Induction Series and Parallel Single Duct Reheat Air Terminal	410
Variable Air Volume Fan Powered Single Duct Air Terminal	413

TABLE OF CONTENTS

Cooled Beam Unit (AirTerminal:SingleDuct:ConstantVolume:CooledBeam).....	416
Constant Volume Dual Duct Air Terminal	418
Variable Air Volume Dual Duct Air Terminal	419
Boilers	421
Simple Hot Water Boiler.....	421
Steam Boiler.....	422
Figure 128. Schematic of Steam Boiler in the Steam loop	423
Figure 129. Schematic of Steam Boiler Operation	424
Table 39. Steam Loop Nomenclature.....	427
References.....	429
Chillers	429
Absorption Chiller.....	429
Indirect Absorption Chiller	433
Figure 130. Schematic Diagram of a Single-Stage Absorption Chiller	434
Combustion Turbine Chiller.....	439
Constant COP Chiller.....	441
Hot Water Heat Recovery from Chillers	441
Figure 131. Diagram of Chiller:Electric with Heat Recovery.....	442
Electric Chiller Model Based on Condenser Entering Temperature.....	443
Electric Chiller Model Based on Condenser Leaving Temperature.....	448
Engine Driven Chiller	454
Ice Thermal Storage.....	456
Simple Ice Storage Model	456
Detailed Ice Storage Model.....	458
Coils	461
Chilled-Water-Based Air Cooling Coil	461

TABLE OF CONTENTS

Figure 132. Simplified Schematic of Cooling/Dehumidifying Coil.....	462
Table 40. Design Inputs (User Inputs).....	462
Table 41. Operating Conditions (From Nodes -- not user inputs).....	462
Table 42. UA Descriptions of Model.....	463
Figure 133. Air Outlet Temperature Vs Air Mass Flow Rate	472
Figure 134. Sensible Load variations Vs Air mass Flow Rate.....	472
Figure 135. Total and Sensible Load variations Vs Air Mass Flow Rate.....	473
Figure 136. Surface Area Fraction Wet Vs Air Mass Flow Rate.....	473
Chilled-Water-Based Detailed Geometry Air Cooling Coil	474
Figure 137. Simplified Schematic of Cooling/Dehumidifying Coil.....	475
Table 43. Coil Geometry and Flow Variables for Coils.....	475
Hot-Water-Based Air Heating Coil	481
Single-Speed Electric DX Air Cooling Coil	482
Figure 138. Transient Sensible and Latent Capacity of a Cooling Coil Over an Operating Cycle	492
Figure 139. Field Data Showing the Net Impact of Part-Load Operation on Sensible Heat Ratio.....	492
Figure 140. Concepts of Moisture Buildup and Evaporation	493
Multi-Speed Electric DX Air Cooling Coil	497
Two-Speed Electric DX Air Cooling Coil	503
Electric Air Heating Coil	504
Gas Air Heating Coil.....	505
Single-Speed Electric Heat Pump DX Air Heating Coil.....	506
Multi-Speed Electric Heat Pump DX Air Heating Coil	515
Desuperheater-Recovery-Based Air Heating Coil.....	520
Figure 141. Desuperheater Heating Coil Used as a Reheat Coil with AirLoopHVAC:UnitaryCoolOnly	521
Desuperheater-Recovery-Based Water Heating Coil	525

TABLE OF CONTENTS

Figure 142. Schematic of Desuperheater Water Heating Coil.....	527
Heat Exchanger Assisted Air Cooling Coil Systems	531
Figure 143. Schematic of a Heat Exchanger Assisted Cooling Coil.....	531
Figure 144. Psychrometric Process for Heat Exchanger Assisted Cooling Coil (Sensible HX Only).....	532
Figure 145. Psychrometric Process for Heat Exchanger Assisted Cooling Coil (Sensible+Latent HX).....	533
Figure 146. Schematic of a heat exchanger assisted DX cooling coil with optional humidistat	534
Figure 147. Schematic of Heat Exchanger Assisted DX Coil with Humidistat and Setpoint Manager	535
Single-Speed Electric Heat Pump DX Water Heating Coil.....	536
Figure 148. Schematic of a Heat Pump Water Heater using Optional Mixer/Splitter Nodes	536
Figure 149. Schematic of the Heat Pump Water Heater DX Coil	537
Water Source Electric DX Air Cooling Coil.....	544
Water Source Electric Heat Pump DX Air Heating Coil.....	544
Steam-Based Air Heating Coil	544
HVAC Controllers.....	545
Control Valve for Water-Based Air System Coils	545
Figure 150. Controller:WaterCoil used with Central Chilled Water Coil	545
Figure 151. Two Setpoint managers used in Controller:WaterCoil	546
Outdoor Air Damper Controller for Air Systems	547
Figure 152. Economizer Limit Controls	549
Outdoor Air Damper Controller for Zone Energy Recovery Ventilator	552
Figure 153. Schematic of the ZoneHVAC:EnergyRecoveryVentilator Compound Object.....	553
Baseboard Heaters	554
Hot Water Baseboard Heater with Only Convection	554

TABLE OF CONTENTS

Electric Baseboard Heater with Only Convection	556
Hot Water Baseboard Heater with Radiation and Convection	557
Cooling Towers	559
One and Two Speed Cooling Towers	559
Figure 154. Cooling Tower Schematic	561
Variable Speed Cooling Towers.....	564
Table 44. Approach Temperature Correlation Coefficients	566
Table 45. Minimum and Maximum Limits for Approach Temperature Correlation Variables	568
Cooling Tower Makeup Water Usage	573
One and Two Speed Fluidcoolers.....	575
Demand Controlled Ventilation	580
Figure 155. Controller:MechanicalVentilation Zone Air Distribution Effectiveness Typical Values (Source: ASHRAE Standard 62.1-2007)	582
Figure 156. Demand Control Ventilation -- Air Flow Paths.....	584
Evaporative Coolers.....	586
Direct Evaporative Cooler	586
Figure 157. Direct Stage Evaporative Cooler.....	587
Figure 158. Psychrometric Chart -- Constant Enthalpy	587
Figure 159. Graph of Saturation Efficiency.....	589
Dry Coil Indirect Evaporative Cooler	589
Figure 160. Evaporative Cooler -- Indirect Dry Coil.....	590
Figure 161. Secondary Air Process -- Indirect Dry Coil Evap Cooler	590
Wet Coil Indirect Evaporative Cooler	591
Figure 162. Wet Coil Indirect Evaporative Cooler	591
Figure 163. Secondary Air Process -- Indirect Wet Coil Evaporative Cooler.....	592
Two Stage Direct/Indirect Evaporative Cooler	593

TABLE OF CONTENTS

Figure 164. Two Stage Evaporative Cooler.....	594
Figure 165. Thermodynamic Process for Supply Air Through Two Stage Evaporative Cooler.....	594
Indirect Evaporative Cooler Special Research Model.....	594
Figure 166. Research Special Indirect Evaporative Cooler.....	595
Figure 167. Research Special Indirect Evaporative Cooler Using Relief Air.....	598
Direct Evaporative Cooler Special Research Model.....	598
Air System Fans.....	600
Overview.....	600
Model.....	600
References.....	603
Air System Compound Component Groups.....	604
Forced-Air Furnace and Central Air Conditioning.....	604
Figure 168. Schematic of the EnergyPlus Furnace (Blow Through Configuration).....	604
Figure 169. Schematic for Blow Through Furnace with High Humidity Control.....	610
Figure 170. Reheat Coil Load when Predicted Zone Temperature is Above Heating Setpoint.....	611
Figure 171. Reheat Coil Load when Predicted Zone Temperature is Below Heating Setpoint.....	611
Unitary Systems.....	611
Unitary System with Changeover-Bypass-Variable Air Volume.....	612
Figure 172. Schematic of a Changeover Bypass VAV Unitary System with Draw Through Fan.....	612
Figure 2.....	613
Figure 173. Schematic of Changeover Bypass VAV Unitary System with Blow Through Fan.....	613
Unitary Single Speed Air-To-Air Heat Pump.....	620

TABLE OF CONTENTS

Figure 174. Schematic of a Blow Through Air-to-Air Heat Pump	621
Unitary Multi-Speed Air-To-Air Heat Pump	625
Figure 175. Schematic of a Multispeed Air-to-Air Heat Pump (Blow-through Configuration).....	626
DX Cooling Package	634
Figure 176. NEEDS NEW FIGURE Schematic of Packaged DX Cooling Subsystem in EnergyPlus DX System in Air Loop for a Blow-Thru Application.....	635
Desiccant Dehumidifier Package	636
Figure 177. Schematic of a Desiccant Dehumidifier with Draw Through Regeneration Fan Placement.....	637
Figure 178. Schematic of a Desiccant Dehumidifier in Blow Through Regeneration Fan Placement.....	637
Unitary Single Speed Water-To-Air Heat Pump.....	640
Figure 179. NEEDS NEW FIGURE Source Side and Load Side Configuration of a BlowThru WaterToAir Heat Pump.....	641
Figure 180. Information Flow Chart for Water-to-Air Heat Pump Equation Fit Model (Tang 2005).....	644
Water To Water Heat Pumps	645
Equation Fit Water To Water Heat Pump Model.....	645
Figure 181. Information Flow Chart for Water-To-Water Heat Pump Equation Fit (Tang 2005).....	647
Parameter Estimation Water-To-Water Heat Pump Model	647
Figure 182. Schematic indicator diagram for a reciprocating Compressor(Jin 2002).....	648
Figure 183. Information Flowchart for Water-To-Water Heat Pump Parameter Estimation Mmodel implementation (Jin 2002).....	650
Figure 184. Flow diagram of EnergyPlus Water to Water HeatPump implementation	652
Heat Exchangers.....	653
Air System Air-To-Air Sensible and Latent Effectiveness Heat Exchanger.....	653

TABLE OF CONTENTS

Figure 185. Schematic of the Sensible and Latent Air-to-Air Heat Exchanger	653
Table 46. Operating Conditions for Defining Heat Exchanger Performance.....	654
Figure 186. Air to Air Heat Exchanger with Supply Air Temperature Control.....	665
Air System Air-To-Air Flat Plate Heat Exchanger.....	665
Air System Air-To-Air Balanced Flow Desiccant Heat Exchanger	666
Figure 187. Schematic of the Balanced Flow Desiccant Heat Exchanger	667
Plant Loop Deep-Ground-To-Water Vertical U-Tube Field Heat Exchanger	676
Figure 188. Short Time Step g Function Curve as an Extension of Long Time Step g Function Curves for Different Configuration of Boreholes (Eskilson 1987, Yavuzturk 1999).....	677
Figure 189. Variable Timestep Ground Loop Heat Exchanger Model Schematic Explaining the g Function Estimation.....	679
Figure 190. Schematic of Variable Time Step Model g Function Calculation.....	680
Figure 191. Schematic Showing the Calculation of Hourly Load from the Sub Houly Loads.	681
Plant Loop Pond-To-Water Heat Exchanger.....	685
Figure 192. Heat Transfer Mechanisms in a Pond (Chiasson 1999).....	685
Figure 193. Pond Model Component Configuration (Chiasson 1999).....	687
Plant Loop Surface-Ground-To-Water Heat Exchanger	690
Plant Loop Water-To-Water Free Cooling Hydronic Heat Exchanger.....	692
Plant Loop Water Side Economizer	693
Figure 194. Typical Integrated Water side Economizer Installation	694
Figure 195. Schematic of Integrated Water side Economizer	694
Figure 196. Typical Non-Integrated Water side Economizer Installation.....	695
Figure 197. Schematic of Non-Integrated Water side Economizer	696

TABLE OF CONTENTS

References	697
Air System Humidifiers	697
Overview	697
Electric Steam Humidifier	697
Zone Internal Gains	700
Sources and Types of Gains	700
Heat Gain from Lights	701
Heat Gain from People	701
Figure 198. Sensible Heat Gain from People Correlation	702
Heat Gain from Baseboard Heat	702
Figure 199. Control of Outdoor Temperature Controlled Baseboard Heat	703
Distribution of Radiant Gains	703
References	704
Pipes	704
Heat Transfer Pipes (Objects: Pipe:Indoor & Pipe:Outdoor)	704
Figure 200. Control Volume drawn around node i	705
Underground Pipe (Object: Pipe:Underground)	707
Figure 201. Pipe:Underground Outer Finite Difference Grid	708
Pumps	709
Variable Speed Pump	710
Constant Speed Pump	711
Pump Heat Addition to the Loop	711
Headered Pumps	712
Condensate Pumps	712
Ideal Loads Air System	713
Overview	713

TABLE OF CONTENTS

Model	713
References	714
District Cooling	715
District Heating	715
Radiant System Models	715
Low Temperature Radiant System Model	715
Figure 202. Single Layered Building Element	716
Figure 203. Multilayered Building Element	717
Figure 204. Two Node State Space Example	721
Figure 205. Two Layer Example for Deriving the Laplace Transform Extension to Include Sources and Sinks	722
Figure 206. Two Node State Space Example with a Heat Source	726
Figure 207. Variable Flow Low Temperature Radiant System Controls	731
Figure 208. Variable Temperature Low Temperature Radiant System Controls	731
Figure 209. Variable Temperature Low Temperature Radiant System Component Details	732
Figure 210. Resolution of Radiant System Response at Varying Time Steps	734
High Temperature Radiant Heater Model	740
Figure 211. Input Description for High Temperature Radiant Heaters	741
Refrigeration Equipment	743
Overview	743
Figure 212. Typical Refrigerated Case and Compressor Rack Equipment Schematic	744
Figure 213. Typical Refrigerated Case and Detailed Refrigeration System Equipment Schematic	744
Refrigeration Compressor Racks	744
Figure 214. Example Of Condenser Heat Recovery To Water Storage Tank	745

TABLE OF CONTENTS

Refrigerated Cases	751
Table 47. Representative Defrost Energy Correction Curve Coefficients for Case Temperature Method.....	757
Table 48. Representative Latent Case Credit Curve Coefficients for Case Temperature Method.....	761
Figure 215. Return Air Factor Versus Under Case HVAC Return Air Fraction.....	762
Detailed Refrigeration Systems.....	763
Figure 216. Comparison of the condensing temperature predicted by four-factor equation to manufacturer's data.....	767
Figure 217. State Points and Energy Flows for Detailed Refrigeration System.....	770
References	772
Setpoint Managers	773
Overview	773
Scheduled	774
Outdoor Air Reset	774
Single Zone Reheat Heating and Cooling	775
Single Zone Heating Only	775
Single Zone Cooling Only	776
Single Zone Minimum Humidity	776
Single Zone Maximum Humidity	776
Mixed Air	778
Outdoor Air Pretreat	778
Warmest Zone Supply Air Reset.....	778
Coldest Zone Supply Air Reset	779
Return Air Bypass Flow.....	779
Warmest Temp Flow	779
Solar Collectors.....	780

TABLE OF CONTENTS

Flat-Plate Solar Collectors	780
Photovoltaic Thermal Flat-Plate Solar Collectors.....	784
Unglazed Transpired Solar Collectors	786
Figure 218. Transpired Collector Heat Balance	790
Figure 219. Transpired Collector Plenum Air Heat Balance.....	794
Table 49. Terrain-Dependent Coefficients (ASHRAE 2001).	797
System Availability Managers	800
Overview	800
Scheduled	800
Scheduled On	800
Scheduled Off	801
Night Cycle.....	801
Night Ventilation	801
Differential Thermostat.....	802
High Temperature Turn Off	802
High Temperature Turn On	803
Low Temperature Turn Off.....	803
Low Temperature Turn On.....	803
Hybrid Ventilation Control	803
Occupant Thermal Comfort.....	806
Background on Thermal Comfort Models.....	806
Table 50. General Nomenclature list for Thermal Comfort Models	806
Mathematical Models for Predicting Thermal Comfort.....	807
Table 51. Seven point Thermal Sensation Scale	808
Table 52. Nine point Thermal Sensation Scale	808
Fanger Comfort Model	808

TABLE OF CONTENTS

Table 53. Nomenclature list for Fanger model	809
Pierce Two-Node Model.....	812
Table 54. Nomenclature list for Pierce Two-Node model	812
KSU Two-Node Model	818
Table 55. Nomenclature list for KSU Two-Node model.....	818
Mean Radiant Temperature Calculation	823
Table 56. Nomenclature and variable list for MRT calculation	823
References.....	824
Trombe Walls.....	825
Figure 220. Building with Trombe Wall.....	826
Passive Trombe Wall	826
Figure 221. Passive Trombe Wall Winter	827
Figure 222. Passive Trombe Wall Summer.....	828
Active Trombe Wall.....	828
Figure 223. Active Trombe Wall Winter.....	830
Water Thermal Tanks (includes Water Heaters).....	830
Mixed Water Thermal Tank.....	830
Figure 224. Water Heater Cycle Control Algorithm	834
Heat Pump Water Heater.....	835
Figure 225. Schematic of a heat pump water heater using optional mixer/splitter nodes	837
Figure 226. Schematic of a Heat Pump Water Heater with Inlet Air from a Zone	837
Figure 227. Schematic of a Heat Pump Water Heater with Inlet Air from Outdoors.....	838
Stratified Water Thermal Tank	841
Water Heating Sizing	844
Table 57. Residential HUD-FHA Minimum	845

TABLE OF CONTENTS

Water Systems.....	847
Water Mains Temperatures.....	847
Water Use Equipment and Connections	848
Unconnected Water Use Equipment.....	848
Figure 228. Hot and Cold Water Mixing	849
Zone Heat Gain from Water Use Equipment	850
Figure 229. Zone Heat Gain from Water Use Equipment.....	851
Connected Water Use Equipment.....	853
Figure 230. Water Use Connections Subsystem.....	854
Water Use Equipment Calculations	855
Drainwater Heat Recovery	856
Figure 231. Water Use Connections Subsystem with Drainwater Heat Recovery	858
Zone Controls.....	860
Thermostatic Zone Control.....	860
Zone Thermostats	861
Operative Temperature Control	863
Table 58. Operative Temperature - Radiative Fraction vs Air Velocity.....	863
Humidistat	864
Thermal Comfort Zone Control	865
Zone Equipment and Zone Forced Air Units.....	867
Air Distribution Terminal Unit	867
Simple Duct Leakage Model	867
Fan Coil Unit	870
Window Air Conditioner.....	872
Packaged Terminal Air Conditioner	874

TABLE OF CONTENTS

Figure 232. Schematic of a Packaged Terminal Air Conditioner with Draw Through Fan Placement.....	875
Packaged Terminal Heat Pump	881
Figure 233. Schematic of a Packaged Terminal Heat Pump (Draw Through Fan Placement).....	881
Energy Recovery Ventilator	887
Unit Heater	891
Unit Ventilator.....	891
Ventilated Slab	894
Figure 235. Basic System for the Ventilated Slab Module	894
Figure 236. Model with Air Delivered to Slab. (Slab Only Mode)	895
Figure 237. Zone Supply Model using Ventilated Slab (Slab and Zone mode)	895
Figure 238. Multiple Slabs model with Several Zones (Series Slabs Mode)	895
Earthtube.....	896
Table 59. Nomenclature for Earthtube Model.....	900
Thermal Chimney Model	902
Figure 239. Basic Composition of Thermal Chimney	903
Table 60. Nomenclature of Thermal Chimney Model	905
CoolTower.....	906
Figure 240. Cooltower Configuration. Temperatures in °C.....	907
Generation	910
Electric Load Center Distribution Manager	910
Electric Load Center Generators.....	911
Generators	911
Internal Combustion Engine	911
Turbine Generator.....	912

TABLE OF CONTENTS

Microturbine Generator	913
Micro-Cogenerator	923
Fuel Cell Cogenerator	926
Custom Fuel Supply for Generators.....	929
Photovoltaic Arrays	931
Simple Model	931
Table 61. Nomenclature for Simple Photovoltaic model.....	932
Equivalent One-Diode Model	932
Table 62. General Nomenclature for the PV model.....	933
Figure 241. Equivalent Circuit in the Four Parameter PV Model.....	934
Sandia Photovoltaic Performance Model	938
Table 63. Nomenclature for Sandia PV model	938
Economics Calculations	943
Component Costs	943
Line Item Costs	943
Adjustments	946
Comparisons	948
Tariff Computation.....	948
Conceptual Framework – Variables and Hierarchy.....	948
Default Order of Computation	950
Computation Steps.....	951
Special Modules/Reporting.....	952
Environmental Impacts.....	952
Types of Pollutants.....	952
Figure 242. Example Annual Atmospheric Pollutants and Water Consumption	953

TABLE OF CONTENTS

Figure 243. Example Annual Total Carbon Equivalent for Major Greenhouse Gases	953
Carbon Equivalent.....	953
Table 64. Carbon Equivalents (IPCC 2001)	954
Fossil Fuel Emissions Factors	954
Table 65. Emission Factors for Natural Gas.....	954
Table 66. Emission Factors for Residual Fuel Oil (No. 4 and No. 6 Fuel Oil).....	955
Table 67. Emission Factors for Distillates (No. 1 and No. 2 Fuel Oil)	955
Table 68. Emission Factors for Residential Oil Furnace	956
Table 69. Emission Factors for LPG (butane and propane).....	956
Table 70. Emission Factors for Gasoline and Diesel.....	957
Table 71. Emission Factors for Coal	957
Off-Site Electricity Generation Emissions	958
Table 72. United States National Average Emission Factors for Electricity Generation	958
Table 73. U. S. State Average Greenhouse Gas Emission Factors for Electricity Generation, in g/MJ.....	959
Table 74. U. S. State Average Criteria Pollutant Emission Factors for Electricity Generation, in g/MJ.....	960
Table 75. United States National Average Water Consumption Factors ^a	961
Table 76. U.S. State Average Water Consumption Factors for Electricity Generation ^a	962
Other Energy-Related Pollutants and Sources of Other Information	963
References.....	963

Overview

Document Overview

This document is organized to give you the best possible look into the EnergyPlus calculations. First, the concepts of modeling in EnergyPlus are presented. These include descriptions of the zone heat balance process, air loop/plant loop processes as well as other important processes for the building simulation.

Discussions during the modeling process may reference specific “object names” as found in the Input/Output Reference document.

The remainder of the document focuses on individual models.

General Modeling Overview

The EnergyPlus program is a collection of many program modules that work together to calculate the energy required for heating and cooling a building using a variety of systems and energy sources. It does this by simulating the building and associated energy systems when they are exposed to different environmental and operating conditions. The core of the simulation is a model of the building that is based on fundamental heat balance principles. Since it is relatively meaningless to state: “based on fundamental heat balance principles”, the model will be described in greater detail in later sections of this document in concert with the FORTRAN code which is used to describe the model. It turns out that the model itself is relatively simple compared with the data organization and control that is needed to simulate the great many combinations of system types, primary energy plant arrangements, schedules, and environments. The next section shows this overall organization in schematic form. Later sections will expand on the details within the blocks of the schematic.

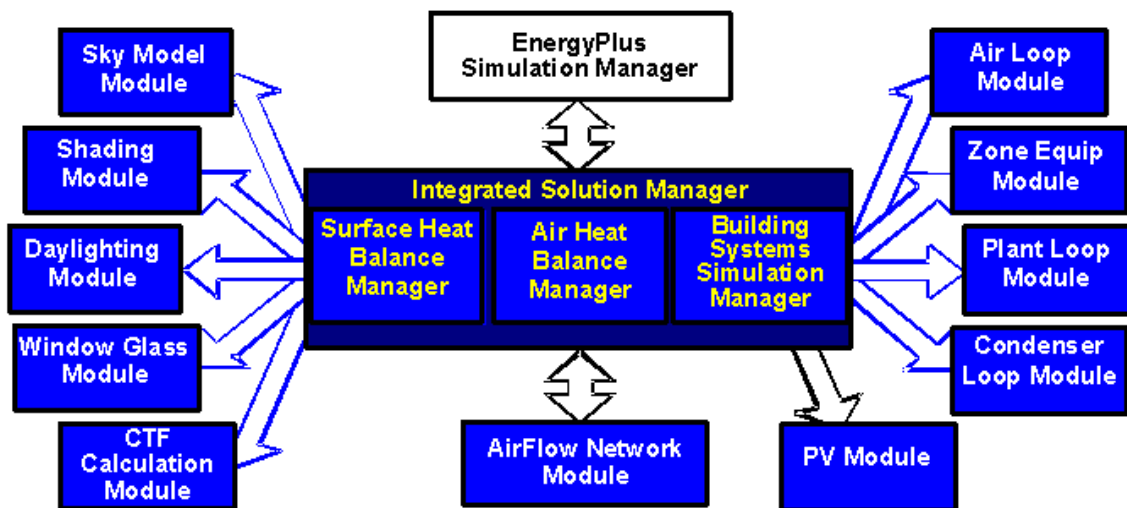


Figure 1. EnergyPlus Program Schematic

Simulation Manager

The simulation manager of EnergyPlus is contained in a single module. The main subroutine is shown below. Flow within the entire program is managed using a series of flags. These paired flags, in order (from the highest to the lowest) are:

Table 1. Simulation Flags

Begin SimulationFlag	End SimulationFlag
BeginEnvironmentFlag	EndEnvironmentFlag(one to many days)
BeginDayFlag	EndDayFlag
BeginHourFlag	EndHourFlag
BeginTimeStepFlag	EndTimeStepFlag

There is also a **WarmupFlag** to signal that the program is in warmup state. The operation of these flags can be seen in the following subroutine. The advantage of using the flag system is that any subroutine throughout the code can determine the exact state of the simulation by checking the status of the flags.

```

SUBROUTINE ManageSimulation      ! Main driver routine for this module
BeginSimFlag = .TRUE.
EndSimFlag = .FALSE.
CALL OpenOutputFiles
CALL GetProjectData
CALL GetEnvironmentInfo          ! Get the number and type of Environments
DO Envrn = 1, NumOfEnvrn        ! Begin environment loop ...
  BeginEnvrnFlag = .TRUE.
  EndEnvrnFlag = .FALSE.
  WarmupFlag = .TRUE.
  DayOfSim = 0
  DO WHILE ((DayOfSim.LT.NumOfDayInEnvrn).OR.(WarmupFlag)) ! Begin day loop ...
    DayOfSim = DayOfSim + 1
    BeginDayFlag = .TRUE.
    EndDayFlag = .FALSE.
    DO HourOfDay = 1, 24      ! Begin hour loop ...
      BeginHourFlag = .TRUE.
      EndHourFlag = .FALSE.
      DO TimeStep = 1, NumOfTimeStepInHour ! Begin time step (TINC) loop ...
        BeginTimeStepFlag = .TRUE.
        EndTimeStepFlag = .FALSE.
        ! Set the End_Flag variables to true if necessary. Note that each flag builds on
        ! the previous level. EndDayFlag cannot be .true. unless EndHourFlag is also .true., etc.
        ! Note that the EndEnvrnFlag and the EndSimFlag cannot be set during warmup.
        ! Note also that BeginTimeStepFlag, EndTimeStepFlag, and the
        ! SubTimeStepFlags can/will be set/reset in the HVAC Manager.
        IF ((TimeStep.EQ.NumOfTimeStepInHour)) THEN
          EndHourFlag = .TRUE.
          IF (HourOfDay.EQ.24) THEN
            EndDayFlag = .TRUE.
            IF ((.NOT.WarmupFlag).AND.(DayOfSim.EQ.NumOfDayInEnvrn)) THEN
              EndEnvrnFlag = .TRUE.
              IF (Envrn.EQ.NumOfEnvrn) THEN
                EndSimFlag = .TRUE.
              END IF
            END IF
          END IF
        END IF
      END DO
    END DO
    CALL ManageWeather
    CALL ManageHeatBalance
    BeginHourFlag = .FALSE.
    BeginDayFlag = .FALSE.
    BeginEnvrnFlag = .FALSE.
    BeginSimFlag = .FALSE.
  END DO
  ! ... End time step (TINC) loop.
END DO
! ... End hour loop.
END DO
! ... End day loop.
END DO
! ... End environment loop.
CALL CloseOutputFiles
RETURN
END SUBROUTINE ManageSimulation

```

Integrated Solution Manager

EnergyPlus is an integrated simulation. This means that all three of the major parts, building, system, and plant, must be solved simultaneously. In programs with sequential simulation, such as BLAST or DOE-2, the building zones, air handling systems, and central plant equipment are simulated sequentially with no feedback from one to the other. The sequential solution begins with a zone heat balance that updates the zone conditions and determines the heating/cooling loads at all time steps. This information is fed to the air handling simulation to determine the system response; but that response does not affect zone conditions. Similarly, the system information is passed to the plant simulation without feedback. This simulation technique works well when the system response is a well-defined function of the air temperature of the conditioned space. For a cooling situation, a typical supply and demand situation is shown schematically in the Figure 2. Here, the operating point is at the intersection of the supply and demand curves.

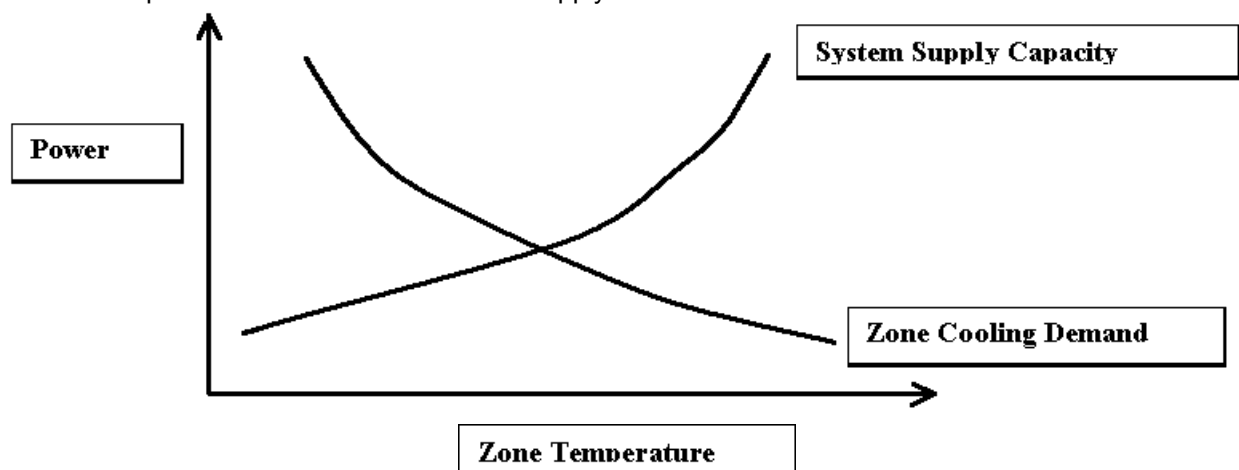


Figure 2. Sequential Simulation Supply/Demand Relationship.

However, in most situations the system capacity is dependent on outside conditions and/or other parameters of the conditioned space. The simple supply and demand situation above becomes a more complex relationship and the system curve is not fixed. The solution should move up and down the demand curve. This doesn't happen in sequential simulation methods and the lack of feedback from the system to the building can lead to nonphysical results. For example, if the system provides too much cooling to a conditioned space the excess is reported by the program as "overcooling". Other categories of unmatched loads exist and are similarly reported by the program. While this kind of reporting enables the affected system or plant components to be properly sized, the system designer would, in most cases, prefer to see the actual change in zone temperature. The same mismatches can occur between the system and plant simulations when they are simulated sequentially.

To obtain a simulation that is physically realistic, the elements have to be linked in a simultaneous solution scheme. The entire integrated program can be represented as a series of functional elements connected by fluid loops as shown in Figure "Schematic of Simultaneous Solution Scheme". In EnergyPlus all the elements are integrated and controlled by the Integrated Solution Manager. The loops are divided into supply and demand sides, and the solution scheme generally relies on successive substitution iteration to reconcile supply and demand using the Gauss-Seidell philosophy of continuous updating.

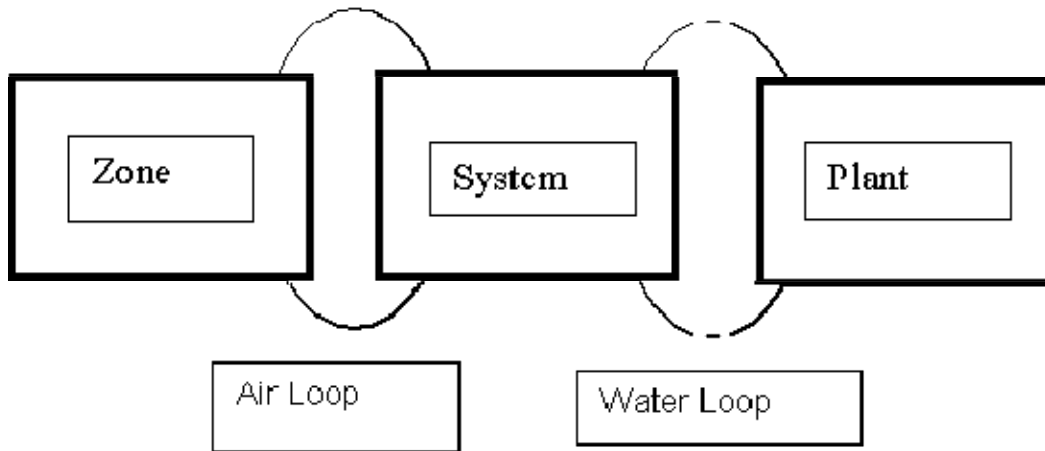


Figure 3. Schematic of Simultaneous Solution Scheme

In the sections which follow, the various individual functions of the integrated solution will be described.

Basis for the Zone and System Integration

The basis for the zone and system integration, Taylor.(1990, 1991), incorporates a shortened simulation time step, typically to between 0.1 and 0.25 hours, and uses a time-marching method having the zone conditions lagged by one time step. The error associated with this approach depends significantly on the time step. The smaller the step size the smaller the error, but the longer the computation time. To permit increasing the time step as much as possible while retaining stability, zone air capacity was also introduced into the heat balance. The resulting method is called "lagging with zone capacitance". Although requiring substantially more time to execute than sequential simulation methods, the improved realism of the simultaneous solution of loads, systems and plants simulation is desirable. This method was fully implemented in the program IBLAST (Integrated Building Loads Analysis and System Thermodynamics) (Taylor 1996) that was used as a basis for EnergyPlus.

The method of lagging with zone capacitance uses information from previous time steps to predict system response and update the zone temperature at the current time. In older sequential programs, one hour is used frequently as a time step because it is convenient for record keeping purposes and it keeps computation time reasonable. But dynamic processes in the zone air can occur on a much shorter time scale than one hour. The time constant, τ , for a zone is on the order of:

$$\tau \approx \frac{\rho V c_p}{|\dot{Q}_{load} + \dot{Q}_{sys}|} \quad (1)$$

where the numerator is the zone air heat capacitance and the denominator is the net rate of heat energy input. Clearly, the value of τ can vary because the zone load and system output change throughout the simulation. Therefore, a variable adaptive time step shorter than one hour is used for updating the system conditions. For stability reasons it was necessary to derive an equation for the zone temperature that included the unsteady zone capacitance term and to identify methods for determining the zone conditions and system response at successive time steps. The formulation of the solution scheme starts with a heat balance on the zone.

$$C_z \frac{dT_z}{dt} = \sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) + \dot{m}_{inf} C_p (T_\infty - T_z) + \dot{Q}_{sys} \quad (2)$$

where:

$$\sum_{i=1}^{N_{sl}} \dot{Q}_i = \text{sum of the convective internal loads}$$

$$\sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) = \text{convective heat transfer from the zone surfaces}$$

$$\dot{m}_{inf} C_p (T_\infty - T_z) = \text{heat transfer due to infiltration of outside air}$$

$$\sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) = \text{heat transfer due to interzone air mixing}$$

$$\dot{Q}_{sys} = \text{system output.}$$

$$C_z \frac{dT_z}{dt} = \text{energy stored in zone air}$$

If the air capacitance is neglected, the steady state system output must be:

$$-\dot{Q}_{sys} = \sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) + \dot{m}_{inf} C_p (T_\infty - T_z) \quad (3)$$

Air systems provide hot or cold air to the zones to meet heating or cooling loads. The system energy provided to the zone, \dot{Q}_{sys} , can thus be formulated from the difference between the supply air enthalpy and the enthalpy of the air leaving the zone as in Equation (4):

$$\dot{Q}_{sys} = \dot{m}_{sys} C_p (T_{sup} - T_z) \quad (4)$$

This equation assumes that the zone supply air mass flow rate is exactly equal to the sum of the air flow rates leaving the zone through the system return air plenum and being exhausted directly from the zone. Both air streams exit the zone at the zone mean air temperature. The result of substituting Equation (4) for \dot{Q}_{sys} in the heat balance Equation (2) is shown in Equation (5):

$$C_z \frac{dT_z}{dt} = \sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) + \dot{m}_{inf} C_p (T_\infty - T_z) + \dot{m}_{sys} C_p (T_{sup} - T_z) \quad (5)$$

The sum of zone loads and system output now equals the change in energy stored in the zone. Typically, the capacitance C_z would be that of the zone air only. However, thermal masses assumed to be in equilibrium with the zone air could be included in this term. In order to calculate the derivative term a finite difference approximation may be used, such as:

$$\frac{dT}{dt} = (\delta t)^{-1} (T_z^t - T_z^{t-\delta t}) + O(\delta t) \quad (6)$$

The use of numerical integration in a long time simulation is a cause for some concern due to the potential build-up of truncation error over many time steps. In this case, the finite difference approximation is of low order that further aggravates the problem. However, the cyclic nature of building energy simulations should cause truncation errors to cancel over each daily cycle so that no net accumulation of error occurs, even over many days of simulation (Walton, 1990). The Euler formula, Equation (6), was employed in Equation (5) to replace the derivative term. All the terms containing the zone mean air temperature were then grouped on the left hand side of the equation. Since the remaining terms are not known at the current time, they were lagged by one time step and collected on the right hand side. This manipulation resulted in Equation (7), the formula for updating the zone mean air temperature:

$$C_z \frac{T_z^t - T_z^{t-\delta t}}{dt} + T_z^t \left(\sum_{i=1}^{N_{surfaces}} h_i A_i + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p + \dot{m}_{inf} C_p + \dot{m}_{sys} C_p \right) = \sum_{i=1}^{N_{sl}} \dot{Q}_i^t + \dot{m}_{sys} C_p T_{supply}^t + \left(\sum_{i=1}^{N_{surfaces}} h_i A_i T_{si} + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p T_{zi} + \dot{m}_{inf} C_p T_{\infty} \right)^{t-\delta t} \quad (7)$$

One final rearrangement was to move the lagged temperature in the derivative approximation to the right side of the equation. The explicit appearance of the zone air temperature was thus eliminated from one side of the equation. An energy balance equation that includes the effects of zone capacitance was then obtained by dividing both sides by the coefficient of T_z :

$$T_z^t = \frac{\sum_{i=1}^{N_{sl}} \dot{Q}_i^t + \dot{m}_{sys} C_p T_{supply}^t + \left(C_z \frac{T_z^{t-\delta t}}{\delta t} + \sum_{i=1}^{N_{surfaces}} h_i A_i T_{si} + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p T_{zi} + \dot{m}_{inf} C_p T_{\infty} \right)^{t-\delta t}}{\frac{C_z}{\delta t} + \left(\sum_{i=1}^{N_{surfaces}} h_i A_i + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p + \dot{m}_{inf} C_p + \dot{m}_{sys} C_p \right)} \quad (8)$$

Equation (8) could be used to estimate zone temperatures, however it was found to severely limit the time step size under some conditions. To correct this, higher order expressions for the first derivative, with corresponding higher order truncation errors, were developed. The goal of this approach was to allow for the use of larger time steps in the simulation than would be possible using the first order Euler form, without experiencing instabilities. Approximations from second through fifth order were tried as reported by Taylor, et al (1990) with the conclusion that the third order finite difference approximation, shown below, gave the best results:

$$\frac{dT_z}{dt} \Big|_t \approx (\delta t)^{-1} \left(\frac{11}{6} T_z^t - 3T_z^{t-\delta t} + \frac{3}{2} T_z^{t-2\delta t} - \frac{1}{3} T_z^{t-3\delta t} \right) + O(\delta t^3) \quad (9)$$

When this form for the derivative is used, equation (7) changes to:

$$C_z (\delta t)^{-1} \left(\frac{11}{6} T_z^t - 3T_z^{t-\delta t} + \frac{3}{2} T_z^{t-2\delta t} - \frac{1}{3} T_z^{t-3\delta t} \right) = \sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) + \dot{m}_{inf} C_p (T_{\infty} - T_z) + \dot{m}_{sys} C_p (T_{sup} - T_z) \quad (10)$$

and the zone temperature update equation becomes:

$$T_z^t = \frac{\sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i T_{si} + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p T_{zi} + \dot{m}_{inf} C_p T_{\infty} + \dot{m}_{sys} C_p T_{supply} - \left(\frac{C_z}{\delta t} \right) \left(-3T_z^{t-\delta t} + \frac{3}{2}T_z^{t-2\delta t} - \frac{1}{3}T_z^{t-3\delta t} \right)}{\left(\frac{11}{6} \right) \frac{C_z}{\delta t} + \sum_{i=1}^{N_{surfaces}} h_i A + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p + \dot{m}_{inf} C_p + \dot{m}_{sys} C} \quad (11)$$

This is the form currently used in EnergyPlus. Since the load on the zone drives the entire process, that load is used as a starting point to give a demand to the system. Then a simulation of the system provides the actual supply capability and the zone temperature is adjusted if necessary. This process in EnergyPlus is referred to as a Predictor/Corrector process. It is summarized below.

Code Reference: the **ZoneTempPredictorCorrector** module performs the calculations.

Zone Capacitance Multiplier

If the ZoneCapacitanceMultiplier = 1.0, this represents just the capacitance of the air volume in the specified zone. If the value is not defined it is set to 1.0. This multiplier can be greater than 1.0 if the zone air capacitance needs to be increased for stability of the simulation. This multiplier increases the capacitance of the air volume by increasing the zone volume that is used in the zone predictor-corrector algorithm in the simulation. This can be done for numerical reasons, such as to increase the stability that will decrease the air temperature deviations at the time step level. Or it can be increased to try and account for the additional capacitance in the air loop not specified in the zone, i.e. dampers, diffusers, duct work, etc., to see the effect on the dynamics of the simulation.

In the source code below we see how the ZoneVolCapMultp increases the zone volume used for the air ratio at the time step in the system. This is constant throughout the simulation.

```
AIRRAT (ZoneNum) = Zone (ZoneNum) %Volume * ZoneVolCapMultp * &
  PsyRhoAirFnPbTdbW (OutBaroPress, REAL (MAT (ZoneNum)), ZoneAirHumRat (ZoneNum)) * &
  PsyCpAirFnWTdb (ZoneAirHumRat (ZoneNum), REAL (MAT (ZoneNum))) / (TimeStepZone * SecInHour)
```

Summary of Predictor-Corrector Procedure

The predictor-corrector scheme can be summarized as follows:

- Using equation (3), an estimate is made of the system energy required to balance the equation with the zone temperature equal to the setpoint temperature.
- With that quantity as a demand, the system is simulated to determine its actual supply capability at the time of the simulation. This will include a plant simulation if necessary.
- The actual system capability is used in equation (11) to calculate the resulting zone temperature.

System Control

Previously, the formulation of a new heat balance equation with an unsteady zone capacitance term was discussed Equation (4). In this equation the updated zone temperature was calculated by removing its explicit dependence from the right hand side and lagging, by one time step, the unknown terms on that side. However, the right hand side still contains implicit dependencies on the zone temperature through the system control logic; the need for heating or cooling in the zones, is based on zone temperature. In real buildings the control system consists of one or more sensing units in the zone, such as a wall thermostat that samples the air temperature and sends signals to a control unit. The controller looks at the difference between the actual zone temperature and the desired temperature to ascertain

if heating or cooling is required and then sends appropriate signals to the system components to drive the zone temperature closer to the desired value.

Although many control systems use only the zone temperature to control the system, most modern energy management systems consider many other variables, such as outside environment conditions. Simulating such controllers would seem to be relatively straightforward in a simulation especially since some of the more complex control problems, such as managing duct pressures and flow rates, are not modeled. However, real controllers have an advantage because they can sample zone conditions, and thus update system response, on a time scale much shorter than any characteristic time of the system or zone. Thus the feedback between zone and system usually results in steady or, at worst, slowly oscillating zone conditions and system operation unless the system is grossly oversized. On the other hand, the numerical model is only able to sample zone conditions at discrete time intervals. In the interest of minimizing computation time, these intervals need to be as long as possible. Frequently, they are of the order of, or longer than, the characteristic times of the system and zones, except in the case of small system capacity in relation to zone capacitance. This situation has the potential for unstable feedback between zone and system, resulting in an oscillatory or diverging solution.

Prior to implementing the new heat balance method in IBLAST, several system control strategies were considered. The primary objective was selection of a control method that would: be numerically stable over a reasonable range of conditions, realistic from the standpoint of looking and operating like an actual system controller, and flexible enough to be applied to all current and projected systems. The method actually implemented in IBLAST, and later EnergyPlus, took advantage of the computational model's "knowledge" of how much energy enters or leaves the zone as a function of zone temperature i.e., the zone load. The real controller, on the other hand, does not have this information. The net zone load is given by Equation (12):

$$\dot{Q}_{load} = \sum_{i=1}^{N_{si}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i (T_{si} - T_z) + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p (T_{zi} - T_z) + \dot{m}_{inf} C_p (T_{\infty} - T_z) \quad (12)$$

This is Equation (4) without the term due to the system. In addition, T_z is now the *desired* zone temperature as defined by the control system setpoints that must be specified for each zone. An assumption was made that if the system has sufficient capacity (based on the desired zone temperature) to meet the zone conditioning requirements (i.e. $\dot{Q}_{sys} = \dot{Q}_{load}$) at the desired zone temperature then those requirements will be met. On the other hand, if the system can not provide enough conditioning to the zone to maintain the desired temperature, then the system provides its maximum output to the zone and the zone temperature is allowed to "float." Equation (12) was used to calculate the system output required to maintain the desired zone temperature; the actual zone temperature update was accomplished using Equation (8). This method was called *predictive system energy balance*. It has many characteristics of a predictor-corrector method since the system response is first approximated based on a predicted zone temperature and then the actual change in zone temperature is determined from that system response. The predictive system energy balance method required that the system controls on air mass flow rate, supply air temperature, etc., be formulated as a function of the zone temperature. However, this was not a serious drawback. The first example considered was a single zone draw through system. Typically, such systems have a cooling coil and heating coil in series, and constant air volume flow rate. Single zone draw through systems run at maximum capacity when turned on so the only way to regulate net system output and keep the zone temperature within the desired range is to turn the system on and off. A simplified schematic of this system type is shown in Figure "Simplified Single Zone Draw Through System".

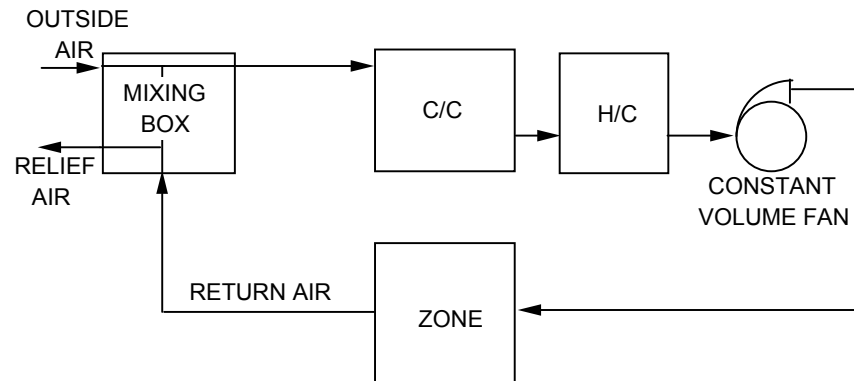


Figure 4. Simplified Single Zone Draw Through System

The amount of heating or cooling provided by the system in relation to the desired zone temperature is given by:

$$\dot{Q}_{sys} = \dot{m}_{sys} C_p \eta (T_{sup} - T_{z,desired}) \quad (13)$$

where η is the fraction of the time step that the system is turned on and varies between 0 and 1. The supply air temperature is also implicitly limited by the effectiveness of the coils and the operating parameters of the central plant components. These interactions are discussed later.

A far more complex, though again simplified, system is the variable air volume (VAV) system, shown in Figure "Simplified Variable Volume System." In VAV systems, the supply air temperature as well as the supply air volume are continuous functions of zone temperature. As shown in Figure "Idealized Variable Volume System Operation.", when the zone temperature is between T_{cl} and T_{cu} , cooling is required and the system varies the supply air flow rate while maintaining a constant supply air temperature. When the zone temperature is between T_{hl} and T_{hu} , heating is required and air is supplied at a constant minimum flow rate while the supply air temperature is varied.

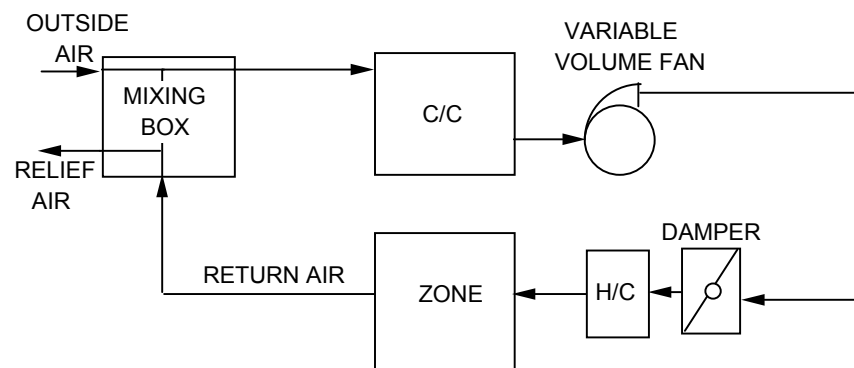


Figure 5. Simplified Variable Volume System.

The next figure (Idealized variable volume system operation) shows idealized behavior of a VAV system; in practice, the air flow rate and temperature are not exact linear functions of zone temperature.

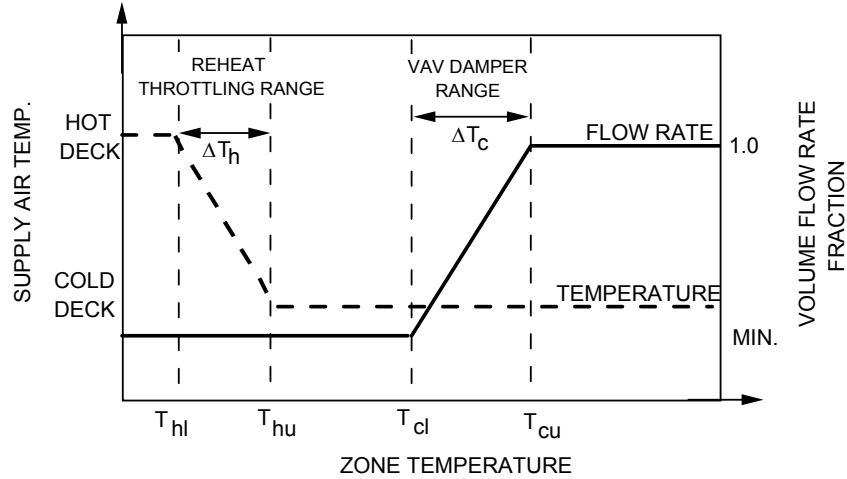


Figure 6. Idealized Variable Volume System Operation.

As long as a VAV system has sufficient capacity, the zone temperatures can be expected to vary within the limits defining the range of operation of the air damper, when cooling, or the throttling range of the reheat coil, when the system is heating. This means that the desired zone temperature, used to predict the system response, is variable and must be calculated in order to determine the system output. For the purposes of this calculation, the following definitions were found useful:

$$\dot{Q}_0 = \sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i T_{si} + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p T_{zi} + \dot{m}_{inf} C_p T_{\infty} \quad (14)$$

$$\dot{Q}_{slope} = \sum_{i=1}^{N_{surfaces}} h_i A_i + \sum_{i=1}^{N_{zones}} \dot{m}_i C_p + \dot{m}_{inf} C_p \quad (15)$$

Equations (14) and (15) are derived, respectively, from the numerator and denominator of Equation (11) but with the system related terms omitted. Also excluded from these expressions are the effects of zone capacitance.

When a zone requires cooling, the VAV system is designed to provide air to that zone at a constant supply air temperature. The amount of cooling is matched to the load by dampers in the supply air duct that vary the air volume flow rate of being supplied to the zone. Assuming that the volume flow rate varies linearly with zone temperature, the volume flow rate of supply air normalized to the maximum flow rate, or supply air fraction, is given by:

$$\eta_c = \eta_{c,\min} + (1 - \eta_{c,\min}) \left(\frac{T_z - T_{c,lower}}{T_{c,upper} - T_{c,lower}} \right); \eta_{c,\min} \leq \eta_c \leq 1.0 \quad (16)$$

Normally, the minimum supply air fraction $\eta_{c,\min}$ must be greater than zero to ensure a supply of fresh air sufficient to eliminate contaminants from the zone.

Conversely, when heating is required in a zone, the VAV system becomes a constant volume flow rate system with a variable supply air temperature. The dampers are set to provide air to the zone at the minimum supply air fraction. Throttling the hot water supply to the reheat coil, which effectively alters the coil's heating capacity, modulates the supply air temperature. Again, assuming the heat energy output varies linearly with zone temperature and normalizing with respect to the maximum coil output gives the following result:

$$\eta_h = \left(\frac{T_{h,upper} - T_z}{T_{h,upper} - T_{h,lower}} \right); 0 \leq \eta_h \leq 1.0 \quad (17)$$

Observe that when η_h is equal to zero, the zone is supplied with air at the cooling coil outlet temperature at the minimum air fraction. Because the control strategies of the VAV system are different whether the system is heating or cooling, two equations are necessary to describe the system output in terms of η_h and η_c . These expressions are as shown in Equations (18) and (19):

$$\dot{Q}_{sys,h} = \eta_h \dot{Q}_{h/c,max} + C_p \rho \dot{V}_{min} (T_{c/c} - T_{z,pred,heat}) \quad (18)$$

$$\dot{Q}_{sys,c} = C_p \rho (\eta_c \dot{V}_{max}) (T_{c/c} - T_{z,pred,cool}) \quad (19)$$

Equation (18) is valid for zone temperatures below $T_{h,upper}$, while Equation (19) is valid for all temperatures above this value. Equating the system output to the zone load, as given by Equation (12), the definitions of η_c and η_h were then used to develop expressions for the predicted zone temperature in the cases of heating and cooling:

$$T_{z,pred,heat} = \frac{\dot{Q}_{h/c,max} T_{h,upper}}{T_{h,upper} - T_{h,lower}} + \dot{Q}_0 + \frac{C_p \rho \dot{V}_{min} T_{c/c}}{\frac{\dot{Q}_{h/c,max}}{T_{h,upper} - T_{h,lower}} + C_p \rho \dot{V}_{min} + \dot{Q}_{slope}} \quad (20)$$

$$T_{z,pred,cool} = \frac{B_1 + \sqrt{B_1^2 + B_2}}{2} \quad (21)$$

where,

$$B_1 = T_{c/c} + T_{c,lower} - \frac{\eta_{c,min} - C_2}{C_1} \quad (22)$$

$$B_2 = 4 \left(\frac{C_3}{C_1} + T_{c/c} \left(\frac{\eta_{c,min}}{C_1} - T_{c,lower} \right) \right) \quad (23)$$

and,

$$C_1 = \frac{1 - \eta_{c,min}}{T_{c,upper} - T_{c,lower}} \quad (24)$$

$$C_2 = \frac{\dot{Q}_{slope}}{C_p \rho \dot{V}_{max}} \quad (25)$$

$$C_3 = \frac{\dot{Q}_0}{C_p \rho \dot{V}_{\max}} \quad (26)$$

Once the predicted zone temperature has been calculated from Equations (20) and (21), the system response may be determined. When a zone requires cooling the system supply air temperature is constant at the cooling coil outlet temperature and the volume flow rate is given by:

$$\dot{V}_{\text{supply}} = \eta_c \dot{V}_{\max} \quad (27)$$

where the supply air fraction η_c is computed from Equation (16). When heating is required by the zone, the system provides air at the minimum volume flow rate and at a temperature given by:

$$T_{\text{supply}} = T_{c/c} + \frac{\eta_h \dot{Q}_{h/c,\max}}{C_p \rho \dot{V}_{\min}} \quad (28)$$

The reheat coil capacity fraction η_h is determined by using Equation (17). Once Equation (27) or (28), has been used, the supply air flow rate and temperature are known. These values are then used in Equation (8) to calculate the updated zone temperature. The equations describing VAV system operation may be solved without iteration if the cooling coil outlet temperature is constant, i.e. if the coil has infinite capacity, and if the reheat coil capacity varies linearly with zone temperature. This is not the case, either in practice or in simulations, when realistic coil models are used. Therefore, an iteration scheme was developed that solved these equations simultaneously with the coil performance models.

Moisture Predictor-Corrector

To preserve the stability of the calculation of the zone humidity ratio, a similar methodology was used for the mass balance as was used by temperature in the heat balance as explained above in the Summary of Predictor-Corrector Procedure. The third order differential approximation derived by a Taylor Series was used in the calculation of the next time steps zone air temperature

This idea was applied to predict and correct, or update, the zone humidity ratio. Writing the transient mass balance equation with the change in the zone humidity ratio = sum of internal scheduled latent loads + infiltration + system in & out + convection to the zone surfaces in the equation below.

$$\rho_{\text{air}} V_z \frac{dW_z}{dt} = \sum k g_{\text{massSched Loads}} + \dot{m}_{\text{inf}} (W_{\infty} - W_z) + \dot{m}_{\text{sys}} W_{\text{sysin}} - \dot{m}_{\text{sysout}} W_z + \sum_{i=1}^{\text{surfs}} A_i h_{mi} \rho_{\text{air}_z} (W_{\text{surfs}_i} - W_z)$$

Then the third order derivative derived from a Taylor Series expansion is defined as:

$$\left. \frac{dW_z}{dt} \right|_t \approx \frac{\left(\frac{11}{6} W_z^t - 3W_z^{t-\delta t} + \frac{3}{2} W_z^{t-2\delta t} - \frac{1}{3} W_z^{t-3\delta t} \right)}{\delta t} + O(\delta t^3).$$

The coefficients of the approximated derivative are very close to the coefficients of the analogous Adams-Bashforth algorithm. Then the approximated derivative is substituted into the mass balance and the terms with the humidity ratio at past time steps are all put on the right hand side of the equation. This third order derivative zone humidity ratio update increases the number of previous time steps that are used in calculating the new zone humidity ratio, and decreases the dependence on the most recent. The higher order derivative approximations have the potential to allow the use of larger time steps by smoothing transitions through sudden changes in zone operating conditions.

$$\frac{\rho_{air} \mathcal{V}_z}{\delta t} \left(\frac{11}{6} \right) W_z^t + \dot{m}_{inf} W_z^t + \dot{m}_{sys_{out}} W_z^t + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_z^t = \sum kg_{massSched\ Loads} + \dot{m}_{inf} W_\infty + \dot{m}_{sys} W_{sys_{in}}$$

$$+ \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i} - \frac{\rho_{air} \mathcal{V}_z}{\delta t} \left(-3W_z^{t-\delta t} + \frac{3}{2} W_z^{t-2\delta t} - \frac{1}{3} W_z^{t-3\delta t} \right)$$

This gives us the basic mass balance equation that will be solved two different ways, one way for the predict step and one way for the correct step.

Moisture Prediction

For the moisture prediction case the equation is solved for the anticipated system response as shown below.

$$\text{PredictedSystemLoad} = \dot{m}_{sys_{out}} W_z^t - \dot{m}_{sys} W_{sys_{in}}$$

$$\text{Massflow} * \text{HumRat} = \text{kg air/sec} * \text{kgWater/kg Air} = \text{kgWater/sec}$$

Then solving the mass balance for the predicted system load or response is:

$$\text{PredictedSystemLoad [kgWater / sec]} = \left[\frac{\rho_{air} \mathcal{V}_z}{\delta t} \left(\frac{11}{6} \right) + \dot{m}_{inf} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} \right] * W_z^t -$$

$$\left[\sum kg_{massSched\ Loads} + \dot{m}_{inf} W_\infty + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i} + \frac{\rho_{air} \mathcal{V}_z}{\delta t} \left(3W_z^{t-\delta t} - \frac{3}{2} W_z^{t-2\delta t} + \frac{1}{3} W_z^{t-3\delta t} \right) \right]$$

Then using the following substitutions the mass balance equation becomes:

$$A = \dot{m}_{inf} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z}$$

$$B = \sum kg_{massSched\ Loads} + \dot{m}_{inf} W_\infty + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i}$$

$$C = \frac{\rho_{air} \mathcal{V}_z}{\delta t}$$

$$\text{PredictedSystemLoad [kgWater / sec]} = \left[\left(\frac{11}{6} \right) * C + A \right] * W_{SetPoint} -$$

$$\left[B + C * \left(3W_z^{t-\delta t} - \frac{3}{2} W_z^{t-2\delta t} + \frac{1}{3} W_z^{t-3\delta t} \right) \right]$$

At the prediction point in the simulation the system mass flows are not known, therefore the system response is approximated. The predicted system moisture load is then he used in the system simulation to achieve the best results possible. The system simulation components that have moisture control will try to meet this predicted moisture load. For example, humidifiers will look for positive moisture loads and add moisture at the specified rate to achieve the relative humidity setpoint. Likewise, dehumidification processes will try to remove moisture at the specified negative predicted moisture load to meet the relative humidity setpoint.

After the system simulation is completed the actual response from the system is used in the moisture correction of step, which is shown next.

Moisture Correction

For the correct step you solve the expanded mass balance equation for the final zone humidity ratio at the current time step. When the system is operating the mass flow for the system out includes the infiltration mass flow rate, therefore the infiltration mass flow rate is not included as a separate term in the mass balance equation. But when the system is off, the infiltration mass flow in is then exhausted out of the zone directly.

For system operating $\dot{m}_{sys_{out}} = \dot{m}_{inf} + \dot{m}_{sys_{in}}$. This ensures that the mass and energy brought into the zone due to infiltration is conditioned by the coils and is part of their load.

$$W_z^t = \frac{\left[\sum kg_{massSched\ Loads} + \dot{m}_{inf} W_\infty + \dot{m}_{sys} W_{sys_{in}} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i} + \frac{\rho_{air} \Psi_z}{\delta t} \left(3W_z^{t-\delta t} - \frac{3}{2} W_z^{t-2\delta t} + \frac{1}{3} W_z^{t-3\delta t} \right) \right]}{\frac{\rho_{air} \Psi_z}{\delta t} \left(\frac{11}{6} \right) + \dot{m}_{sys_{out}} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z}}$$

For system off, the mass flow in due to infiltration is exhausted from the zone. Therefore if the system is shut down for a longer time, then the zone should stabilize at the outside air humidity ratio and upon system startup the added mass and heat in the zone air is handled by the system.

$$W_z^t = \frac{\left[\sum kg_{massSched\ Loads} + \dot{m}_{inf} W_\infty + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i} + \frac{\rho_{air} \Psi_z}{\delta t} \left(3W_z^{t-\delta t} - \frac{3}{2} W_z^{t-2\delta t} + \frac{1}{3} W_z^{t-3\delta t} \right) \right]}{\frac{\rho_{air} \Psi_z}{\delta t} \left(\frac{11}{6} \right) + \dot{m}_{inf} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z}}$$

and then using the same A, B, and C parameters from the prediction step modified with actual zone mass flows with the system ON and OFF:

If (ZoneMassFlowRate > 0.0) Then

$$B = \sum kg_{massSched\ Loads} + \dot{m}_{inf} W_{\infty} + \dot{m}_{sys_{in}} W_{sys} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i}$$

$$A = \dot{m}_{inf} + \dot{m}_{sys_{out}} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z}$$

$$C = \frac{\rho_{air} V_z}{\delta t}$$

Else If (ZoneMassFlowRate <= 0.0) **Then**

$$B = \sum kg_{massSched\ Loads} + (\dot{m}_{inf} + \dot{m}_{Exhaust}) W_{\infty} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z} W_{surfs_i}$$

$$A = \dot{m}_{inf} + \dot{m}_{Exhaust} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air_z}$$

End If

Then inserting them in the mass balance equation, it simplifies to:

$$W_z^t = \left[\frac{B + C * \left(3W_z^{t-\delta t} - \frac{3}{2}W_z^{t-2\delta t} + \frac{1}{3}W_z^{t-3\delta t} \right)}{\left(\frac{11}{6} \right) * C + A} \right]$$

This is implemented in the Correct Zone Air Humidity Ratio step in EnergyPlus. This moisture update equation is used for the Conduction Transfer Function (CTF) solution algorithm, in addition to the moisture penetration depth with conduction transfer function (EMPD) solution algorithm. The equations are identical except that the convection to the zone surfaces is non-zero for the moisture cases. This moisture update allows both methods to be updated in the same way, and the only difference will be the additional moisture capacitance of the zone surfaces for the Effective Moisture Penetration Depth (EMPD) solution approach.

Summary of Time Marching Solution

EnergyPlus models building performance over time spans of days to years using a time marching method based on *timesteps*. This section provides more information on issues related to timestep formulation.

Summary of Timestep Model Formulation

An EnergyPlus simulation covers a certain period of time, such as a day or a year, that is broken down into a series of discrete bins of time that are referred to as *timesteps*. The program marches through time by recalculating model equations at each timestep. The figure below diagrams some of these basic concepts.

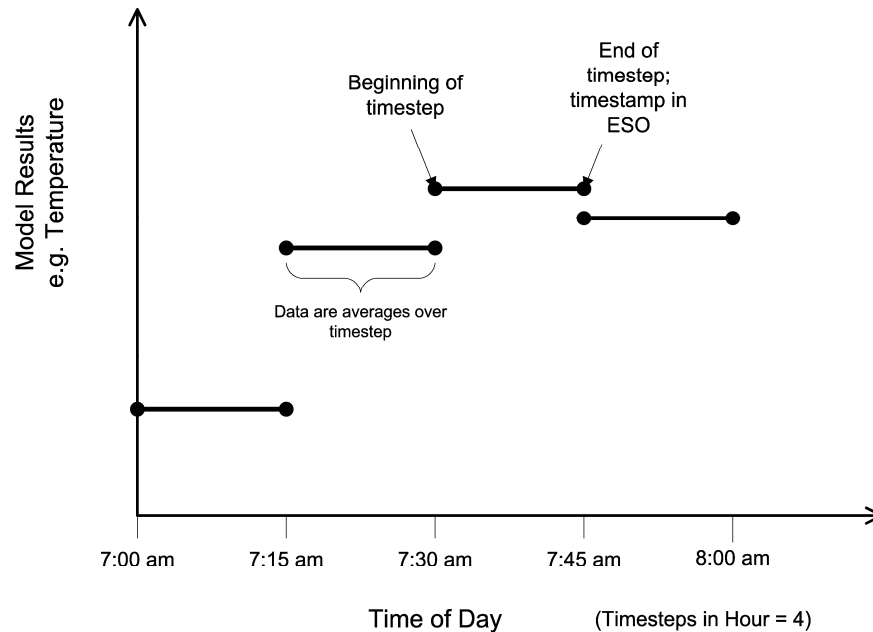


Figure 7. Diagram of TimeStep Formulation

Most models in EnergyPlus are quasi-steady energy balance equations used to predict the conditions present during each timestep. Various input data and boundary conditions for the models are time-varying and a “staircase” approach is used where such values are determined for a particular timestep and then essentially held constant over the entire timestep. Predictions for state variables, such as temperature, are averages over the timestep. Predictions for summed variables, such as energy use, are simple totals over the timestep.

EnergyPlus produces time-series results for selected output variables at selected frequencies. The time values associated with the time-series data, or timestamps, are output for the end of the timestep, but the values of the variables are for the entire bin of time prior to the timestamp. When data are reported at coarser frequencies such as hourly, then the results are averages or simple totals for all the timesteps that are contained within the larger bin of time.

To simplify solutions that would otherwise need to be simultaneous, models sometimes use data that are “lagged” which means that the values used in the calculations for the current timestep are actually results from the previous timestep. Many models just use the most current results available and so may use lagged data for a first iteration, but then use current data that are not lagged for subsequent iterations.

Zone Update Method

A zone is not necessarily a single room but is usually defined as a region of the building or a collection of rooms subject to the same type of thermal control and having similar internal load profiles that, subsequently, can be grouped together. Zones can interact with each other thermally through adjacent surfaces and by intermixing of zone air. In EnergyPlus, the conditions in each zone are updated by Equation (8), which uses previously calculated values of the zone conditions. This means that EnergyPlus does not have to iterate to find a self consistent solution of the updated zone conditions. However, because heat transfer through each zone’s surfaces and interzone mixing of air still occur, the new space temperatures must be computed at the same simulation time and on the same time step in all zones, even though conditions in one zone may be changing much more rapidly than conditions in the other zones. We have previously documented the method used to update the zone

temperature at each time step. This method, called the predictor corrector method, has proved to be satisfactory over a wide range of conditions.

Variable Timestep

The need for a variable timestep in EnergyPlus was identified during development of its predecessor IBLAST. Prior to the integration of the central plant simulation in IBLAST, a time step Δt for the zone temperature update of 0.25 hours (15 minutes) was found to give stable results without a large increase in computation time. The first step in integrating plant was to implement the detailed coil models and coil control strategies without actually adding the plant models themselves. This meant that the user had to specify the coil water inlet temperature and the maximum coil inlet water flow rate to run the simulation. The real life analogy would be a chiller of very large, though not infinite, capacity. The coil capacity was controlled by adjusting the water flow rate, but the effect of the plant on the chilled water temperature was eliminated. After implementation of this step, experience with the program showed that updating the zone temperatures on a fixed time step frequently resulted in instabilities unless a very short time step was used. However, as the time step got shorter the time required to execute the program got prohibitively high.

Clearly, an adaptive time step was required. This would shorten the time step to maintain stability of the zone air energy balance calculation when zone conditions were changing rapidly and expand it to speed computation when zone conditions were relatively unchanging. But, the adaptive time step could not be applied easily to the surface heat transfer calculations, even using interpolation methods to determine new temperature and flux histories. The problem of updating the zone temperature was resolved by using a two-time-step approach in which the zone air temperature is updated using an adaptive time step that ensures stability. In this two time level scheme, the contributions to the zone loads from the surfaces, and user specified internal loads are updated at the default or user specified time step that is constant. This is often referred to as the "zone" time step. The contributions to the zone loads from HVAC system response, infiltration, and air mixing are updated at a second variable time step, referred to as the "system" time step. The system time step is limited to between one minute and the zone time step. The lower limit of the system time step can be increased to larger than one minute by the user with a System Convergence Limits object (which may be useful to decrease simulation run times at the expense of some accuracy).

The program's decision to adapt the time step is made by first using the usual zone time step and executing the full predictor-corrector calculations to find resulting zone temperatures. The maximum temperature change experienced by any zone in the model is determined. If this maximum zone temperature change is more than a preset limit of 0.3°C, then the simulation switches to using the shorter system time step. The number of system time steps (within a particular zone time step) is modeled from the results for the maximum zone temperature change (just obtained from the corrector) by assuming that temperature change is linear. The number of system time steps indicated by the temperatures is:

$$\left(\frac{\text{Maximum Zone Temperature Change}}{\text{Maximum Zone Temperature Difference } \{.3C\}} \right) + 1.0$$

The limit for the number of system time steps is:

$$\left(\frac{\text{Zone Time Step}}{\text{Minimum System Time Step}} \right)$$

EnergyPlus takes the smallest of these two, truncates them to a whole number and calculates a system time step as:

$$\text{System Time Step} = \left(\frac{\text{Zone Time Step}}{\text{Number of System Time Steps}} \right)$$

The same length of time is used for all the system time steps within a particular zone time step, but the system time step length can vary between different zone time steps. Report variables that use the “detailed” frequency show results at the system time step time scale.

This approach can be justified because the internal loads and surface temperatures vary on a different and longer time scale than the system response and the zone air temperature.

When the simulation down-steps from using the zone time step to the system time step, linear interpolation is used to generate zone condition history terms ($T_z^{t-\delta t}$, $T_z^{t-2\delta t}$, $T_z^{t-3\delta t}$) for the system time steps using the history data for the zone time step. If two consecutive zone time steps both down-step to have the same number of system time steps, then the system time step history is preserved and used directly rather than being interpolated from zone time step history. The zone temperature update is made for each system time step using the same equation (11) as for the zone time step except that the zone temperature history and time difference (δt) terms are for the system time step instead of for the zone time step.

Simultaneous Solution of Plant/System Water Loop

Simultaneous solution of the system and plant operating parameters required that the temperature of the water entering the coils must be the same as the temperature leaving the chillers or boilers. In addition, the temperature of the return water from the coils must be equal to the chiller or boiler entering water temperature. In practice so long as the plant is not out of capacity the leaving water temperature from chillers and boilers is constant and equal to the design value. No iteration was required to match system and plant boundary conditions. However, if either the chiller or boiler plant was overloaded then the temperature of the water leaving the plant was not equal to the design value and the maximum output of the plant could change because of the off-design conditions. An iterative scheme using the secant method to predict successive updates to the plant leaving water conditions was therefore employed to solve for the water loop conditions with the plant operating at its maximum capacity. The simulation of the system and plant loops is described in greater detail in the later sections.

References

- Ceylan, H. T., and G. E. Myers. 1980. Long-time Solutions to Heat Conduction Transients with Time-Dependent Inputs. ASME Journal of Heat Transfer, Volume 102, No. 1, pp. 115-120.
- Hittle, D. C. 1979. Calculating Building Heating and Cooling Loads Using the Frequency Response of Multilayered Slabs, Ph.D. Thesis, University of Illinois, Urbana, IL.
- Hittle, D. C., and R. Bishop. 1983. An Improved Root-Finding Procedure for Use in Calculating Transient Heat Flow Through Multilayered Slabs. International Journal of Heat and Mass Transfer, Vol. 26, No. 11, pp. 1685-1693.
- Ouyang, K., and F. Haghghat. 1991. A Procedure for Calculating Thermal Response Factors of Multi-layered Walls--State Space mMethod. Building and Environment, Vol. 26, No. 2, pp. 173-177.
- Seem, J. E. 1987. Modeling of Heat Transfer in Buildings, Ph.D. Thesis, University of Wisconsin, Madison, WI.
- Strand, R. K. 1995. Heat Source Transfer Functions and Their Application to Low Temperature Radiant Heating Systems, Ph.D. Thesis, University of Illinois, Urbana, IL.
- Taylor, R. D., C.O. Pedersen, L.K. Lawrie. 1990. *Simultaneous Simulation of Buildings and Mechanical Systems in Heat Balance Based Energy Analysis Programs*, Proceedings of the 3rd International Conference on System Simulation in Buildings, Liege, Belgium, December 3-5, 1990.

Taylor, R.D., C.O. Pedersen, D.E. Fisher, R. J. Liesen, L.K. Lawrie. 1991. Impact of Simultaneous Simulation of Buildings and Mechanical Systems in Heat Balance Based Energy Analysis Programs on System Response and Control, Conference Proceedings IBPSA Building Simulation '91, Nice, France, August 20-22, 1991.

Surface Heat Balance Manager / Processes

Conduction Through The Walls

Conduction Transfer Function Module

The most basic time series solution is the response factor equation which relates the flux at one surface of an element to an infinite series of temperature histories at both sides as shown by Equation (29):

$$q''_{ko}(t) = \sum_{j=0}^{\infty} X_j T_{o,t-j\delta} - \sum_{j=0}^{\infty} Y_j T_{i,t-j\delta} \quad (29)$$

where q'' is heat flux, T is temperature, i signifies the inside of the building element, o signifies the outside of the building element, t represents the current time step, and X and Y are the response factors.

While in most cases the terms in the series decay fairly rapidly, the infinite number of terms needed for an exact response factor solution makes it less than desirable. Fortunately, the similarity of higher order terms can be used to replace them with flux history terms. The new solution contains elements that are called conduction transfer functions (CTFs). The basic form of a conduction transfer function solution is shown by the following equation:

$$q''_{ki}(t) = -Z_o T_{i,t} - \sum_{j=1}^{nz} Z_j T_{i,t-j\delta} + Y_o T_{o,t} + \sum_{j=1}^{nz} Y_j T_{o,t-j\delta} + \sum_{j=1}^{nq} \Phi_j q''_{ki,t-j\delta} \quad (30)$$

for the inside heat flux, and

$$q''_{ko}(t) = -Y_o T_{i,t} - \sum_{j=1}^{nz} Y_j T_{i,t-j\delta} + X_o T_{o,t} + \sum_{j=1}^{nz} X_j T_{o,t-j\delta} + \sum_{j=1}^{nq} \Phi_j q''_{ko,t-j\delta} \quad (31)$$

for the outside heat flux ($q''=q/A$)

where:

X_j = Outside CTF coefficient, $j= 0,1,\dots,nz$.

Y_j = Cross CTF coefficient, $j= 0,1,\dots,nz$.

Z_j = Inside CTF coefficient, $j= 0,1,\dots,nz$.

Φ_φ = Flux CTF coefficient, $j= 1,2,\dots,nq$.

T_i = Inside face temperature

T_o = Outside face temperature

q''_{ko} = Conduction heat flux on outside face

q'' = Conduction heat flux on inside face

The subscript following the comma indicates the time period for the quantity in terms of the time step δ . Note that the first terms in the series (those with subscript 0) have been separated from the rest in order to facilitate solving for the current temperature in the solution scheme. These equations state that the heat flux at either face of the surface of any generic building element is linearly related to the current and some of the previous temperatures at both the interior and exterior surface as well as some of the previous flux values at the interior surface.

The final CTF solution form reveals why it is so elegant and powerful. With a single, relatively simple, linear equation with constant coefficients, the conduction heat transfer through an element can be calculated. The coefficients (CTFs) in the equation are constants that only need to be determined once for each construction type. The only storage of data required are the CTFs themselves and a limited number of temperature and flux terms. The formulation is valid for any surface type and does not require the calculation or storage of element interior temperatures.

Calculation of Conduction Transfer Functions

The basic method used in EnergyPlus for CTF calculations is known as the state space method (Ceylan and Myers 1980; Seem 1987; Ouyang and Haghighat 1991). Another common, older method used Laplace transformations to reach the solution; the Laplace method was used in BLAST (Hittle, 1979; Hittle & Bishop, 1983). The basic state space system is defined by the following linear matrix equations:

$$\frac{d[\mathbf{x}]}{dt} = [\mathbf{A}][\mathbf{x}] + [\mathbf{B}][\mathbf{u}]$$

$$[\mathbf{y}] = [\mathbf{C}][\mathbf{x}] + [\mathbf{D}][\mathbf{u}]$$

where \mathbf{x} is a vector of state variables, \mathbf{u} is a vector of inputs, \mathbf{y} is the output vector, t is time, and \mathbf{A} , \mathbf{B} , \mathbf{C} , and \mathbf{D} are coefficient matrices. Through the use of matrix algebra, the vector of state variables (\mathbf{x}) can be eliminated from the system of equations, and the output vector (\mathbf{y}) can be related directly to the input vector (\mathbf{u}) and time histories of the input and output vectors.

This formulation can be used to solve the transient heat conduction equation by enforcing a finite difference grid over the various layers in the building element being analyzed. In this case, the state variables are the nodal temperatures, the environmental temperatures (interior and exterior) are the inputs, and the resulting heat fluxes at both surfaces are the outputs. Thus, the state space representation with finite difference variables would take the following form:

$$\frac{d \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix}}{dt} = [\mathbf{A}] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [\mathbf{B}] \begin{bmatrix} T_i \\ T_o \end{bmatrix}$$

$$\begin{bmatrix} q''_i \\ q''_o \end{bmatrix} = [\mathbf{C}] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [\mathbf{D}] \begin{bmatrix} T_i \\ T_o \end{bmatrix}$$

where $T_1, T_2, \dots, T_{n-1}, T_n$ are the finite difference nodal temperatures, n is the number of nodes, T_i and T_o are the interior and exterior environmental temperatures, and q''_i and q''_o are the heat fluxes (desired output).

Seem (1987) shows that for a simple one layer slab with two interior nodes as in Figure 7 and convection at both sides the resulting finite difference equations are given by:

$$C \frac{dT_1}{dt} = hA(T_o - T_1) + \frac{T_2 - T_1}{R}$$

$$C \frac{dT_2}{dt} = hA(T_i - T_2) + \frac{T_1 - T_2}{R}$$

$$q''_i = h(T_i - T_2)$$

$$q''_o = h(T_1 - T_o)$$

where:

$$R = \frac{\ell}{kA},$$

$$C = \frac{\rho c_p \ell A}{2}, \text{ and}$$

A is the area of the surface exposed to the environmental temperatures.

In matrix format:

$$\begin{bmatrix} \frac{dT_1}{dt} \\ \frac{dT_2}{dt} \end{bmatrix} = \begin{bmatrix} -\frac{1}{RC} - \frac{hA}{C} & \frac{1}{RC} \\ \frac{1}{RC} & -\frac{1}{RC} - \frac{hA}{C} \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} \frac{hA}{C} & 0 \\ 0 & \frac{hA}{C} \end{bmatrix} \begin{bmatrix} T_o \\ T_i \end{bmatrix}$$

$$\begin{bmatrix} q''_o \\ q''_i \end{bmatrix} = \begin{bmatrix} 0 & -h \\ h & 0 \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} 0 & h \\ -h & 0 \end{bmatrix} \begin{bmatrix} T_o \\ T_i \end{bmatrix}$$

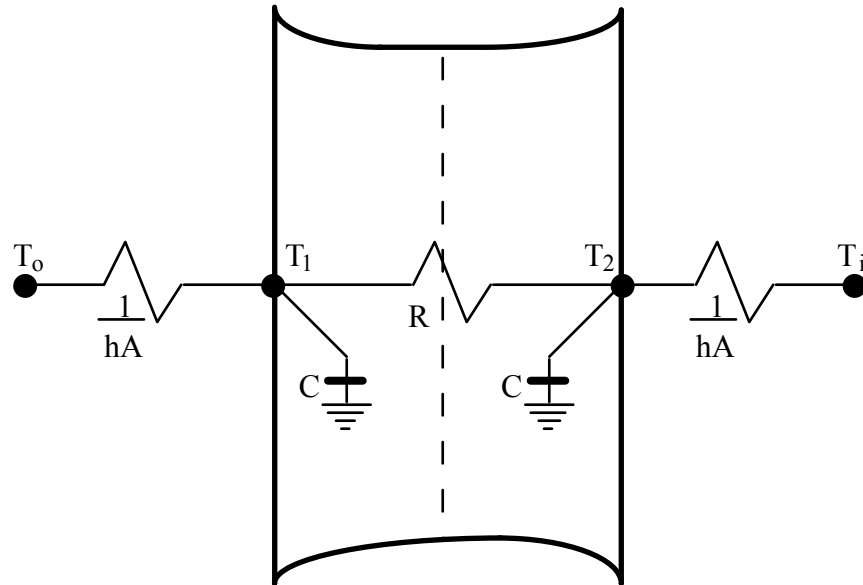


Figure 8. Two Node State Space Example.

The important aspect of the state space technique is that through the use of matrix algebra the state space variables (nodal temperatures) can be eliminated to arrive at a matrix equation that gives the outputs (heat fluxes) as a function of the inputs (environmental temperatures) only. This eliminates the need to solve for roots in the Laplace domain. In addition, the resulting matrix form has more physical meaning than complex functions required by the Laplace transform method.

The accuracy of the state space method of calculating CTFs has been addressed in the literature. Ceylan and Myers (1980) compared the response predicted by the state space method to various other solution techniques including an analytical solution. Their results showed that for an adequate number of nodes the state space method computed a heat flux at the surface of a simple one layer slab within 1% of the analytical solution. Ouyang and Haghighat (1991) made a direct comparison between the Laplace and state space methods. For a wall composed of insulation between two layers of concrete, they found almost no difference in the response factors calculated by each method.

Seem (1987) summarizes the steps required to obtain the CTF coefficients from the A, B, C, and D matrices. While more time consuming than calculating CTFs using the Laplace Transform method, the matrix algebra (including the calculation of an inverse and exponential matrix for A) is easier to follow than root find algorithms. Another difference between the Laplace and State Space methods is the number of coefficients required for a solution. In general, the State Space method requires more coefficients. In addition, the number of temperature and flux history terms is identical ($n_z = n_q$). Note that as with the Laplace method that the actual number of terms will vary from construction to construction.

Two distinct advantages of the State Space method over the Laplace method that are of interest when applying a CTF solution for conduction through a building element are the ability to obtain CTFs for much shorter time steps and the ability to obtain 2- and 3-D conduction transfer functions. While not implemented in the Toolkit, both Seem (1987) and Strand (1995) have demonstrated the effectiveness of the State Space method in handling these situations that can have important applications in buildings.

EnergyPlus CTF Calculations

Conduction transfer functions are an efficient method to compute surface heat fluxes because they eliminate the need to know temperatures and fluxes within the surface. However, conduction transfer function series become progressively more unstable as the

time step decreases. This became a problem as investigations into short time step computational methods for the zone/system interactions progressed because, eventually, this instability caused the entire simulation to diverge. This phenomenon was most apparent for thermally massive constructions with long characteristic times and, correspondingly, requiring a large number of terms in the CTF series. This indicates that the problem is related to round-off and truncation error and is in no way an indictment of the CTF method itself. Methods that develop CTF series from finite difference approximations to the heat conduction equation (Meyers, 1980; Seem, 1987) were considered to address this problem. Seem's method did give better accuracy and stability at short time steps than the current BLAST technique but, the method still had difficulty computing stable CTF series for time steps of less than 1/4 hour for the heaviest constructions in the BLAST library.

The zone heat gains consist of specified internal heat gains, air exchange between zones, air exchange with the outside environment, and convective heat transfer from the zone surfaces. Of these, the surface convection load requires the most complicated calculations because a detailed energy balance is required at the inside and outside surface of each wall, floor, and roof. In addition, the transient heat conduction in the material between the surfaces must be solved. This solution gives the inside and outside temperatures and heat fluxes that must be known in order to calculate the convection component to the zone load for each zone surface. BLAST uses a conduction transfer function CTF method attributed to Hittle (1980) to solve the transient conduction problem for each surface. The method results in a time series of weighting factors that, when multiplied by previous values of the surface temperatures and fluxes and the current inside and outside surface temperatures, gives the current inside and outside heat flux. The method is easily applied to multilayered constructions for which analytical solutions are unavailable. In addition, determining the series of CTF coefficients is a one time calculation, making the method much faster than finite difference calculations.

A problem with CTF methods is that the series time step is fixed; that is, a CTF series computed for a one hour time step takes information at $t-1$ hours, $t-2$ hours, etc. and computes conditions at the current time t . As time advances the oldest term in the input series is dropped and the data moved back one time step to allow the newest value to be added to the series. For convenience, the time step used to determine the CTF series should be the same as the time step used to update the zone mean air temperature in the zone energy balance. But, as the time step used to calculate the CTF series gets shorter, the number of terms in the series grows. Eventually, with enough terms, the series becomes unstable due to truncation and round-off error. Heavy constructions, such as slab-on-grade floors (12" heavyweight concrete over 18" dirt), have accuracy and stability problems at time steps as large as 0.5 hours when modeled by Hittle's CTF method. In an attempt to overcome this problem, Hittle's method was replaced by Seem's method (1987) in IBLAST. This resulted in some improvement in stability at shorter time steps, but not enough to allow IBLAST to run at a 0.1 hour time step without restricting the types of surfaces that could be used.

Even though CTF methods require that values of the surface temperatures and fluxes be stored for only a few specific times before the current time, the temperature and flux histories are, actually, continuous functions between those discrete points. However, there is no way to calculate information at these intermediate times once a series has been initialized. The terms in the temperature and flux histories are out of phase with these points. However, they can be calculated by shifting the phase of the temperature and flux histories by only a fraction of a time step. This procedure would allow a CTF series computed for a time step Δt , to be used to compute information at times $t+\Delta t/2$, $t+\Delta t/3$, $t+\Delta t/4$, or any other arbitrary fraction of the time step, so long as the surface temperatures and flux values were still Δt apart. Several ways of doing this are described below.

The method shown in the Figure 9 maintains two sets of histories out of phase with each other. The figure shows how this would work for two sets of histories out of phase by one half of a time step. More sets of temperature and flux histories could be used, allowing the simulation time step to take on values: 1/3, 1/4, 1/5, etc., of the minimum time step allowed for the CTF calculations. The time step between inputs to the CTF series would be the

smallest convenient interval at which the CTF series is stable. This scenario is illustrated in this figure for two separate sets of temperature and flux histories. Cycling through each history, in order, allowed calculations of the zone energy balance to be performed with updated surface information at a shorter time step than one CTF history series would otherwise allow. This method required no interpolation between the series once each set of histories was initialized. However, if the smallest time step for a stable CTF series was large compared to the zone temperature update time step, significant memory was required to store all the sets of histories.

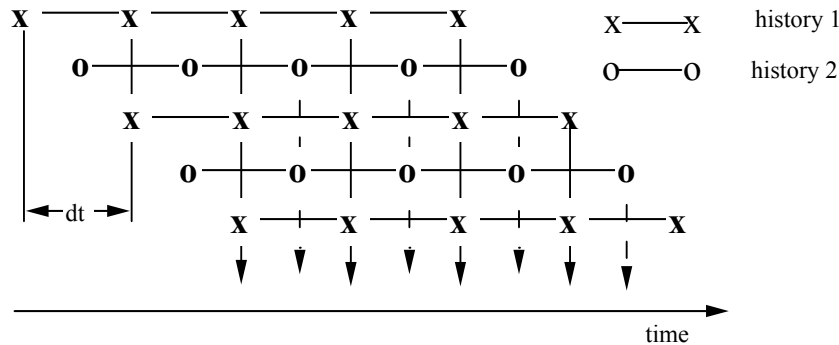


Figure 9. Multiple, staggered time history scheme

Another method is shown in Figure "Sequential interpolation of new histories" that uses successive interpolations to determine the next set of temperature and flux histories. The current history is interpolated directly from the previous history set using the required time phase shift between the two. This method required permanent storage for only one set of temperature and flux histories at a time, but smoothed out temperature and flux data as more interpolations were performed. As a result, at concurrent simulation times current values of history terms were different from previous "in phase" history terms. This was unacceptable from a physical point of view, because it allowed current information to change data from a previous time.

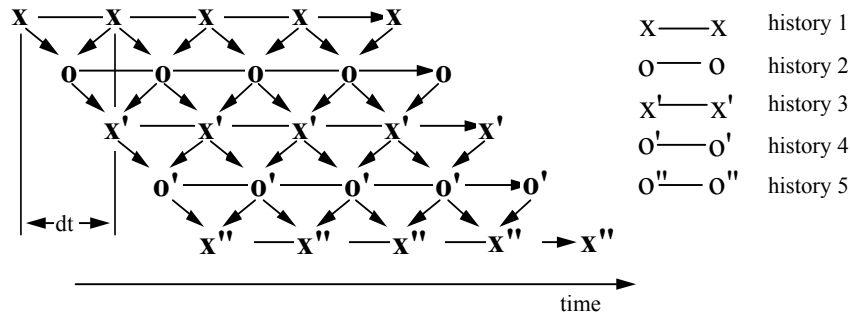


Figure 10. Sequential interpolation of new histories

A final method, shown in Figure "Master history with interpolation", was something of a hybrid of the previous two methods. One "master" history set was maintained and updated for all time; this solved the problem of current events propagating information backwards in time. When surface fluxes needed to be calculated at times out of phase with this master history a new, temporary history was interpolated from the master values. This method proved to be the best of the three options described because it eliminated propagation of information backwards in time and only required concurrent storage of two sets of temperature and flux histories. This method was subsequently incorporated into the IBLAST program in conjunction with Seem's procedure for calculating the coefficients of the CTF series.

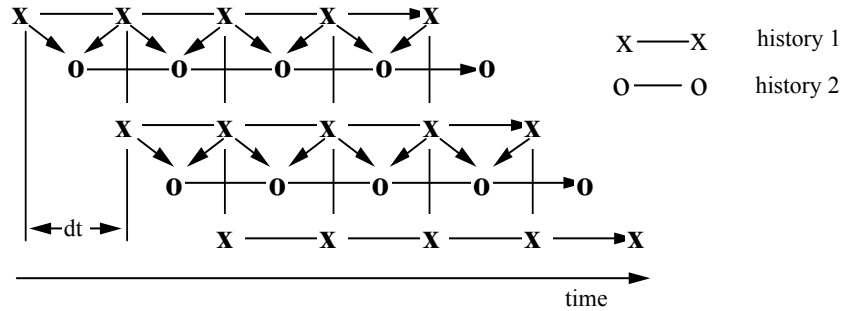


Figure 11. Master history with interpolation

References

- Ceylan, H. T., and G. E. Myers. 1980. Long-time Solutions to Heat Conduction Transients with Time-Dependent Inputs. *ASME Journal of Heat Transfer*, Volume 102, No. 1, pp. 115-120.
- Hittle, D. C. 1979. Calculating Building Heating and Cooling Loads Using the Frequency Response of Multilayered Slabs, Ph.D. Thesis, University of Illinois, Urbana, IL.
- Hittle, D. C., and R. Bishop. 1983. An Improved Root-Finding Procedure for Use in Calculating Transient Heat Flow Through Multilayered Slabs. *International Journal of Heat and Mass Transfer*, Vol. 26, No. 11, pp. 1685-1693.
- Ouyang, K., and F. Haghghat. 1991. A Procedure for Calculating Thermal Response Factors of Multi-layered Walls--State Space mMethod. *Building and Environment*, Vol. 26, No. 2, pp. 173-177.
- Seem, J. E. 1987. Modeling of Heat Transfer in Buildings, Ph.D. Thesis, University of Wisconsin, Madison, WI.
- Strand, R. K. 1995. Heat Source Transfer Functions and Their Application to Low Temperature Radiant Heating Systems, Ph.D. Thesis, University of Illinois, Urbana, IL.
- Taylor, R. D., C.O. Pedersen, D.E. Fisher, R. J. Liesen, L.K. Lawrie. 1990. *Simultaneous Simulation of Buildings and Mechanical Systems in Heat Balance Based Energy Analysis Programs*, Proceedings of the 3rd International Conference on System Simulation in Buildings, Liege, Belgium, December 3-5, 1990.
- Taylor, R.D., C.O. Pedersen, D.E. Fisher, R. J. Liesen, L.K. Lawrie. 1991. Impact of Simultaneous Simulation of Buildings and Mechanical Systems in Heat Balance Based Energy Analysis Programs on System Response and Control, Conference Proceedings IBPSA Building Simulation '91, Nice, France, August 20-22, 1991.

Conduction Finite Difference Solution Algorithm

EnergyPlus models follow fundamental heat balance principles very closely in almost all aspects of the program. However, the simulation of building surface constructions has relied on a transfer function transformation carried over from BLAST. This has all the usual restrictions of a transformation-based solution: constant properties, and fixed values of some parameters. As the energy analysis field moves toward simulating more advanced constructions, such as phase change materials (PCM), it becomes necessary to step back from transformations to more fundamental forms. Accordingly, a conduction finite difference solution algorithm has been incorporated into EnergyPlus. This does not replace the CTF solution algorithm, but complements it for cases where the user needs to simulate phase change materials or variable thermal conductivity. It is also possible to use the finite difference algorithm for zone time steps as short as one minute, corresponding to the system time step.

The algorithm uses an implicit finite difference scheme coupled with an enthalpy-temperature function to account for phase change energy accurately. The implicit formulation for an internal node is shown in equation (32).

$$\frac{\rho c_p \Delta x (T_{i,new} - T_{i,old})}{\Delta t} = \frac{k(T_{i-1,new} - T_{i,new})}{\Delta x} + \frac{k(T_{i+1,new} - T_{i,new})}{\Delta x} \quad (32)$$

Where: T is node temperature

Subscripts:

i = node being modeled

i+1 = adjacent node to interior of construction

i-1 = adjacent node to exterior of construction

,new = new temperature at end of time step

,old = temperature at end of previous time step

Δt = calculation time step

Δx = finite difference layer thickness (always less than construction layer thickness)

C_p = specific heat of material

ρ = density of material

Then, this equation is accompanied by a second equation that relates enthalpy and temperature.

$$h_i = HTF(T_i) \quad (33)$$

where HTF is an enthalpy-temperature function that uses user input data.

Equations such as (32) and (33) are formed for all nodes in a construction. It turns out that there are four types: external surface nodes, internal surface nodes, internal nodes such as shown above, and then nodes occurring at the material interfaces. The formulation of all node types is basically the same. Because the solution is implicit, a Gauss-Seidel iteration scheme is used to update to the new node temperatures in the construction. Because of this, the node enthalpies get updated each iteration, and then they are used to develop a variable C_p if a phase change material is being simulated. This is done by including a third equation for C_p .

$$C_p = \frac{h_{i,new} - h_{i,old}}{T_{i,new} - T_{i,old}} \quad (34)$$

The iteration scheme assures that the correct enthalpy, and therefore the correct C_p is used in each time step, and the enthalpy of the material is accounted for accurately. Of course, if the material is regular, the user input constant C_p is used.

The algorithm also has a provision for including a temperature coefficient to modify the thermal conductivity. The thermal conductivity is obtained from:

$$k = k_o + k_1(T_i - 20) \quad (35)$$

where:

k_o is the 20°C value of thermal conductivity(normal idf input)

k_1 is the change in conductivity per degree temperature difference from 20°C

These additional property information values are put into the input file as explained in the Input/Output Reference Document, but it consists simply of a value for k_1 and set of enthalpy temperature pairs that describe the enthalpy of the phase change material in straight line segments with respect to temperature.

A graph showing the effect of a large PCM on the outside surface of a zone is shown below. The phase change temperature was 30°C, and the flat temperature response during the phase change is obvious. This example was run with a zone time step of one minute to show that such small time steps can be done with the finite difference solution technique. It is more efficient to set the zone time step shorter than those used for the CTF solution algorithm. It should be set to 12 time steps per hour or greater, and can range up to 60. The finite difference algorithm actually works better with shorter zone time steps. The computation time has a minimum at a zone time step around two minutes (30 time steps/hr), and increases for shorter or longer zone time steps.

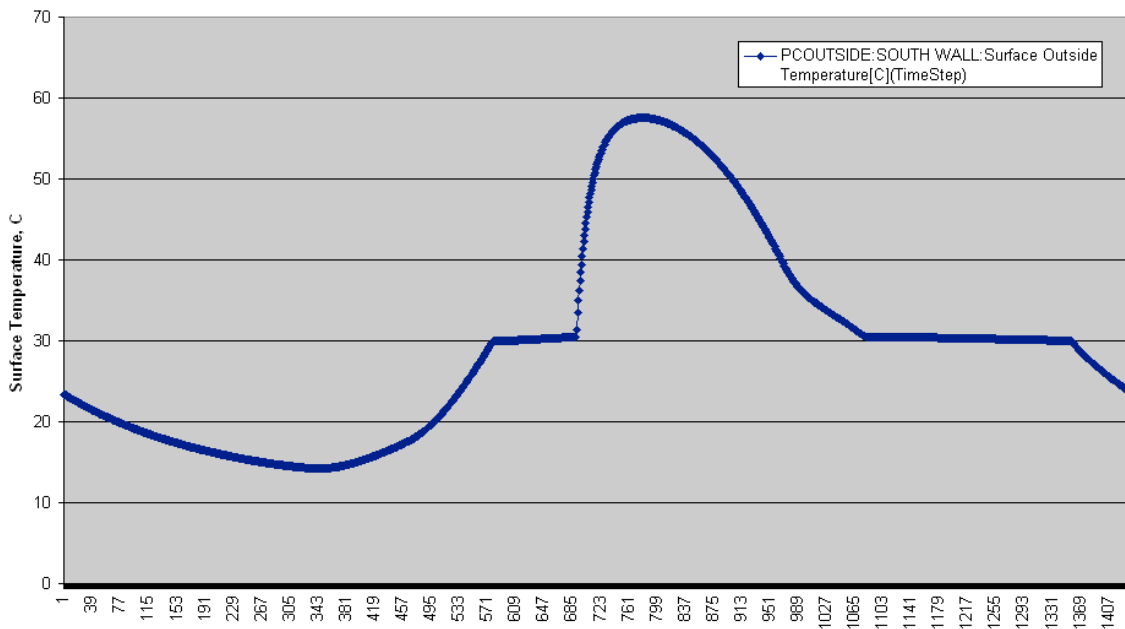


Figure 12. Effects of Large PCM on Outside Zone Surface

Combined Heat and Moisture Transfer (HAMT) Model

Overview

The combined heat and moisture transfer finite (HAMT) solution algorithm is a completely coupled, one-dimensional, finite element, heat and moisture transfer model simulating the movement and storage of heat and moisture in surfaces simultaneously from and to both the internal and external environments. As well as simulating the effects of moisture buffering, HAMT is also able to provide temperature and moisture profiles through composite building walls, and help to identify surfaces with high surface humidity.

HAMT Nomenclature

Dependencies on moisture content are indicated by a superscript ^w, on heat by a superscript ^h and vapor pressure by a superscript ^v.

Table 2. Combined Heat and Moisture Transfer Model Nomenclature

Symbol	Units	Meaning
T	°C	Temperature
RH, ϕ	%, fraction	Relative humidity
W	kg/m ³	Moisture Content
$\frac{\partial H}{\partial T}$	J/m ³ C	Moisture dependent heat storage capacity
$\frac{\partial w}{\partial \phi}$	kg/m ³	Moisture dependent moisture storage capacity
k^w	W/mC	Moisture dependent thermal conductivity
h_v	J/kg	Evaporation enthalpy of water (= 2, 489, 000J/kg)
δ	kg/msPa	Vapor diffusion coefficient in air
μ	-	Moisture dependent vapor diffusion resistance factor
P	Pa	Vapor pressure
P_{ambient}	Pa	Ambient air pressure
C	J/kgC	Specific heat capacity of dry material
c^w	J/KgC	Specific heat capacity of water (=4,180J/kg°C@ 20°C)
ρ	kg/m ³	Material Density
ρ^w	kg/m ³	Density of water (= 1000kg/m ³)
D^w	m ² /s	Liquid Transport Coefficient
A	m ²	Contact Surface area
ΔV_i	m ³	Cell Volume
t	s	Time
$\Delta \tau$	s	Time step between calculations
x	m	Distance between cell centres
C_i^h	J/C	Heat Capacitance of cell i
C_i^w	kg	Moisture Capacitance of cell i
R_{ij}^h	C/W	Heat Resistance between cells i and j
R_{ij}^v	sPa/kg	Vapor Resistance between cells i and j
R_{ij}^w	s/kg	Liquid Moisture Resistance between cells i and j
q_i^v	W	Heat due to Vaporisation
q_i^{adds}	W	Heat from additional Sources
P	m ³ /m ³	Material Porosity
p(as a superscript)	s	Present Time Step
i,j	-	Cell indices

HAMT Model Description

Equations (36) and (37) are derived from heat and moisture balance equations and are taken from [Künzel, H.M. (1995)]. They describe a theoretical model for the transfer of heat and moisture through a material.

$$\frac{\partial H}{\partial T} \frac{\partial T}{\partial \tau} = \frac{\partial}{\partial x} \left(k^w \frac{\partial T}{\partial x} \right) + h_v \frac{\partial}{\partial x} \left(\frac{\delta}{\mu} \frac{\partial T}{\partial x} \right) \quad (36)$$

The three terms in equation (36) describe the storage, transport and generation of heat respectively.

$$\frac{\partial w}{\partial \phi} \frac{\partial \phi}{\partial \tau} = \frac{\partial}{\partial x} \left(D^w \frac{\partial w}{\partial \phi} \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial x} \left(\frac{\delta}{\mu} \frac{\partial T}{\partial x} \right) \quad (37)$$

The three terms in equation (37) describe the storage of moisture, the transport of liquid moisture and the transport of vapor respectively. The equation to calculate the vapor diffusion coefficient in air (δ) used in the third term of both equations, is also taken from Künzel,

$$\delta = \frac{(2 \times 10^{-7} \times (T + 273.15)^{0.81})}{P_{ambient}} \quad (38)$$

The heat storage capacity ($\partial H / \partial T$) depends on the moisture content w of the material by the following equation.

$$\frac{\partial H}{\partial T} = (c\rho + c^w w) \quad (39)$$

The moisture content of the material w and the vapor diffusion resistance factor μ depend on the relative humidity inside the material. The parameters $\partial w / \partial \phi$, k^w and D^w are also moisture dependent parameters.

The following sections describe how the above equations are used within the HAMT model.

Surfaces, Material Layers and Cells

“Surfaces” are made of a number of layers of potentially any combination of materials. Each surface is split into its constituent materials and is then split up further into cells through its depth. HAMT will generate no more than 10 cells per material with widths that are thinner near the boundaries of each material where most changes are expected and detail is needed.

Heat Transfer

Equation 1 can be re-written and used to describe the heat storage and transfer through the i^{th} cell in a surface.

$$(c_i \rho_i + c^w w_i) \Delta V_i \frac{T_i^{p+1} - T_i^p}{\Delta \tau} = \sum_j k_{ij}^w A_{ij} \frac{T_j^{p+1} - T_i^{p+1}}{x_{ij}} + \sum_j h_v \frac{\delta_{ij}}{\mu_{ij}} A_{ij} \frac{p_j^{p+1} - p_i^{p+1}}{x_{ij}} \quad (40)$$

In the one dimensional case there are only two adjacent cells each labelled j. The heat generated due to vaporisation q_i^v can be calculated separately.

$$q_i^v = \sum_j h_v \frac{\delta_{ij}}{\mu_{ij}} A_{ij} \frac{p_j^{p+1} - p_i^{p+1}}{x_{ij}} \quad (41)$$

Rearranging equation (36) and including other sources of heat (q_i^{adds}) such as radiation from other surfaces in the calculation gives the temperature in a cell in the next time step as,

$$T_i^{p+1} = \frac{\sum_j \frac{T_j^{p+1}}{R_{ij}^h} + q_i^v + q_i^{adds} + C_i^h \frac{T_i^p}{\Delta \tau}}{\frac{C_i^h}{\Delta \tau} + \sum_j \frac{1}{R_{ij}^h}} \quad (42)$$

where $C_i^h = (c_i \rho_i + c^w w_i) \Delta V_i$ is thermal heat capacitance of cell i and $R_{ij}^h = \frac{x_{ij}}{k_{ij} A_{ij}}$ is the thermal resistance between cells i and j.

This equation can be solved using the Gauss-Seidel iteration technique. The i^{th} cell temperature is calculated whilst the j^{th} cell temperatures are kept as up to date as possible. The iteration is stopped when the maximum difference between two consecutive calculations in all cells is less than a threshold of 0.002°C.

Moisture Content w

The moisture content (w) of a cell is needed for the calculation of the heat transfer through the cell as it affects the thermal resistance and heat capacitance. The moisture content of cells is calculated from the relative humidity (RH) of the material. The relationship between w and the RH for each material is known as the sorption isotherm and measured data points are entered into EnergyPlus as a series of coordinates. HAMT interpolates between the measurements to obtain the moisture content of a material for any RH value. The sorption isotherm input is via the MaterialProperty:HeatAndMoistureTransfer:SorptionIsotherm object and is described in the Input Output Reference document.

Porosity P

The porosity of a material (P) is an input variable and defined as the maximum fraction, by volume, of a material that can be taken up with moisture. It is used to calculate the maximum point on the sorption isotherm curve. The porosity is entered for each material via the MaterialProperty:HeatAndMoistureTransfer:Settings object, as described in the Input Output Reference document.

Moisture Dependant Thermal Conductivity k^w

The thermal conductivity (k^w) of the cell is determined by interpolating between data points of thermal conductivity versus the moisture content of the material, entered into EnergyPlus via the MaterialProperty:HeatAndMoistureTransfer:ThermalConductivity object. The moisture content is determined via the sorption isotherm which gives the moisture content as a function of Relative Humidity.

Moisture Dependant Moisture Diffusion Coefficient μ

This is used in the third term of equation (36) to describe the heat transfer due to vapor movement. It is determined by interpolating between data points of moisture diffusion

coefficient versus the moisture content of the material, entered into EnergyPlus via the MaterialProperty:HeatAndMoistureTransfer:Diffusion object. A simple linear interpolation is used to obtain the conductivity between measured points.

Moisture Transfer

Moisture, as well as heat, is transported through materials as either liquid (w) or vapor (p). There are two different potentials that control the movement through the material. Liquid transfer is driven by differences in relative humidity whereas vapor transfer is driven by differences in vapor pressure. Materials also have a capacity to store moisture. Equation (37) can be re-written for a discrete cell in a continuous material.

$$\frac{dw}{d\phi_i} \Delta V_i \frac{\phi_i^{p+1} - \phi_i^p}{\Delta \tau} = \sum_j k_{ij} A_{ij} \frac{\phi_j^{p+1} - \phi_i^{p+1}}{x_{ij}} + \sum_j \frac{\delta_{ij}}{\mu_{ij}} A_{ij} \frac{p_j^{p+1} - p_i^{p+1}}{x_{ij}} \quad (43)$$

Equation (43) can be rearranged to provide the relative humidity of the i^{th} cell in the next time step.

$$\phi_i^{p+1} = \frac{\sum_j \frac{\phi_j^{p+1}}{R_{ij}^w} + \sum_j \frac{p_i^{p+1}}{R_{ij}^v} + C_i^w \frac{\phi_i^p}{\Delta \tau}}{\frac{C_i^w}{\Delta \tau} + \sum_j \frac{1}{R_{ij}^w} + \sum_j \frac{p_i^{sat}}{R_{ij}^v}} \quad (44)$$

where $C_i^w = \frac{dw}{d\phi_i} \Delta V_i$ is the “Moisture Capacitance” of cell i ,

$$R_{ij}^w = \frac{x_{ij}}{A_{ij} D_{ij}^w \frac{dw}{d\phi}} \quad (45)$$

is the moisture resistance between cells i and j and $R_{ij}^v = \frac{\mu_{ij} x_{ij}}{A_{ij} \delta_{ij}}$ is the vapor resistance between cells i and j .

Equation (44) can be used together with the heat equation (42) in an alternate step by step fashion to calculate the new temperature and relative humidity profiles for each cell for the next time step.

Liquid Transport Coefficient D^w

The Moisture Dependant Liquid Transport Coefficient is entered as a series of moisture density and liquid transport coefficient data points. There are two different coefficients, one for suction, where the surface is wet due to rain, and one for redistribution where the surface is no longer wet. If the weather file has a rain flag it is used to switch between these two types of coefficient. HAMT-SUCTION and HAMT-REDISTRIBUTION.

Moisture Dependent Moisture Capacity $\frac{\partial w}{\partial \phi}$

This is simply the gradient of moisture sorption isotherm at the RH of the material.

Convective Heat Transfer

The internal and external heat transfer coefficients are used to calculate the thermal resistance of the boundary layer between the zone air and the surface of the surface. They are either supplied by the user via the advanced surface concepts object "SurfaceProperty:ConvectionCoefficients" or, if these are not provided, dynamic values are calculated.

Convective Vapor Transfer

The internal and external vapor transfer coefficients are used to calculate the resistance to vapour transfer of the boundary layer between the zone air and the surface of the surface. They are also either supplied by the user via the advanced surface concept object SurfaceProperties:VaporCoefficients. If these are not provided then dynamic values are calculated based on the convective heat transfer coefficients.

Initial Moisture Content

At the start of an EnergyPlus simulation "warm up" days are used to ensure that the temperatures of surfaces come to equilibrium with the environment before the simulation starts proper. Moisture content within some building fabrics can take a very long time to come to equilibrium with its environment and it is therefore necessary to set initial or typical values of moisture content for each material to be used at the start of the simulation. These initial values are entered for each material via the MaterialProperty:HeatAndMoistureTransfer:Settings object as described in the Input Output Reference document.

Using the Model

As an illustration of the use of the Heat and Moisture Transfer (HAMT) model, the material properties for a small sample of six generic materials have been provided in the EnergyPlus Reference DataSets (MoistureMaterials.idf). The properties were synthesised from the Annex 24 database [Kumar Kumaran, M. (1996)], supplemented, when required, by data from the database of the WUFI model [WUFI (1999)] and are therefore not related to any unique, measured material. Users should consult material property catalogues and other primary sources when the properties of a specific material are required.

Moisture and heat from the surfaces are used by EnergyPlus to calculate the room air temperature and moisture content. EnergyPlus with HAMT works best with as short a time step as possible. However the optimum time step which gives a good prediction for a short computing time will very much depend on the nature of the weather and type of building. Buildings with frequent and large changes in internal and external temperature will need a small time step, maybe even 60 steps per hour. Slowly evolving temperatures and relative humidity's will not require such a short time step and 20, or even 6, steps per hour may be sufficient.

References

- Künzel, H.M. (1995) Simultaneous Heat and Moisture Transport in Building Components. One- and two-dimensional calculation using simple parameters. IRB Verlag 1995
- Holman, J.P. (2002) Heat Transfer, Ninth Edition. McGraw-Hill
- Winterton, R.H.S. (1997) Heat Transfer. (Oxford Chemistry Primers; 50) Oxford University Press
- Kumar Kumaran, M. (1996) IEA ANNEX 24, Final Report, Volume 3
- WUFI (1999) version 2.2 Simultaneous Heat and Moisture Transport in Building components. Fraunhofer IBP, Holzkirchen, Germany

Effective Moisture Penetration Depth (EMPD) Model

Overview

Moisture has little effect on heating system performance, but a profound effect on the performance of air conditioning systems. In order to accurately describe building performance during periods when cooling is needed, it is very important to know the moisture conditions of the building. If one assumes that all building moisture is contained in the room air, then one ignores the fact that the materials that bound the room (e.g. wall surfaces, furnishings, linens, etc.) store and release moisture. Thus, to assume that the only moisture that effects cooling system performance is contained in the room air is a false, and it can lead to significant error in the prediction of room moisture conditions and cooling system loads.

The EMPD (Effective Moisture Penetration Depth) model is a simplified, lumped approach to simulate surface moisture adsorption and desorption.

EMPD Model Description

The EMPD concept assumes that a thin layer (δ_M) close to the wall surface behaves dynamically and exchanges moisture with the air domain when exposed to cyclic air moisture pulses. For short periods where the cyclic integral of the total moisture adsorption and desorption is near zero (i.e. there is no net moisture storage), the EMPD concept has been shown to be a reasonable approximation of reality (Kerestecioglu et al, 1989). In other words, the following constraint must be met:

$$\int_{\tau_1}^{\tau_2} \frac{dU}{d\tau} d\tau = 0 \quad (46)$$

where, $\tau_2 - \tau_1$ denotes the finite time interval over which the equation holds. The EMPD model assumes no spatial distribution of moisture content across the thickness (L) of the solid; rather, a thin layer (δ_M) of uniform moisture content (U) is assumed to represent the total moisture content of the solid. This may be mathematically stated as:

$$\int_0^L U(x) dx = U \delta_M \quad (47)$$

For most building materials, the equilibrium moisture sorption isotherm can be defined by the following general equation (Kerestecioglu et al. 1988):

$$U = a\varphi^b + c\varphi^d \quad (48)$$

where

$$\varphi \approx \frac{W^*}{W_{sat}^*} \quad (49)$$

and

$$W_{sat}^* = \frac{1}{R_v \rho_a T^*} \exp\left(23.7093 - \frac{4111}{T^* - 35.45}\right) \quad (50)$$

Given that $U=U(W^*, T^*)$, the moisture content may be differentiated with respect to time in the following manner:

$$\frac{du}{d\tau} = \frac{\partial U}{\partial W^*} \frac{dW^*}{d\tau} + \frac{\partial U}{\partial T^*} \frac{dT^*}{d\tau} = A_T \frac{dW^*}{d\tau} - B_\rho \frac{dT^*}{d\tau} \quad (51)$$

where A_T and B_ρ are the isothermal moisture capacity and thermo-gradient coefficient, respectively. From Eqs. (49), (49) and (50), they can be expressed as:

$$A_T = \frac{ab\phi^b + cd\phi^d}{W^*} \quad (52)$$

and

$$B_\rho = - \left[\frac{1}{T^*} - \frac{4111}{(T^* - 35.45)^2} \right] * (ab\phi^b + cd\phi^d) \quad (53)$$

The lumped mass transfer equation for the i-th solid domain may be written as

$$(A\rho_b\delta_M)_i \frac{dU_i}{d\tau} = h_{M,i}A_i(W_r - W_i^*) \quad (54)$$

Using Eqs. (51), (52), (53) and (54), one obtains the final equation needed for closure moisture transfer at internal surface.

$$(A_i\rho_b\delta_M A_T)_i \frac{dW_i^*}{d\tau} = h_{M,i}A_i(W_r - W_i^*) + (A\rho_b\delta_M B_\rho)_i \frac{dT_i^*}{d\tau} \quad (55)$$

The energy equation for the envelope contains the surface temperature and is given by the conduction equation

$$\rho C_p \frac{dT}{d\tau} = \nabla \cdot (k\nabla T) \quad (56)$$

with the boundary conditions at interior surface

$$-k\nabla T = -q_T'' + h_r(T^* - T_r) + \lambda h_M(W^* - W_r) \quad (57)$$

A more detailed account of the numerical solution procedure can be found in Kerestecioglu et al. (1988).

EMPD Value Determination

An effective moisture penetration depth may be determined from either experimental or detailed simulation data by using actual surface areas and moisture vapor diffusivity. An empirical function derived from the detailed simulation may be used to determine the EMPD value (Kerestecioglu et al, 1989):

$$\delta_M = 12.567024 - 12.21373 * \exp(-267.0211 * D_v^{0.7} * \xi^{-0.7}) \quad (58)$$

where

$$\xi = \left| \frac{\Delta\phi}{\Delta\tau} \right| \quad (59)$$

Figure 13 gives the EMPD values to be used for various vapor diffusivities evaluated at different ambient excitations.

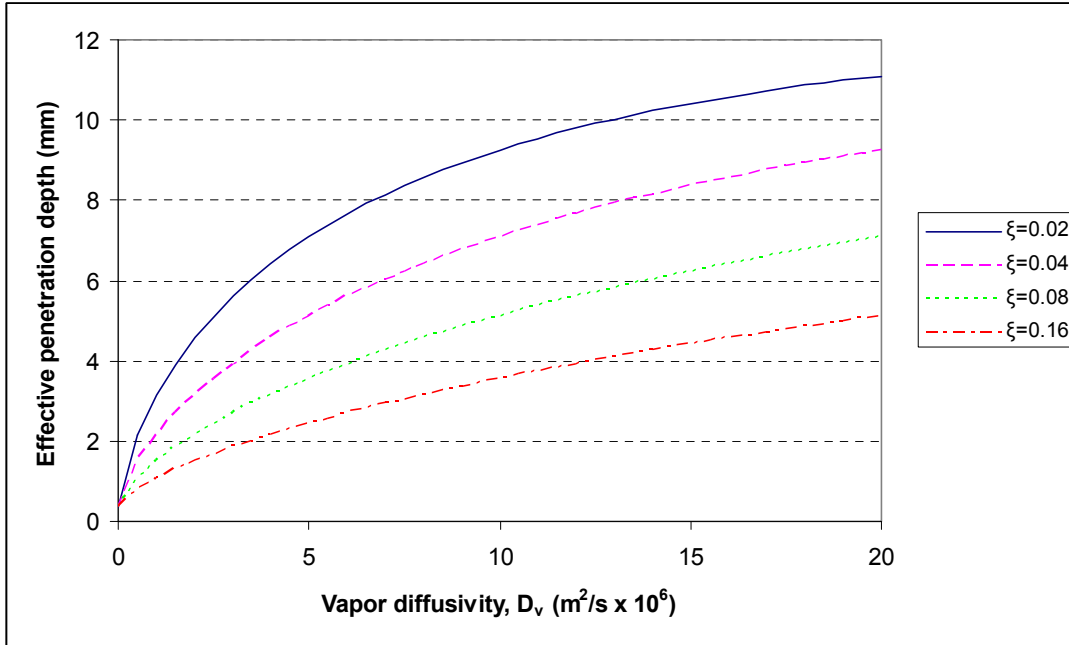


Figure 13. Limit of Effective Penetration Depth Values for Various Vapor Diffusivities at Different Ambient Excitations.

EMPD Nomenclature

A	= Area [m^2]
A_T	= Isothermal moisture capacity [m^3/kg]
B_p	= Thermo-gradient coefficient [$kg/kg-K$]
C_p	= Specific heat [$J/kg.K$]
h_M	= Convective mass transfer coeff. [kg/m^2-s]
h_T	= Convective heat transfer coeff. [W/m^2-K]
k	= Thermal conductivity [$W/m-K$]
L	= Length [m]
q''_T	= Imposed heat flux [W/m^2]
R_v	= Ideal gas constant [$461.52 J/kg-K$]
T	= Temperature [K]
U	= Moisture content [kg/kg]
W	= Humidity ratio [kg/kg]

Greek letters

δ_M	= Effective penetration depth for moisture equation [m]
λ	= Heat of vaporization [J/kg]
ρ	= Density [kg/m^3]
τ	= Time [s]

ϕ = Relative humidity [0 to 1]
 ξ = Ambient moisture excitation rate [1/h]

Subscripts and superscripts

a = Air
 b = Bulk
 * = Surface
 i = i-th surface

References

Kerestecioglu, A., Swami, M., Dabir, R., Razzaq, N., and Fairey, P., 1988, "Theoretical and Computational Investigation of Algorithms for Simultaneous Heat and Moisture Transport in Buildings," FSEC-CR-191-88, Florida Solar Energy Center, Cape Canaveral, FL.

Kerestecioglu, A., M. Swami and A. Kamel, 1989, "Theoretical and Computational Investigation of Simultaneous Heat and Moisture Transfer in Buildings: Effective Penetration Depth Theory." ASHRAE Winter Meeting, Atlanta, GA.

Kerestecioglu, A., M. V. Swami, P. Brahma, L. Gu, P. Fairey, and S. Chandra, 1989, "FSEC 1.1 User's Manual," Florida Solar Energy Center, Cape Canaveral, FL

Outside Surface Heat Balance

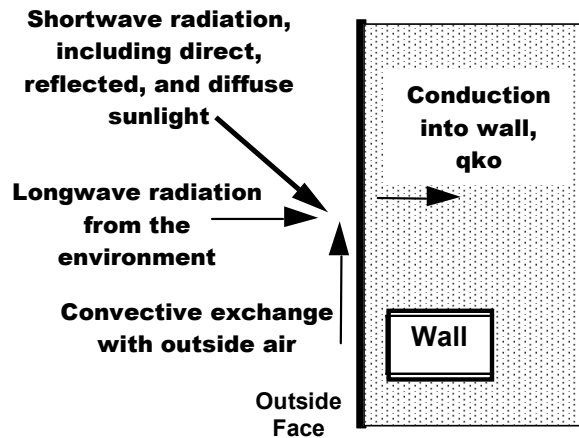


Figure 14. Outside Heat Balance Control Volume Diagram

The heat balance on the outside face is:

$$q''_{\alpha sol} + q''_{LWR} + q''_{conv} - q''_{ko} = 0 \quad (60)$$

where:

$q''_{\alpha sol}$ = Absorbed direct and diffuse solar (short wavelength) radiation heat flux.

q''_{LWR} = Net long wavelength (thermal) radiation flux exchange with the air and surroundings.

q''_{conv} = Convective flux exchange with outside air.

q''_{ko} = Conduction heat flux (q/A) into the wall.

All terms are positive for net flux to the face except the conduction term, which is traditionally taken to be positive in the direction from outside to inside of the wall. Simplified procedures generally combine the first three terms by using the concept of a *sol-air temperature*. Each of these heat balance components is introduced briefly below.

External Shortwave Radiation

$q''_{\alpha sol}$ is calculated using procedures presented later in this manual and includes both direct and diffuse incident solar radiation absorbed by the surface face. This is influenced by location, surface facing angle and tilt, surface face material properties, weather conditions, etc.

External Longwave Radiation

q''_{LWR} is a standard radiation exchange formulation between the surface, the sky, and the ground. The radiation heat flux is calculated from the surface absorptivity, surface temperature, sky and ground temperatures, and sky and ground view factors.

The longwave radiation heat exchange between surfaces is dependent on surface temperatures, spatial relationships between surfaces and surroundings, and material properties of the surfaces. The relevant material properties of the surface, emissivity ε and absorptivity α , are complex functions of temperature, angle, and wavelength for each participating surface. However, it is generally agreed that reasonable assumptions for building loads calculations are (Chapman 1984; Lienhard 1981):

- each surface emits or reflects diffusely and is gray and opaque ($\alpha = \varepsilon$, $\tau = 0$, $\rho = 1 - \varepsilon$)
- each surface is at a uniform temperature
- energy flux leaving a surface is evenly distributed across the surface,
- the medium within the enclosure is non-participating.

These assumptions are frequently used in all but the most critical engineering applications.

Table 3. Nomenclature List of Variables.

Mathematical variable	Description	Units	Range
q''_{LWR}	Exterior surface long wave radiation flux	W/m ²	-
h_r	Linearized radiative heat transfer coefficient to air temperature	W/(m ² K)	-
T_{surf}	Surface Outside face temperatures	K	-
T_{air}	Outside air temperature	K	-
T_{gnd}	Environmental ground surface temperature	K	-
T_{sky}	Sky Effective temperature	K	-

F_{gnd}	view factor of wall surface to ground surface	-	0~1
F_{sky}	View factor of wall surface to sky	-	0~1
F_{air}	View factor of wall surface to air	-	0~1
ε	Surface long-wave emissivity	-	0~1
σ	Stefan-Boltzmann constant	W/m^2-K^4	0.000000 567

Consider an enclosure consisting of building exterior surface, surrounding ground surface, and sky. Using the assumptions above, we can determine the long wave radiative heat flux at the building exterior surface (Walton 1983; McClellan and Pedersen 1997). The total longwave radiative heat flux is the sum of components due to radiation exchange with the ground, sky, and air.

$$q''_{LWR} = q''_{gnd} + q''_{sky} + q''_{air} \quad (61)$$

Applying the Stefan-Boltzmann Law to each component yields:

$$q''_{LWR} = \varepsilon \sigma F_{gnd} (T_{surf}^4 - T_{gnd}^4) + \varepsilon \sigma F_{sky} (T_{surf}^4 - T_{sky}^4) + \varepsilon \sigma F_{air} (T_{surf}^4 - T_{air}^4) \quad (62)$$

where

ε =long-wave emittance of the surface

σ =Stefan-Boltzmann constant

F_{gnd} = view factor of wall surface to ground surface temperature

F_{sky} = view factor of wall surface to sky temperature

F_{air} =view factor of wall surface to air temperature

T_{surf} = outside surface temperature

T_{gnd} = ground surface temperature

T_{sky} = sky temperature

T_{air} = air temperature

Linearized radiative heat transfer coefficients are introduced to render the above equation more compatible with the heat balance formulation,

$$q''_{LWR} = h_{r,gnd} (T_{surf} - T_{gnd}) + h_{r,sky} (T_{surf} - T_{sky}) + h_{r,air} (T_{surf} - T_{air}) \quad (63)$$

where

$$h_{r, gnd} = \frac{\varepsilon \sigma F_{gnd} (T_{surf}^4 - T_{gnd}^4)}{T_{surf} - T_{gnd}} \quad (64)$$

$$h_{r, sky} = \frac{\varepsilon \sigma F_{sky} (T_{surf}^4 - T_{sky}^4)}{T_{surf} - T_{sky}} \quad (65)$$

$$h_{r, air} = \frac{\varepsilon \sigma F_{air} (T_{surf}^4 - T_{air}^4)}{T_{surf} - T_{air}} \quad (66)$$

The longwave view factors to ground and sky are calculated with the following expressions (Walton 1983):

$$F_{gnd} = 0.5(1 + \cos \phi) \quad (67)$$

$$F_{sky} = 0.5(1 - \cos \phi) \quad (68)$$

where ϕ is the tilt angle of the surface. The view factor to the sky is further split between sky and air radiation by:

$$\beta = \sqrt{0.5(1 + \cos \phi)} \quad (69)$$

The ground surface temperature is assumed to be the same as the air temperature. The final forms of the radiative heat transfer coefficients are shown here.

$$h_{r, gnd} = \frac{\varepsilon \sigma F_{gnd} (T_{surf}^4 - T_{air}^4)}{T_{surf} - T_{air}} \quad (70)$$

$$h_{r, sky} = \frac{\varepsilon \sigma F_{sky} \beta (T_{surf}^4 - T_{sky}^4)}{T_{surf} - T_{sky}} \quad (71)$$

$$h_{r, air} = \frac{\varepsilon \sigma F_{sky} (1 - \beta) (T_{surf}^4 - T_{air}^4)}{T_{surf} - T_{air}} \quad (72)$$

References

- ASHRAE. 1993. 1993 ASHRAE Handbook – Fundamentals. Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- Chapman, A. J. 1984. Heat Transfer, 4th Edition, New York: Macmillan Publishing Company.
- Lienhard, J. H. 1981. A Heat Transfer Textbook, Englewood Cliffs, N.J.: Prentice-Hall, Inc.
- McClellan, T. M., and C. O. Pedersen. 1997. Investigation of Outside Heat Balance Models for Use in a Heat Balance Cooling Load Calculation. ASHRAE Transactions, Vol. 103, Part 2, pp. 469-484.
- Walton, G. N. 1983. Thermal Analysis Research Program Reference Manual. NBSSIR 83-2655. National Bureau of Standards.

Atmospheric Variation

All buildings are located in the troposphere, the lowest layer of the atmosphere. The troposphere extends from sea level to an altitude of 11 km. Throughout the troposphere, air temperature decreases almost linearly with altitude at a rate of approximately 1°C per 150 m. Barometric pressure decreases more slowly. Wind speed, on the other hand, increases with altitude.

Because the atmosphere changes with altitude (defined as *height above ground* in this case), tall buildings can experience significant differences in local atmospheric properties between the ground floor and the top floor. Buildings interact with the atmosphere through convective heat transfer between the outdoor air and the exterior surfaces of the building envelope, and through the exchange of air between the outside and inside of the building via infiltration and ventilation.

Impetus for using this modeling is illustrated in the next table. Using a 70 story (284 meters) building as an example, the atmospheric variables are significant.

Table 4. Atmospheric Variables at Two Different Altitudes above Ground Level

Variable	1.5 Meters	284 meters	Absolute Diff	Percent Diff
Air Temperature	15°C	13.15°C	1.85°C	12.3%
Barometric Pressure	101,325 Pa	97,960 Pa	3,365 Pa	3.3%
Wind Speed	2.46 m/s	7.75 m/s	5.29 m/s	215%

Comparing the annual energy usage between 60 discretely modeled floors of a building, it turns out that the effect due to wind speed change is dominant over the first ten floors. But at floor 25, surprisingly, the effect due to air temperature has caught up and is about equal to the effect of wind speed. Above floor 25 the effect due to air temperature is now dominant. Clearly it is desirable to model air temperature variation with altitude for high-rise buildings.

To accommodate atmospheric variation EnergyPlus automatically calculates the local outdoor air temperature and wind speed separately for each zone and surface that is exposed to the outdoor environment. The zone centroid or surface centroid are used to determine the height above ground. Only local outdoor air temperature and wind speed are currently calculated because they are important factors for the exterior convection calculation for surfaces (see Exterior Convection below) and can also be factors in the zone infiltration and ventilation calculations. Variation in barometric pressure, however, is considered when using the Airflow Network objects.

Local Outdoor Air Temperature Calculation

Variation in outdoor air temperature is calculated using the U.S. Standard Atmosphere (1976). According to this model, the relationship between air temperature and altitude in a given layer of the atmosphere is:

$$T_z = T_b + L(H_z - H_b) \quad (73)$$

where

T_z = air temperature at altitude z

T_b = air temperature at the base of the layer, i.e., ground level for the troposphere

L = air temperature gradient, equal to -0.0065 K/m in the troposphere

H_b = offset equal to zero for the troposphere

H_z = geopotential altitude.

The variable H_z is defined by:

$$H_z = \frac{Ez}{(E + z)} \quad (74)$$

where

$E = 6,356$ km, the radius of the Earth

$z =$ altitude.

For the purpose of modeling buildings in the troposphere, altitude z refers to the height above ground level, not the height above sea level. The height above ground is calculated as the height of the centroid, or area-weighted center point, for each zone and surface.

The air temperature at ground level, T_b , is derived from the weather file air temperature by inverting the equation above:

$$T_b = T_{z,met} - L \left(\frac{Ez_{met}}{E + z_{met}} - H_b \right) \quad (75)$$

where

$T_{z,met}$ = weather file air temperature (measured at the meteorological station)

z_{met} = height above ground of the air temperature sensor at the meteorological station.

The default value for z_{met} for air temperature measurement is 1.5 m above ground. This value can be overridden by using the Site:WeatherStation object.

Local Wind Speed Calculation

Chapter 16 of the Handbook of Fundamentals (ASHRAE 2005). The wind speed measured at a meteorological station is extrapolated to other altitudes with the equation:

$$V_z = V_{met} \left(\frac{\delta_{met}}{z_{met}} \right)^{\alpha_{met}} \left(\frac{z}{\delta} \right)^{\alpha} \quad (76)$$

where

$z =$ altitude, height above ground

$V_z =$ wind speed at altitude z

$\alpha =$ wind speed profile exponent at the site

$\delta =$ wind speed profile boundary layer thickness at the site

$z_{met} =$ height above ground of the wind speed sensor at the meteorological station

$V_{met} =$ wind speed measured at the meteorological station

$\alpha_{met} =$ wind speed profile exponent at the meteorological station

$\delta_{met} =$ wind speed profile boundary layer thickness at the meteorological station.

The wind speed profile coefficients α , δ , α_{met} , and δ_{met} , are variables that depend on the roughness characteristics of the surrounding terrain. Typical values for α and δ are shown in the following table:

Table 5. Wind Speed Profile Coefficients (ASHRAE Fundamentals 2005).

Terrain Description	Exponent, α	Boundary Layer Thickness, δ (m)
Flat, open country	0.14	270
Rough, wooded country	0.22	370
Towns and cities	0.33	460
Ocean	0.10	210
Urban, industrial, forest	0.22	370

The terrain types above map to the options in the *Terrain* field of the Building object. The *Terrain* field can be overridden with specific values for α and δ by using the Site:HeightVariation object.

The default value for z_{met} for wind speed measurement is 10 m above ground. The default values for α_{met} and δ_{met} are 0.14 and 270 m, respectively, because most meteorological stations are located in an open field. These values can be overridden by using the Site:WeatherStation object.

Outdoor/Exterior Convection

Heat transfer due to exterior convection is modeled using the classical formulation:

$$Q_c = h_{c,ext} A (T_{surf} - T_{air}) \quad (77)$$

where

Q_c = rate of exterior convective heat transfer

$h_{c,ext}$ = exterior convection coefficient

A = surface area

T_{surf} = surface temperature

T_{air} = outdoor air temperature

Substantial research has gone into the formulation of models for estimating the exterior convection coefficient. Since the 1930's there have been many different methods published for calculating this coefficient, with much disparity between them (Cole and Sturrock 1977; Yazdani and Klems 1994). EnergyPlus offers a choice of six algorithms: Simple, Detailed, BLAST, TARP, MoWiTT, and DOE-2. See the SurfaceConvectionAlgorithm:Outside object in the Input Output Reference document.

Note that when the outside environment indicates that it is raining, the exterior surfaces (exposed to wind) are assumed to be wet. The convection coefficient is set to a very high number (1000) and the outside temperature used for the surface will be the wet-bulb temperature. (If you choose to report this variable, you will see 1000 as its value.)

In addition to the correlation choices described below, it is also possible to override the convection coefficients on the outside of any surface by two means:

- Use the SurfaceProperty:ConvectionCoefficients object in the input file to set the convection coefficient value on either side of any surface.
- Use the SurfaceProperty:OtherSideCoefficients object in the input file to set heat transfer coefficients and temperatures on surfaces.

Both of the options can be applied using general schedules. Specific details are given in the Input Output Reference document.

Simple Algorithm

The simple algorithm uses surface roughness and local surface windspeed to calculate the exterior heat transfer coefficient. The basic equation used is:

$$h = D + EV_z + FV_z^2 \quad (78)$$

where

h = heat transfer coefficient

V_z = local wind speed calculated at the height above ground of the surface centroid

D, E, F = material roughness coefficients

The roughness correlation is taken from Figure 1, Page 22.4, ASHRAE Handbook of Fundamentals (ASHRAE 1989). The roughness coefficients are shown in the following table:

Table 6. Roughness Coefficients D, E, and F.

Roughness Index	D	E	F	Example Material
1 (Very Rough)	11.58	5.894	0.0	Stucco
2 (Rough)	12.49	4.065	0.028	Brick
3 (Medium Rough)	10.79	4.192	0.0	Concrete
4 (Medium Smooth)	8.23	4.0	-0.057	Clear pine
5 (Smooth)	10.22	3.1	0.0	Smooth Plaster
6 (Very Smooth)	8.23	3.33	-0.036	Glass

Note that the simple correlation yields a combined convection and radiation heat transfer coefficient. Radiation to sky, ground, and air is included in the exterior convection coefficient for this algorithm.

All other algorithms yield a *convection only* heat transfer coefficient. Radiation to sky, ground, and air is calculated automatically by the program.

Detailed Algorithm

Table 7. Nomenclature List of Variables.

Variable	Description	Units	Range
A	Surface area of the surface	m ²	≠ 0
h_c	Surface exterior convective heat transfer coefficient	W/(m ² K)	-
h_f	Forced convective heat transfer coefficient	W/(m ² K)	-
h_n	Natural convective heat transfer coefficient	W/(m ² K)	-
P	Perimeter of surface	m	-
R_f	Surface roughness multiplier	-	-
T_{air}	Local outdoor air temperature calculated at the height above ground of the surface centroid	°C	-
T_{so}	Outside surface temperature	°C	-
ΔT	Temperature difference between the surface and air,	°C	-
V_z	Local wind speed calculated at the height above ground of the surface centroid	m/s	-
W_f	Wind direction modifier	-	-
ϕ	Angle between the ground outward normal and the surface outward normal	degree	-
Roughness Index	Surface roughness index (6=very smooth, 5=smooth, 4=medium smooth, 3=medium rough, 2=rough, 1=very rough)	-	1~6

The Detailed, BLAST, and TARP convection models are very similar. In all three models, convection is split into forced and natural components (Walton 1981). The total convection coefficient is the sum of these components.

$$h_c = h_f + h_n \quad (79)$$

The forced convection component is based on a correlation by Sparrow, Ramsey, and Mass (1979):

$$h_f = 2.537W_f R_f \left(\frac{PV_z}{A} \right)^{1/2} \quad (80)$$

where

$$W_f = 1.0 \text{ for windward surfaces}$$

or

$$W_f = 0.5 \text{ for leeward surfaces} \quad (81)$$

Leeward is defined as greater than 100 degrees from normal incidence (Walton 1981).

The surface roughness multiplier R_f is based on the ASHRAE graph of surface conductance (ASHRAE 1981) and may be obtained from the following table:

Table 8. Surface Roughness Multipliers (Walton 1981).

Roughness Index	R_f	Example Material
1 (Very Rough)	2.17	Stucco
2 (Rough)	1.67	Brick
3 (Medium Rough)	1.52	Concrete
4 (Medium Smooth)	1.13	Clear pine
5 (Smooth)	1.11	Smooth Plaster
6 (Very Smooth)	1.00	Glass

The natural convection component h_n is calculated in the same way as the interior "Detailed" model. The detailed natural convection model correlates the convective heat transfer coefficient to the surface orientation and the difference between the surface and zone air temperatures (where $\Delta T = \text{Surface Temperature} - \text{Air Temperature}$). The algorithm is taken directly from Walton (1983). Walton derived his algorithm from the ASHRAE Handbook (2001), Table 5 on p. 3.12, which gives equations for natural convection heat transfer coefficients in the turbulent range for large, vertical plates and for large, horizontal plates facing upward when heated (or downward when cooled). A note in the text also gives an approximation for large, horizontal plates facing downward when heated (or upward when cooled) recommending that it should be half of the facing upward value. Walton adds a curve fit as a function of the cosine of the tilt angle to provide intermediate values between vertical and horizontal. The curve fit values at the extremes match the ASHRAE values very well.

For no temperature difference OR a vertical surface the following correlation is used:

$$h = 1.31 |\Delta T|^{1/3} \quad (82)$$

For ($\Delta T < 0.0$ AND an upward facing surface) OR ($\Delta T > 0.0$ AND an downward facing surface) an enhanced convection correlation is used:

$$h = \frac{9.482 |\Delta T|^{1/3}}{7.283 - |\cos \Sigma|} \quad (83)$$

where Σ is the surface tilt angle.

For ($\Delta T > 0.0$ AND an upward facing surface) OR ($\Delta T < 0.0$ AND an downward facing surface) a reduced convection correlation is used:

$$h = \frac{1.810|\Delta T|^{\frac{1}{3}}}{1.382 + |\cos \Sigma|} \quad (84)$$

where Σ is the surface tilt angle.

BLAST Algorithm

As of version 1.2.1, the BLAST algorithm is identical to the ASHRAE Detailed algorithm. Previously the only difference was that the assumed height of the meteorological wind speed measurement was 9.14 m instead of 10 m.

TARP Algorithm

As of version 1.3.0, the TARP algorithm is identical to the ASHRAE Detailed algorithm. Previously the only difference was in the calculation of the wind speed profile.

MoWiTT Algorithm

Table 9. Nomenclature List of Variables.

Variable	Description	Units	Range
A	Constant	W/(m ² K(m/s) ^b)	-
B	Constant	-	-
C _t	Turbulent natural convection constant	W/(m ² K ^{4/3})	-
h _c	Surface exterior convective heat transfer coefficient	W/(m ² K)	-
T _{so}	Outside surface temperature	°C/K	-
ΔT	Temperature difference between the surface and air	°C/K	-

The MoWiTT model is based on measurements taken at the Mobile Window Thermal Test (MoWiTT) facility (Yazdanian and Klems 1994). The correlation applies to very smooth, vertical surfaces (e.g. window glass) in low-rise buildings and has the form:

$$h_c = \sqrt{\left[C_t (\Delta T)^{\frac{1}{3}} \right]^2 + \left[a V_z^b \right]^2} \quad (85)$$

Constants a, b and turbulent natural convection constant C_t are given in Table 10.

NOTE: The MoWiTT algorithm may not be appropriate for rough surfaces, high-rise surfaces, or surfaces that employ movable insulation.

Table 10. MoWiTT Coefficients (Yazdanian and Klems 1994).

Wind Direction	C_t	a	b
(Units)	$W/m^2K^{4/3}$	$W/m^2K(m/s)^b$	-
Windward	0.84	2.38	0.89
Leeward	0.84	2.86	0.617

DOE-2 Algorithm

Table 11. Nomenclature List of Variables.

Variable	Description	Units	Range
a	Constant	$W/(m^2K(m/s)^b)$	-
b	Constant	-	-
h_c	Surface exterior convective heat transfer coefficient	$W/(m^2K)$	-
$h_{c, \text{glass}}$	Convective heat transfer coefficient for very smooth surfaces (glass)	$W/(m^2K)$	-
h_n	Natural convective heat transfer coefficient	$W/(m^2K)$	-
R_f	Surface roughness multiplier	-	-
T_{so}	Outside surface temperature	$^{\circ}C/K$	-
ΔT	Temperature difference between the surface and air,	$^{\circ}C/K$	-
ϕ	Angle between the ground outward normal and the surface outward normal	radian	-

Description of the Model and Algorithm

The DOE-2 convection model is a combination of the MoWiTT and BLAST Detailed convection models (LBL 1994). The convection coefficient for very smooth surfaces (e.g. glass) is calculated as:

$$h_{c, glass} = \sqrt{h_n^2 + [aV_z^b]^2} \quad (86)$$

h_n is calculated using Equation (83) or Equation (84) . Constants a and b are given in Table 10.

For less smooth surfaces, the convection coefficient is modified according to the equation

$$h_c = h_n + R_f(h_{c, glass} - h_n) \quad (87)$$

where R_f is the roughness multiplier given by Table 8.

Exterior/External Conduction

The conduction term, q''_{ko} , can in theory be calculated using a wide variety of heat conduction formulations. Typically in EnergyPlus, the Conduction Transfer Function (CTF) method is used. The available models are described in this section: Conduction Through The Walls.

References

- ASHRAE. 1981. 1981 ASHRAE Handbook – Fundamentals, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- ASHRAE. 1989. 1989 ASHRAE Handbook – Fundamentals, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- ASHRAE. 1993. 1993 ASHRAE Handbook – Fundamentals, Chapter 3, Heat Transfer, I-P & S-I Editions, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- ASHRAE. 2001. 2001 ASHRAE Handbook – Fundamentals, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- ASHRAE. 2005. 2005 ASHRAE Handbook – Fundamentals, Chapter 16, Air Flow Around Buildings, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.
- Cole, R. J., and N. S. Sturrock. 1977. The Convective Heat Exchange at the External Surface of Buildings. Building and Environment, Vol. 12, p. 207.
- Ellis, P.G., and P.A. Torcellini. 2005. "Simulating Tall Buildings Using EnergyPlus", Proceedings of the Ninth International IBPSA Conference, Building Simulation 2005, Montreal, Canada, August 15-18, 2005.
- Lawrence Berkeley Laboratory (LBL). 1994. DOE2.1E-053 source code.
- Sparrow, E. M., J. W. Ramsey, and E. A. Mass. 1979. Effect of Finite Width on Heat Transfer and Fluid Flow about an Inclined Rectangular Plate. Journal of Heat Transfer, Vol. 101, p. 204.
- U.S. Standard Atmosphere. 1976. U.S. Government Printing Office, Washington, D.C.
- Walton, G. N. 1981. Passive Solar Extension of the Building Loads Analysis and System Thermodynamics (BLAST) Program, Technical Report, United States Army Construction Engineering Research Laboratory, Champaign, IL.
- Walton, G. N. 1983. Thermal Analysis Research Program Reference Manual. NBSSIR 83-2655. National Bureau of Standards.
- Yazdaniyan, M. and J. H. Klems. 1994. Measurement of the Exterior Convective Film Coefficient for Windows in Low-Rise Buildings. ASHRAE Transactions, Vol. 100, Part 1, p. 1087.

Inside Heat Balance

The heart of the heat balance method is the internal heat balance involving the inside faces of the zone surfaces. This heat balance is generally modeled with four coupled heat transfer components: 1) conduction through the building element, 2) convection to the air, 3) short wave radiation absorption and reflectance and 4) long wave radiant interchange. The incident short wave radiation is from the solar radiation entering the zone through windows and emittance from internal sources such as lights. The long wave radiation interchange includes the absorption and emittance of low temperature radiation sources, such as all other zone surfaces, equipment, and people.

The heat balance on the inside face can be written as follows:

$$q''_{LWX} + q''_{SW} + q''_{LWS} + q''_{ki} + q''_{sol} + q''_{conv} = 0 \quad (88)$$

where:

q''_{LWX} = Net long wave radiant exchange flux between zone surfaces.

q''_{SW} = Net short wave radiation flux to surface from lights.

q''_{LWS} = Long wave radiation flux from equipment in zone.

q''_{ki} = Conduction flux through the wall.

q''_{sol} = Transmitted solar radiation flux absorbed at surface.

q''_{conv} = Convective heat flux to zone air.

Each of these heat balance components is introduced briefly below.

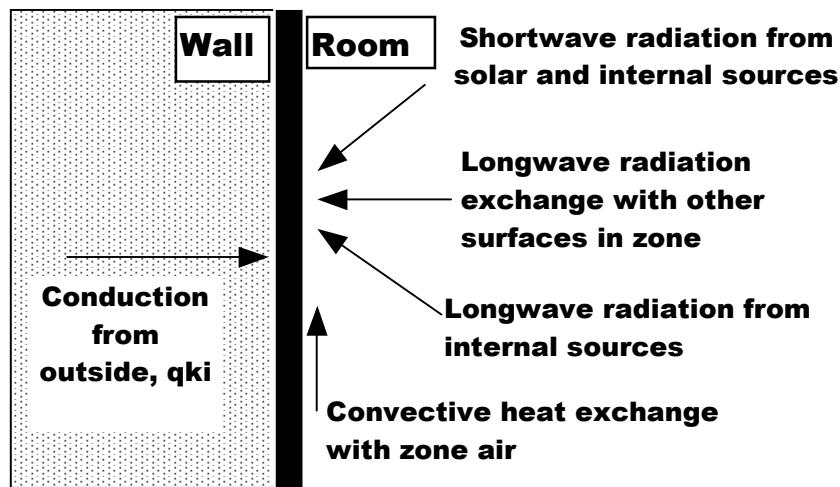


Figure 15. Inside Heat Balance Control Volume Diagram

Internal Long-Wave Radiation Exchange

LW Radiation Exchange Among Zone Surfaces

There are two limiting cases for internal LW radiation exchange that are easily modeled:

- The zone air is completely transparent to LW radiation.
- The zone air completely absorbs LW radiation from the surfaces within the zone.

The limiting case of completely absorbing air has been used for load calculations and also in some energy analysis calculations. This model is attractive because it can be formulated simply using a combined radiation and convection heat transfer coefficient from each surface

to the zone air. However, it oversimplifies the zone surface exchange problem, and as a result, the heat balance formulation in EnergyPlus treats air as completely transparent. This means that it does not participate in the LW radiation exchange among the surfaces in the zone. The model, which considers room air to be completely transparent, is reasonable physically because of the low water vapor concentrations and the short mean path lengths. It also permits separating the radiant and convective parts of the heat transfer at the surface, which is an important attribute of the heat balance method.

EnergyPlus uses a grey interchange model for the long wave radiation among zone surfaces. This model is based on the “ScriptF” concept developed by Hottel (Hottel and Sarofim, Radiative Transfer, Chapter 3, McGraw Hill, 1967). This procedure relies on a matrix of exchange coefficients between pairs of surfaces that include all exchange paths between the surfaces. In other words all reflections, absorptions and re-emissions from other surfaces in the enclosure are included in the exchange coefficient, which is called ScriptF. The major assumptions are that all surface radiation properties are grey and all radiation is diffuse. Both assumptions are reasonable for building zone interchange.

The ScriptF coefficients are developed by starting with the traditional direct radiation view factors. In the case of building rooms and zones, there are several complicating factors in finding the direct view factors—the main one being that the location of surfaces such as thermal mass representing furniture and partitions are not known. The other limitation is that the exact calculation of direct view factors is computationally very intensive even if the positions of all surfaces are known. Accordingly, EnergyPlus uses a procedure to approximate the direct view factors. The procedure has two steps:

- 1) Determine the total area of other surfaces “seen” by a surface.
- 2) Approximate the direct view factor from surface 1 to surface 2 as the ratio of the area of surface 2 to the total area “seen” by surface 1.

The determination of the “seen” area has several constraints:

- No surface sees itself.
- All surfaces see thermal mass surfaces.
- No surface facing within 10 degrees of another surface is seen by the other surface.
- All surfaces see roofs, floors and ceilings (subject to the preceding facing direction constraint).

Because the approximate view factors may not satisfy the basic requirements of reciprocity (two surfaces should exchange equal amounts of heat in each direction), and completeness (every surface should have a direct view factor sum of 1.0), EnergyPlus does a view factor fix operation before they are used in the ScriptF determination. Normally both of the requirements are satisfied, but in some special situations they are not, and special rules are applied.

- If a user includes less than four surfaces in a zone, only reciprocity is enforced.
- If the area of one surface in a zone is greater than the sum of the areas of all other surfaces, reciprocity only is enforced, but sometimes, for very large surfaces, that enforcement becomes impossible, and the view factors are modified so that only the large surface is seen by very small surfaces.

Warning messages are produced for both of these cases, and the results should be examined very carefully to ascertain that they are reasonable. The suggested action for the second case (the extra large surface) is to divide the large surface into several smaller surfaces; then the enclosure will be treated as normal.

Once the ScriptF coefficients are determined, the long wave radiant exchange is calculated for each surface using:

$$q_{i,j} = A_i F_{i,j} (T_i^4 - T_j^4)$$

where F_{ij} is the ScriptF between surfaces i and j.

Thermal Mass and Furniture

Furniture in a zone has the effect of increasing the amount of surface area that can participate in the radiation and convection heat exchanges. It also adds participating thermal mass to the zone. These two changes both affect the response to temperature changes in the zone and also affect the heat extraction characteristics.

The proper modeling of furniture is an area that needs further research, but the heat balance formulation allows the effect to be modeled in a realistic manner by including the furniture surface area and thermal mass in the heat exchange process.

LW Radiation From Internal Sources

The traditional model for this source is to define a radiative/convective split for the heat introduced into a zone from equipment. The radiative part is then distributed over the surfaces within the zone in some prescribed manner. This, of course, is not a completely realistic model, and it departs from the heat balance principles. However, it is virtually impossible to treat this source in any more detail since the alternative would require knowledge of the placement and surface temperatures of all equipment.

Internal Short-Wave Radiation

SW Radiation from Lights

The short wavelength radiation from lights is distributed over the surfaces in the zone in some prescribed manner.

Transmitted Solar

Transmitted solar radiation is also distributed over the surfaces in the zone in a prescribed manner. It would be possible to calculate the actual position of beam solar radiation, but that would involve partial surface irradiation, which is inconsistent with the rest of the zone model that assumes uniform conditions over an entire surface. The current procedures incorporate a set of prescribed distributions. Since the heat balance approach can deal with any distribution function, it is possible to change the distribution function if it seems appropriate.

Convection to Zone Air

The convection flux is calculated using the heat transfer coefficients as follows:

$$q''_{conv} = h_c (T_a - T_s) \quad (89)$$

The inside convection coefficients (h_c) can be calculated using one of several models. Currently the implementation uses coefficients based on natural convection correlations (3 options) and mixed and forced convection (1 option).

Interior Conduction

This contribution to the inside surface heat balance is the wall conduction term, q''_{ki} shown in Equation (30). This represents the heat transfer to the inside face of the building element. Again, a CTF formulation is used to determine this heat flux.

Interior Convection

Four inside convection models are included in EnergyPlus: two natural convection models, a mixed / forced convection model, and a trombe wall convection model. Reference "Inside Convection Algorithm" object in the Input Output Reference document and the inside convection field for each zone. An overall default for the simulation is selected in the "SurfaceConvectionAlgorithm:Inside" object and can be overridden by selecting a different

option in a zone description. These models are explained in the following sections. In addition to the correlation choices described below, it is also possible to override the convection coefficients on the inside of any surface by using the "SurfaceProperty:ConvectionCoefficients" object in the input file to set the convection coefficient value on the inside of any surface. The values can be specified directly or with schedules. Specific details are given in the Input Output Reference document.

Detailed Natural Convection Algorithm

The detailed natural convection model correlates the convective heat transfer coefficient to the surface orientation and the difference between the surface and zone air temperatures (where $\Delta T = \text{Surface Temperature} - \text{Air Temperature}$). The algorithm is taken directly from Walton (1983). Walton derived his algorithm from the ASHRAE Handbook (2001), Table 5 on p. 3.12, which gives equations for natural convection heat transfer coefficients in the turbulent range for large, vertical plates and for large, horizontal plates facing upward when heated (or downward when cooled). A note in the text also gives an approximation for large, horizontal plates facing downward when heated (or upward when cooled) recommending that it should be half of the facing upward value. Walton adds a curve fit as a function of the cosine of the tilt angle to provide intermediate values between vertical and horizontal. The curve fit values at the extremes match the ASHRAE values very well.

For no temperature difference OR a vertical surface the following correlation is used:

$$h = 1.31|\Delta T|^{\frac{1}{3}} \quad (90)$$

For ($\Delta T < 0.0$ AND an upward facing surface) OR ($\Delta T > 0.0$ AND an downward facing surface) an enhanced convection correlation is used:

$$h = \frac{9.482|\Delta T|^{\frac{1}{3}}}{7.283 - |\cos \Sigma|} \quad (91)$$

where Σ is the surface tilt angle.

For ($\Delta T > 0.0$ AND an upward facing surface) OR ($\Delta T < 0.0$ AND an downward facing surface) a reduced convection correlation is used:

$$h = \frac{1.810|\Delta T|^{\frac{1}{3}}}{1.382 + |\cos \Sigma|} \quad (92)$$

where Σ is the surface tilt angle.

Simple Natural Convection Algorithm

The simple convection model uses constant coefficients for different heat transfer configurations, using the same criteria as the detailed model to determine reduced and enhanced convection. The coefficients are also taken directly from Walton (1983). Walton derived his coefficients from the surface conductances for $\epsilon=0.90$ found in the ASHRAE Handbook (1985) in Table 1 on p. 23.2. The radiative heat transfer component was estimated at $1.02 * 0.9 = 0.918$ BTU/h-ft²-F and then subtracted off. Finally the coefficients were converted to SI units to yield the values below.

For a vertical surface:

$$h = 3.076$$

For a horizontal surface with reduced convection:

$$h = 0.948$$

For a horizontal surface with enhanced convection:

$$h = 4.040$$

For a tilted surface with reduced convection:

$$h = 2.281$$

For a tilted surface with enhanced convection:

$$h = 3.870$$

Ceiling Diffuser Algorithm

The ceiling diffuser algorithm is based on empirical correlations developed by Fisher and Pedersen (1997). The correlation was reformulated to use the room outlet temperature as the reference temperature. The correlations are shown below.

For Floors:

$$h = 3.873 + 0.082 * ACH^{0.98} \quad (93)$$

The correlation for floors is illustrated in the following figure:

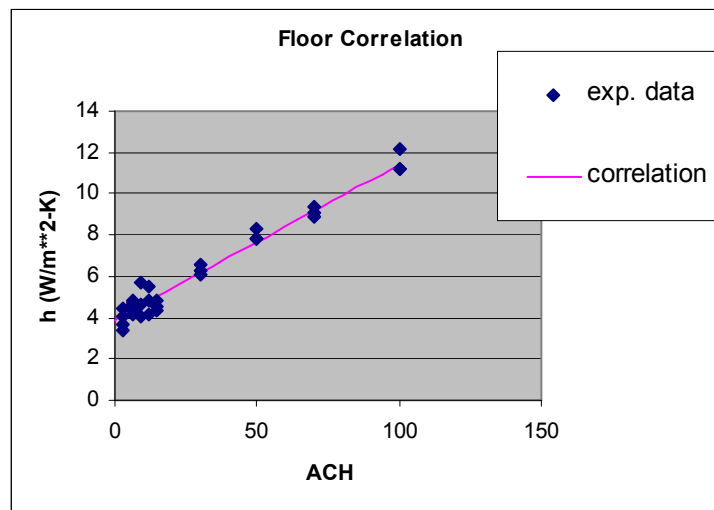


Figure 16. Ceiling Diffuser Correlation for Floors

For ceilings:

$$h = 2.234 + 4.099 * ACH^{0.503} \quad (94)$$

The correlation for ceilings is illustrated in the following figure:

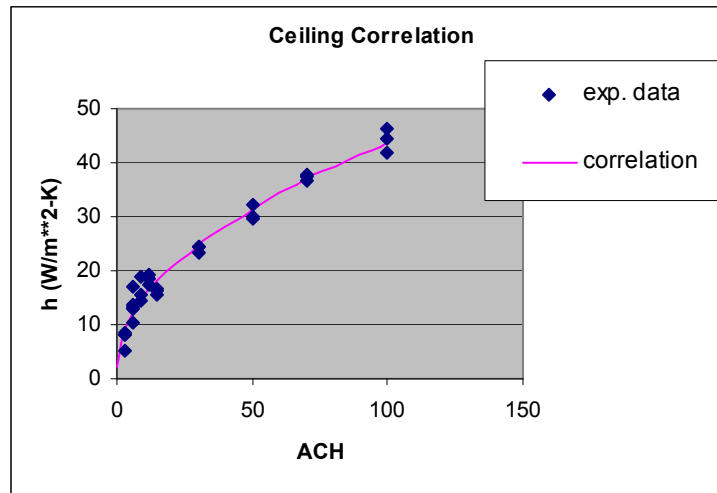


Figure 17. Ceiling Diffuser Correlation for Ceilings

For Walls:

$$h = 1.208 + 1.012 * ACH^{0.604} \quad (95)$$

The correlation for walls is illustrated in the following figure:

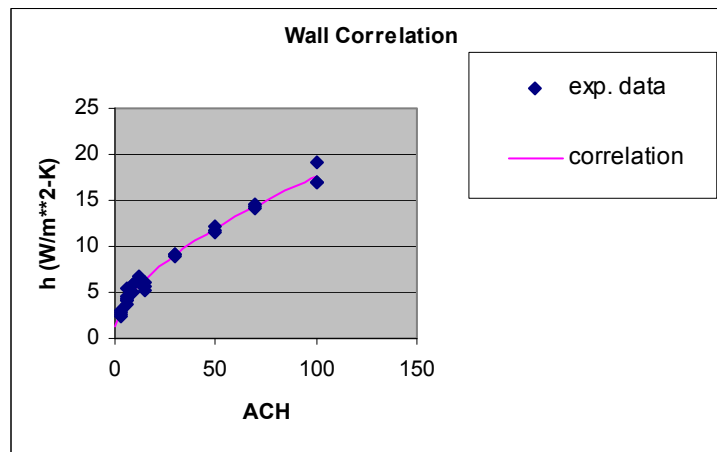


Figure 18. Ceiling Diffuser Correlation for Walls

Trombe Wall Algorithm

The Trombe wall algorithm is used to model convection in a "Trombe wall zone", i.e. the air space between the storage wall surface and the exterior glazing. (See the later sections on Passive and Active Trombe Walls below for more information about Trombe walls.) The algorithm is identical to the convection model (based on ISO 15099) used in Window5 for convection between glazing layers in multi-pane window systems. The use of the algorithm for modeling an unvented Trombe wall has been validated against experimental data by Ellis (2003).

This algorithm gives the convection coefficients for air in a narrow vertical cavity that is sealed and not ventilated. This applies both to the air gap in between panes of a window or to the air gap between the Trombe wall glazing and the inner surface (often a selective surface). These convection coefficients are really the only difference between a normal zone

and a Trombe zone. The rest of the zone heat balance is the same, e.g., transmitted solar, long-wave radiation between surfaces, etc.

For a vertical cavity, the correlation from ISO 15099 is:

$$NU1 = 0.0673838 Ra^{\frac{1}{3}} \quad \text{for } 5E4 < Ra < 1E6$$

$$NU1 = 0.028154 Ra^{0.4134} \quad \text{for } 1E4 < Ra < 5E4$$

$$NU1 = 1 + 1.7596678E-10 Ra^{2.2984755} \quad \text{for } Ra \leq 1E4$$

$$NU2 = 0.242 \left(\frac{Ra}{A} \right)^{0.272}$$

$$NU = \text{MAX}(NU1, NU2)$$

where

Nu = Nusselt number

Ra = Rayleigh number

A = aspect ratio of cavity

This is then used in EnergyPlus as follows:

Net convection coefficient from glazing to wall is:

$$h_{net} = k \left(\frac{NU}{L} \right)$$

where

k = conductivity of air

L = air gap thickness

Convection coefficient applied to each wall separately and actually used in the zone heat balance is:

$$h_c = 2h_{net}$$

References

ASHRAE. 1985. 1985 ASHRAE Handbook – Fundamentals, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.

ASHRAE. 2001. 2001 ASHRAE Handbook – Fundamentals, Atlanta: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.

Ellis, Peter G. 2003. Development and Validation of the Unvented Trombe Wall Model in EnergyPlus. Master's Thesis, University of Illinois at Urbana-Champaign.

Fisher, D.E. and C.O. Pedersen. 1997. "Convective Heat Transfer in Building Energy and Thermal Load Calculations", ASHRAE Transactions, Vol. 103, Pt. 2.

ISO. 2003. ISO 15099:2003. Thermal performance of windows, doors, and shading devices – Detailed calculations. International Organization for Standardization.

Walton, G. N. 1983. Thermal Analysis Research Program Reference Manual. NBSSIR 83-2655. National Bureau of Standards.

Infrared Radiation Transfer Material

One of the important technical features of EnergyPlus is that the fundamental zone model includes infrared (IR) radiation exchange among all surfaces within the zone. A zone can consist of a single room or multiple rooms having similar thermal characteristics. The other fundamental characteristic of a zone in EnergyPlus is that the air within the zone is modeled with the assumption that it is well-stirred, so it is characterized by a single uniform air temperature. Normally, this corresponds to the desired physical state of the air within a zone. Practically, this is accomplished with the mixing caused by the air distribution system or even with the addition of ceiling fans. However, there are situations where the well-stirred model condition does not fit the physical conditions within a space (e.g., large atria spaces, displacement ventilation systems, and under floor air distribution systems). In these cases, the air temperature varies spatially within the zone, but the surfaces in the regions that have different air temperatures still exchange IR radiation with the other surfaces in the zone, even those in regions having a different air temperature.

Historically, several approaches have been taken to model such situations. They all involve unrealistically modifying the convective heat transfer characteristics within a single zone to produce reasonable total heat transfer for the zone while maintaining the zone IR radiation in its basic form. The IRT model takes a different, more fundamental approach. The space is divided into subzones, each having the basic well-stirred air model, but surfaces in these subzones are able to exchange IR radiation with other surfaces throughout the original space. Any convective air exchange between subzones is handled using the existing flexible capabilities within EnergyPlus. In other words, the subzones are standard EnergyPlus zones but they have been given the capability of allowing IR radiation to be exchanged with surfaces in adjacent zones.

The IR radiation exchange is accomplished by introducing special IR Transparent (IRT) surfaces to link adjacent zones. These surfaces transmit the IR radiation incident on them to the surfaces in adjacent zone. The special surfaces have no thermal resistance, and they also have zero convective heat transfer coefficients on both sides. Consider a stratified atrium as an example. The atrium would be divided into stacked vertical zones, with each zone having the special IRT surface between it and the next lower zone. All physical surfaces in the two zones exchange radiation with the IRT surface, but it does not impede the transfer of radiation from one zone to the other. On each side it interacts as a black body surface and any radiation incident on it is absorbed. Since it has no other heat transfer mechanisms, it comes to equilibrium with zero net radiation transfer.

Schematic diagrams of the application of the IRT surfaces are shown in Figure 19 and Figure 20. The first figure shows it applied between two stacked zones that could be part of an atrium. All radiation incident on the IRT in the lower and upper zone is completely absorbed by the surface. It is prevented from participating in the zone air heat balance by fixing the heat transfer coefficients on either side at zero.

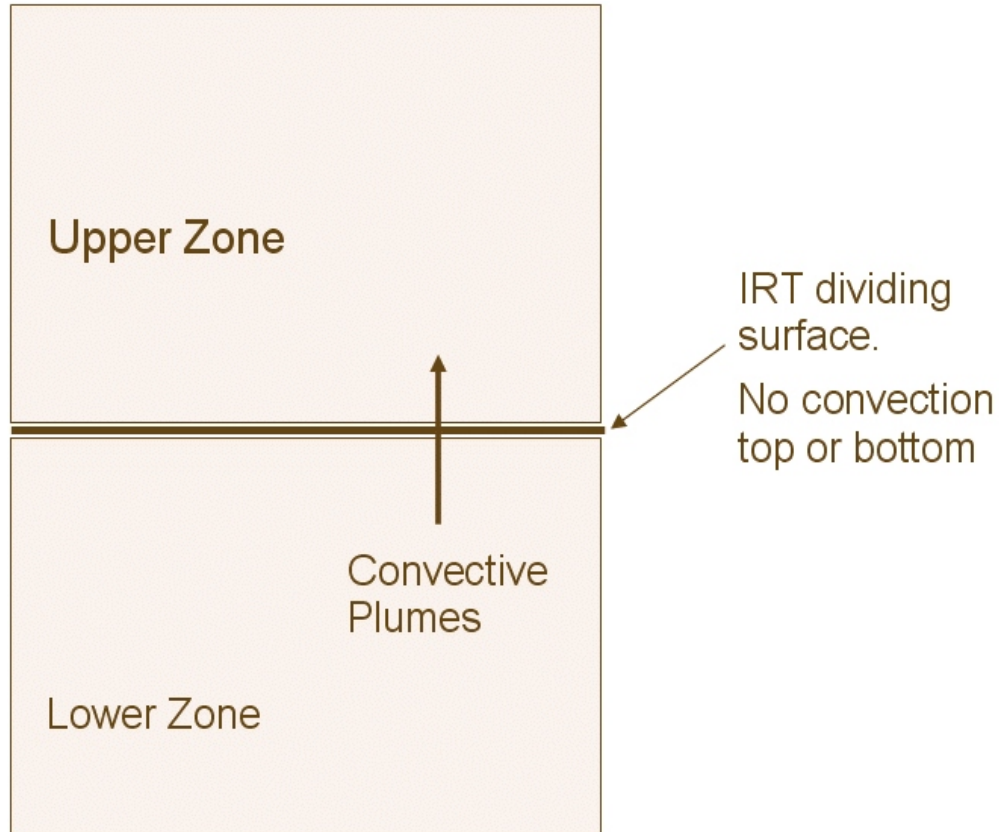


Figure 19. Schematic View of Stacked Zones

The same arrangement can be used for simulating a underfloor air distribution system (UFAD). The two zones represent the lower occupied (mixed) zone and the upper stratified zone. Since the upper zone is being modeled as a mixed EnergyPlus zone, it is not precisely the stratified zone concept. However, if a user has concern about having the entire upper part of the space at a single average temperature, the space could be modeled with two stacked upper zones. In that case the stratified temperature profile would be established by the relative size of the mixing from the convective plumes. The user would have to supply those estimates from external knowledge of the behavior of UFAD systems and plumes.

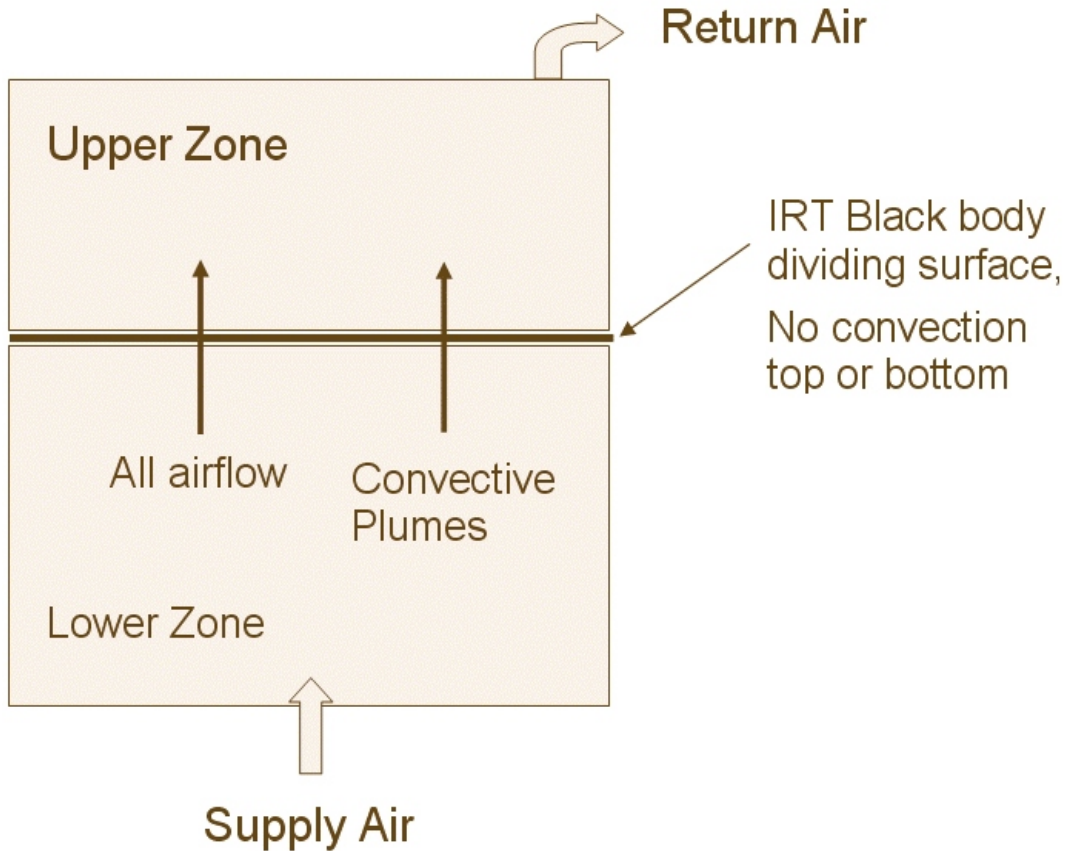


Figure 20. Schematic Model of a UFAD Zone

Radiation Exchange Basics

The behavior of the IRT surface can be analyzed by considering three parallel black plates infinite in extent, as shown in Figure 21. The outside plates are held at two different temperatures, and the intermediate plate assumes an equilibrium temperature when the net heat flux goes to zero. All plates have black body surface characteristics.

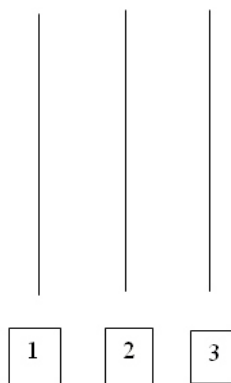


Figure 21. Radiation Exchange Model

The radiant exchange between surface 1 and 2 is:

$$q_{1-2} = \sigma A(T_1^4 - T_2^4) \tag{96}$$

The exchange between surfaces 2 and 3 is:

$$q_{2-3} = \sigma A (T_2^4 - T_3^4) \quad (97)$$

Equating heat fluxes and eliminating T_2 gives:

$$q_{1-3} = \frac{\sigma}{2} A (T_1^4 - T_3^4) \quad (98)$$

where:

q is the heat flux in W/m^2

σ is the Stephan Boltzman constant

A is the plate area in m^2 , and

T is the temperature in K.

Equation (98) shows that the presence of a black body surface between a source and a sink reduces the heat flux by a factor of two. The same result occurs when the IRT surface is between two zones in EnergyPlus. In that case the adjacent zones behave as black body cavities at some equivalent temperature. In order to account for this reduction, the IRT area must be doubled. This can be done without any difficulty in the EnergyPlus radiant exchange routine because the radiation view factors are determined by an approximate procedure that is based on the areas of the surfaces. Thus, doubling the surface area of the IRT surface results in the correct transfer of radiation through the IRT surface. The doubling will occur automatically in the program as described in the Input Output Reference document.

It should be noted that, because of the black body behavior of the IRT surface, any visible or solar short wavelength radiation incident on the surface will be absorbed and included with the long wavelength (IR) exchange with the adjacent zone. No energy will be lost, but zones with IRT surfaces should not be used in any lighting analyses.

Radiation Transfer Surface Details

Specifying an Infrared Transparent (IRT) surface

The Infrared Transparent (IRT) surface is similar to a resistance-only surface. The add object for this type of surface is shown below. The fields indicate that the surface will actually participate in the transfer of visible and solar radiation by doing a wavelength transformation and making all short wavelength radiation that is incident on the surface into long wavelength radiation and having it participate in the long wavelength radiant exchange. The Material:InfraredTransparent object requires only a name. All other parameters are set internally.

The Infrared Transparent surface should not participate in a convective/conductive exchange between the zones it separates. In order to minimize this effect, the SurfaceProperty:ConvectionCoefficients object must be used. Outside and Inside values for the surface's convection coefficients should be on the order of .1. Further examples are given in the Input Output Reference document.

Behavior Checks

The behavior of multi zones separated with infrared transparent surfaces can be checked with a simple comparison. Begin with a single zone model as shown below. This model has a south-facing window, and four walls exposed to wind and sun, and a roof exposed to wind and sun.

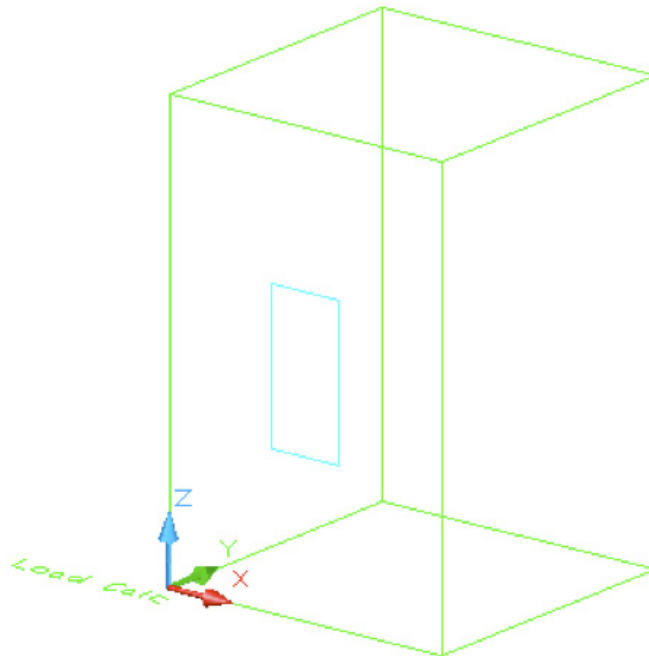


Figure 22. Single Zone Model

The single zone model will be compared with a stacked three zone model that has zones separated by interzone infrared transparent surfaces. This model is shown below.

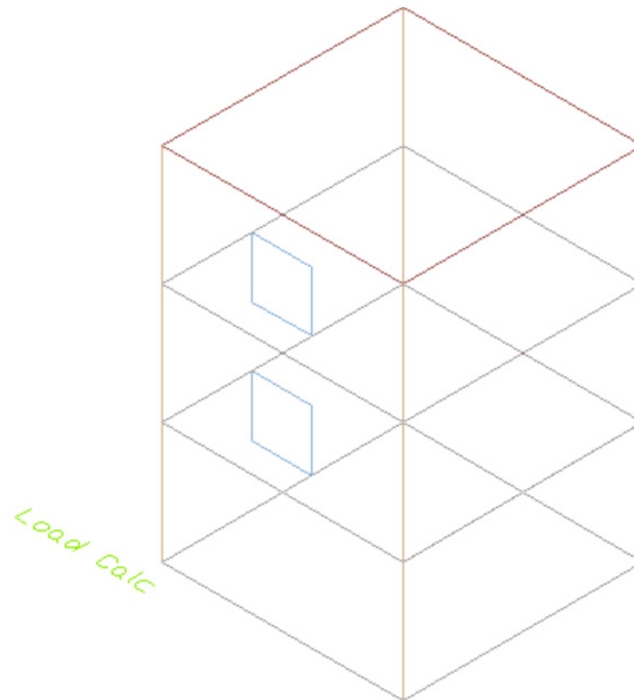


Figure 23. Stacked Three Zone Model

The two upper zones have south facing windows whose total area is the same as the area of the single window in the single zone model. The top and the sides are again exposed to sun and wind. The separating surfaces are modeled as IRT surfaces. All zones in both models are controlled at the same setpoint temperature using purchased air.

The sensible heating results are shown below. The results show the sum of the sensible cooling load for the three stacked zones and the single zone. It is clear that the IRT surfaces are very effective in transmitting infrared radiation between the zones. Some small differences, such as those shown, will occur because of the conversion from short wavelength solar to infrared through the special IRT dividing surfaces.

Comparison of Single Zone and Three Stacked Zones

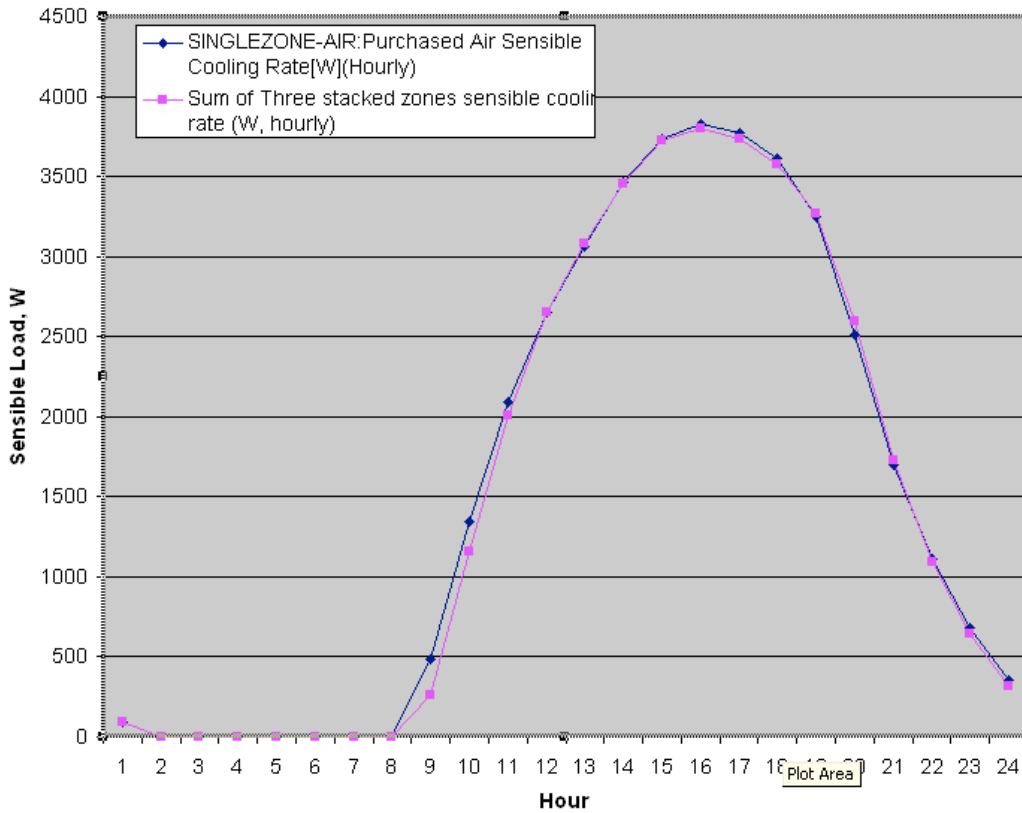


Figure 24. Comparison of Single and Three Stacked Zones

Transparent Insulation Material (TIM)

Introduction

The input object "SurfaceControl:MovableInsulation" allows modeling Transparent Insulation Materials (TIM) that were originally designed for use in solar collector systems, where there was a need to increase the insulation in the solar collector without dramatically reducing solar energy transmittance. Transparent Insulation provides both these properties, insulation from heat loss and transmittance of solar energy. The combination of these properties is achieved, because Transparent Insulation is a transmitter of short wave radiation but a barrier to long wave radiation. Therefore short wave solar radiation passes through the Transparent Insulation and long wave heat radiation is insulated by the transparent insulation. Incident solar energy falling on the transparent insulation is reflected and re-reflected within the material and eventually falls on the absorber. In addition, transparent insulation materials also have increase thermal resistance due to conduction in comparison to standard glass.

Transparent Insulation is now used in the housing industry as a passive solar feature. It is attached to the walls of houses for insulation and solar energy gains are transmitted to the house during the right ambient conditions. The walls of the house act as a thermal mass, absorbing the sunlight at the surface and converting it into heat which is slowly transmitted to the inside of the house.

Comparison of Opaque and Transparent Insulation

A qualitative comparison between the performance of Transparent Insulation and opaque insulation is shown diagrammatically in the figure below. The upper half of the figure represents approximate heat transfer through the wall cross-section for both transparent and opaque insulation cases. The lower half of this figure shows representative temperature variations through the wall cross-sections for different solar conditions.

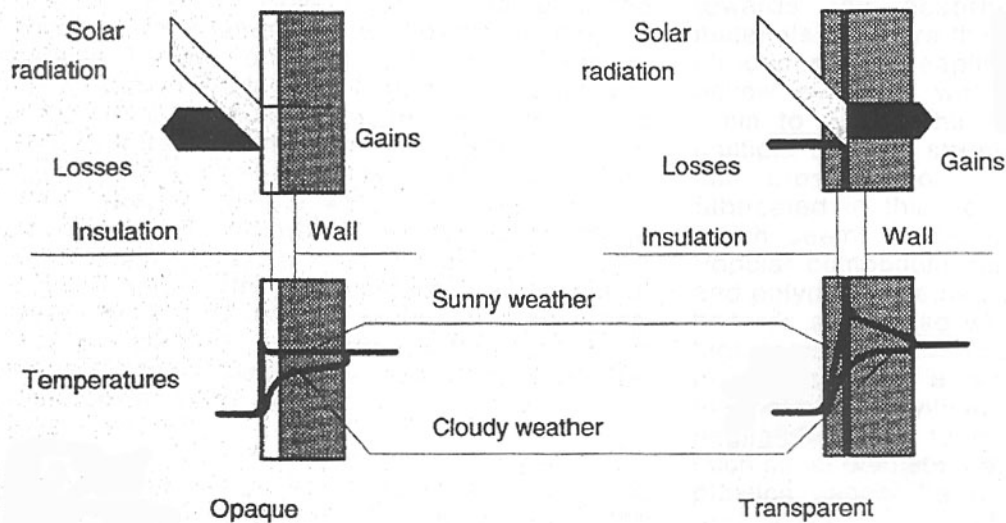


Figure 25. Energy Flows of Opaquely and Transparently Insulated Walls (Wood and Jesch 1993).

While both types of insulation reduce energy losses from the building via conduction through the building surfaces, transparent insulation allows solar radiation to penetrate deeper into the surface construction. This increases the construction internal temperature and can result in heat being conducted into the building under the proper weather conditions. This can be seen in the lower half of the above figure during a sunny day. The temperature plot shows a maximum between the transparent insulation and the rest of the surface construction. As a result, the temperature gradient results in heat transfer from this point into the interior space, causing a heating effect on the zone. Thus, the advantage of transparent insulation is that, like opaque insulation, it reduces winter heat transfer losses during low or no solar conditions and has the possibility of providing heating during sunny winter days. It should be noted that this same effect in summer could be detrimental to the cooling loads of a building since the introduction of solar radiation closer to the space will increase the solar heating within the zone. Most systems counteract this with a shading device or with sophisticated transparent insulation systems.

Types of Transparent Insulation Materials

Transparent insulation can be classified into four general categories:

- Absorber Parallel Covers
- Cavity Structures
- Absorber Vertical Covers
- Quasi-Homogeneous Structures

Cross-sections of each of these types is shown in the figure below. The arrows in these diagrams indicate solar rays and the path these rays trace as they are transmitted through the transparent insulation layer. The most advantageous set-up (see absorber-parallel below) would send most of the rays downward towards the interior of the building while minimizing the rays that are reflected back to the exterior environment.

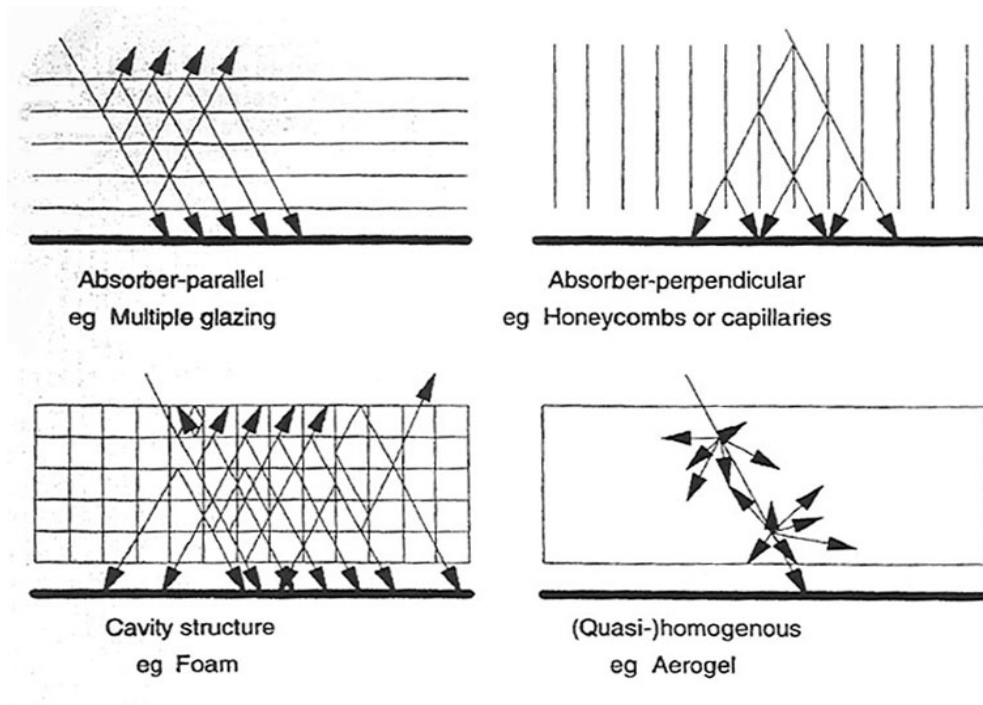


Figure 26. Geometrical Categories of Classification for Transparent Insulation Material (Wood and Jesch 1993).

TIM- Basic Mathematical Model

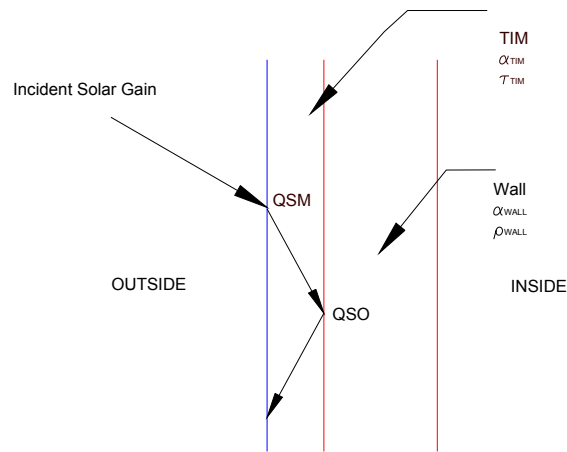


Figure 27. Cross Section of TIM and wall, showing energy flow

Mathematical model to calculate amount of energy absorbed at the surface of moveable insulation (TIM) and at the Outside surface of the Wall.

$$QSM = \alpha_{TIM} \cdot Incident\ Solar \tag{99}$$

The total solar gain on any exterior surface is a combination of the absorption of direct and diffuse solar radiation given by

$$\text{Incident Solar} = (I_b \cdot \cos \theta \cdot \frac{S_s}{S} + I_s \cdot F_{ss} + I_g \cdot F_{sg}) \quad (100)$$

Where,

α = solar absorptance of the surface

θ = angle of incidence of the sun's rays

S = area of the surface

S_s = sunlit area of the surface

I_b = intensity of the beam (direct) radiation

I_s = intensity of the sky diffuse radiation

I_g = intensity of the beam (direct) radiation

F_{ss} = angle factor between the surface and the sky

F_{sg} = angle factor between the surface and the ground

Now,

$$\alpha_{wall} + \rho_{wall} = 1 \quad (101)$$

The model for TIM is simplified in that it assumes that absorption of solar radiation takes place at the inside and outside of the TIM only, not throughout the material. In addition, the model assumes that the solar radiation absorbed during the first pass through the TIM affects the outside surface of the TIM while the solar radiation reflected at the outer wall surface that gets absorbed during the back reflection will affect the inside TIM surface (which is also the outside surface of the wall). Thus, the heat absorbed at the outside of the TIM is as shown in Equation (99).

The heat absorbed at the inside of the TIM/outside of the wall includes two components. The first component is the amount of solar that is transmitted through the TIM and absorbed at the inside of the wall. This is characterized by the following equation:

$$\text{First pass solar absorbed by wall} = (\tau_{TIM} \cdot \text{Incident Solar}) \cdot (\alpha_{wall})$$

The amount of solar absorbed by the TIM and aggregated at the inside surface of the TIM (outside wall surface) is:

$$\text{Amount of back reflection absorbed by TIM} = (\tau_{TIM} \cdot \text{Incident Solar}) \cdot (1 - \alpha_{wall}) \cdot \alpha_{TIM} \quad (102)$$

The heat absorbed at the interface between the wall and the TIM includes both of these components. Thus, QSO is equal to:

$$QSO = (\tau_{TIM} \cdot \text{Incident Solar}) \cdot \{ \alpha_{wall} + (1 - \alpha_{wall}) \cdot \alpha_{TIM} \} \quad (103)$$

Substituting the definition for QSM into this equation and rearranging results in:

$$QSO = \left\{ \frac{\tau_{TIM} \cdot QSM}{\alpha_{TIM}} \right\} \cdot \{ \alpha_{wall} + (1 - \alpha_{wall}) \cdot \alpha_{TIM} \}$$

$$QSO = (\tau_{TIM} \cdot QSM) \cdot \left\{ \left(\frac{\alpha_{wall}}{\alpha_{TIM}} \right) + (1 - \alpha_{wall}) \right\} \quad (104)$$

Where,

Q_{SM} = Short wave radiant flux absorbed at surface of Movable Insulation

Q_{SO} = Short wave radiant flux absorbed at surface of Wall.

α_{TIM} = Absorptance of TIM

τ_{TIM} = Transmittance of TIM.

α_{WALL} = Absorptance of Wall.

ρ_{WALL} = Reflectance of Wall surface

Following is the FORTRAN Code used in the HeatBalanceSurfaceManager module, to determine the short wave radiation absorbed on outside of movable insulation and the short wave radiation absorbed on outside of opaque surface of the wall.

```

IF (Surface (SurfNum) %MaterialMovInsulExt.GT.0) &
CALL EvalOutsideMovableInsulation (SurfNum, HMovInsul, RoughIndexMovInsul, AbsExt)
IF (HMOVINSUL > 0) THEN ! Movable outside insulation in place
  QRadSWOutMvIns (SurfNum) = QRadSWOutAbs (SurfNum) * AbsExt &
  /Material (Construct (ConstrNum) %LayerPoint (1)) %AbsorpSolar
! For Transparent Insulation
  QRadSWOutAbs (SurfNum) = Material (Surface (SurfNum) %MaterialMovInsulExt) %Trans &
  *QRadSWOutMvIns (SurfNum) * &
  ( (Material (Construct (ConstrNum) %LayerPoint (1)) %AbsorpSolar / AbsExt) &
  + (1 - Material (Construct (ConstrNum) %LayerPoint (1)) %AbsorpSolar) )

```

Sample Test Run Cases: – Comparison

A series of test cases were run in EnergyPlus to test the TIM model. The building was a very simple box with walls facing north, south, east, and west, all of which are exterior walls. Transparent Insulation Material has been applied to the south wall (except as noted in the table below). The program was run for this fictional 1 zone building located in Chanute AFB IL, for two design days, (21st June and 21st January). The main purpose of these runs was to verify that the transparent insulation model was predicting results that were reasonable using a simple test case. The winter design day was also modified in some runs to have a clearness of 1.0 so that the effect that solar radiation during winter-time conditions could be studied.

The Transparent Insulation material is conceived by applying a SurfaceControl:MoveableInsulation on the exterior. In the test cases, the TIM had the following thermal properties:

0.05, ! Thickness {m}
 0.90, ! Solar transmittance at normal incidence
 0.031, ! Solar reflectance at normal incidence: front side
 0.031, ! Solar reflectance at normal incidence: back side
 0.90, ! Visible transmittance at normal incidence
 0.05, ! Visible reflectance at normal incidence: front side
 0.05, ! Visible reflectance at normal incidence: back side
 0.0, ! IR transmittance at normal incidence
 0.84, ! IR emissivity: front side
 0.84, ! IR emissivity: back side
 0.04; ! Conductivity {W/m-K}

The Wall Construction is defined as an EXTWALL80 composed of 1" Stucco, 4" Common Brick and ¾" Plaster or Gypboard.

The following two tables shows data for two series of runs. The first "summer table" illustrates the execution of a summer design day. The second "winter table" shows winter conditions with clearness=0 (the typical default for a winter design day) and clearness=1 (to illustrate solar radiation with other winter conditions). Test cases included no movable

insulation, moveable opaque insulation, and TIM on the exterior (south wall unless otherwise noted). Savings reported are heating and cooling loads for the design days only. The results showed that the TIM model was performing reasonably well and was producing results that were within expectations.

Table 12. TIM with Summer Conditions

EXTWALL80 Construction	Conductivity	Thick- ness.	Sensible Cooling Energy	Energy Saved
	[W/m-K]	[m]	[J]	[J]
Normal case Without any Insulation	0.000	0.000	3.37E+08	0.00E+00
With Dense Insulation Present	0.040	0.025	3.17E+08	2.05E+07
With Dense Insulation Present	0.040	0.050	3.09E+08	2.84E+07
With Dense Insulation Present	0.040	0.100	3.02E+08	3.53E+07
With TIM Present	0.040	0.025	4.27E+08	-9.01E+07
With TIM Present	0.040	0.050	4.63E+08	-1.26E+08
With TIM Present	0.040	0.100	4.89E+08	-1.52E+08
With TIM Present -R value = (0.05m,0.04W/m-K)	0.035	0.044	4.63E+08	-1.26E+08
With TIM Present (EAST WALL)	0.040	0.050	5.49E+08	-2.12E+08
With TIM Present (NORTH WALL)	0.040	0.050	3.63E+08	-2.57E+07
With TIM Present (WEST WALL)	0.040	0.050	5.64E+08	-2.27E+08

Table 13. TIM with Winter Conditions

EXTWALL80 Construction	Conduc- tivity	Thick- ness.	Sensible Heating Energy	Energy Saved Winter Clear- ness=0	Sensible Heating Energy	Energy Saved Winter Clear- ness=1
	[W/m-K]	[m]	[J]	[J]	[J]	[J]
Normal case Without any Insulation	0.000	0.000	1.47E+09	0.00E+00	1.05E+09	0.00E+00
With Dense Insulation Present	0.040	0.025	1.30E+09	1.70E+08	9.76E+08	7.40E+07
With Dense Insulation Present	0.040	0.050	1.26E+09	2.10E+08	9.73E+08	7.70E+07
With Dense Insulation Present	0.040	0.100	1.22E+09	2.50E+08	9.74E+08	7.60E+07
With TIM Present	0.040	0.025	1.30E+09	1.70E+08	5.66E+08	4.84E+08
With TIM Present	0.040	0.050	1.26E+09	2.10E+08	4.41E+08	6.09E+08
With TIM Present	0.040	0.100	1.22E+09	2.50E+08	3.57E+08	6.93E+08
With TIM Present -R	0.035	0.044	1.26E+09	2.10E+08	4.40E+08	6.10E+08

value = (0.05m,0.04W/m-K)						
With TIM Present (EAST WALL)	0.040	0.050	1.26E+09	2.10E+08	7.36E+08	3.14E+08
With TIM Present (NORTH WALL)	0.040	0.050	1.24E+09	2.30E+08	8.31E+08	2.19E+08
With TIM Present (WEST WALL)	0.040	0.050	1.24E+09	2.30E+08	7.07E+08	3.43E+08

References

P.O. Braun, A. Goetzberger, J. Schmid, and W. Stahl. Transparent Insulation of Building Facades- Steps from Research to Commercial applications, Fraunhofer Institute for Solar Energy Systems, Oltmannsstrasse 22, D-7800 Freiburg, Germany.

Thermotropic materials and Systems for Overheating Protection.

<http://www.ise.fhg.de/Projects/Solbuild/materials.html>

Robert Hausner. Arbeitsgemeinschaft Erneuerbare energie, Transparent Insulation- Areas of Application, Society for Renewable Energy. <http://www.aee.at/verz/english/tin.html>

Werner J. Platzer. Transparent Insulation materials: a review, Fraunhofer Institute for Solar Energy Systems, Oltmannsstr. 5, D-79100 Freiburg, Germany.

Volker Wittwer. The use of Transparent Insulation Materials and Optical Switching Layers in Window Systems, Fraunhofer Institute for Solar Energy Systems, Oltmannsstr. 5, D-79100 Freiburg, Germany.

M. Wood and L.F. Jesch. 1993. Transparent insulation technology: a technical note, Ambient Press Limited.

Facade Modules with back-ventilated Transparent Insulation- Research and Development toward Series Application. <http://www.ise.fhg.de/Projects/development99/art4.html>

Two 0-Energy Houses, http://www.smartarch.nl/smartgrid/items/oo5_chur.htm

Advanced Building Technologies – Transparent Insulation Materials (TIM).

<http://www.enermodal.com/advancedtech/transp.html>

Transparent Insulation, http://www.esv.or.at/service/info-material/diverse/twd/index_e.htm

G. Verbeeck, H. Hens. Transparent Insulation: an alternative solution for summer discomfort. <http://www.bwk.kuleuven.ac.be/bwk/sr99/bwf.htm#bf1.15>

sto, Die neue Transparenz: Warmedamm-Verbund- system StoTherm Solar.

E.Lindauer, H.Leonhardt. Brauchwasservorerwärmung mit transparent gedämmten Bauteilen (Hybridsystem), Fraunhofer- Institut für Bauphysik.

Surface Heat Balance With Moveable Insulation

Basic Heat Balance Cases

A heat balance must exist at the outside surface-air interface. The incoming conductive, convective, and radiative fluxes must sum up to zero:

$$\text{Conductive} + \text{Convective} + \text{Radiative} = 0 \quad (105)$$

In contrast to the internal surface heat balance that treats all surfaces simultaneously, the external thermal balance for each surface is performed independent of all other surfaces. This implies that there is no direct interaction between the individual surfaces.

TARP includes four possible representations for the basic outside surface heat balance. The first two depend on which of the optimal surface conductance algorithms the user selects. The simple outside surface conductance that includes both the convective and thermal interchange between the surface and the environment in a single coefficient, is represented by the thermal network in Figure 28. Equation (105) can also be expressed as:

$$[KOP_t + Y_0 \cdot TI_t - X_0 \cdot TO_t] + [HO \cdot (T_a - TO_t)] + QSO = 0 \quad (106)$$

This can be solved for the outside surface temperature.

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_t + HO \cdot T_a}{X_0 + HO} \right] \quad (107)$$

The detailed outside surface conductance model considers convection and radiant interchange with the sky and with the ground as separate factors. Its use in the outside thermal balance is shown in Figure 29. In this case, equation (105) can be expanded to give

$$[KOP_t + Y_0 \cdot TI_t - X_0 \cdot TO_t] + [HA \cdot (T_a - TO_t) + HS \cdot (T_s - TO_t) + HG \cdot (T_g - TO_t)] + QSO = 0 \quad (108)$$

This can be solved for the outside surface temperature:

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_t + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g}{X_0 + HA + HS + HG} \right] \quad (109)$$

The third and fourth representations occur when the outside surface has been covered with movable insulation. The insulation has a conductance of UM. The thermal network in Figure 30 represents this case. The insulation must be mass-less because it is not generally possible to perform a correct thermal balance at the juncture of two surfaces each modeled by CTF.

The equation for the thermal balance between the surface and the insulation is

$$[KOP_t + Y_0 \cdot TI_t - X_0 \cdot TO_t + UM \cdot (TM - TO_t)] + QSO = 0 \quad (110)$$

Which can be rewritten to solve for TO :

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_t + UM \cdot TM}{X_0 + UM} \right] \quad (111)$$

Depending on whether or not the detailed or simple algorithm for surface conductance is being used, there are two expressions for TM, the outside temperature of the insulation. For the simple conductance:

$$TM = \left[\frac{QSM + UM \cdot TO_t + HO \cdot T_a}{UM + HO} \right] \quad (112)$$

For the detailed conductance:

$$TO_t = \left[\frac{QSM + UM \cdot TO_t + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g}{UM + HA + HS + HG} \right] \quad (113)$$

In this case the values of HA, HS and HG must be found by using an estimated value of TM in place of TO.

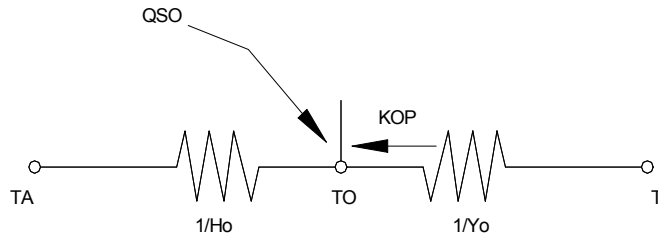


Figure 28. Thermal Network for Simple Outside Surface Coefficient

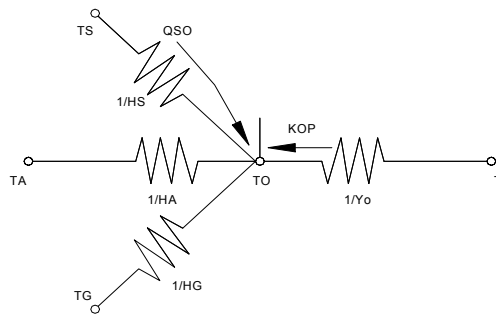


Figure 29. Thermal Network for Detailed Outside Surface Coefficient

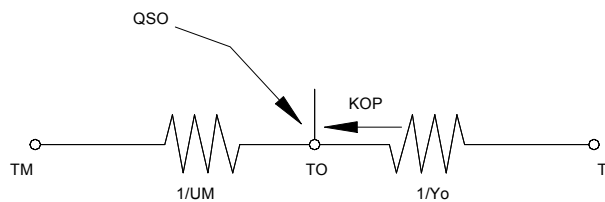


Figure 30. Thermal Network for Outside Moveable Insulation

Heat Balance Cases

TO_t and TI_t are related through the Y_0 CTF. However TI_t is also unknown. While it is possible to combine the outside and the inside surface heat balances to compute TO_t and TI_t simultaneously, TARP uses a simpler procedure where TO_t is based on a previous value of TI . When Y_0 is small, as occurs in well insulated or very massive surfaces, TI_t can be replaced by TI_{t-1} (which is known for the previous hour's heat balance) without significantly

effecting the value of TO_t . When Y_0 is large, TO and TI can so strongly be coupled that separate outside and inside heat balances do not work because the environment and zone temperatures have negligible influence on the heat balances. The TARP uses the inside surface heat balance to couple TO_t with TZ and TR . These two temperatures are less strongly influenced by TO and allow a reasonable heat balance. On the first heat balance iteration, TZ and TR are the values at time $t-1$. The user may optionally require that TO_t be recomputed with every iteration of TI_t . In this case TZ and TR have values from the previous iteration and a true simultaneous solution is achieved. In most conventional constructions, recomputing TO_t does not significantly change the computed zone loads and temperatures. The inside surface heat balance is given by

$$TI_t = \left[\frac{KIP_t + QSI + HC \cdot TZ + HR \cdot TR + Y_0 \cdot TO}{Z_0 + HC + HR} \right] \quad (114)$$

The surface heat balances can be combined in eight ways according to conditions for calculations of the outside surface temperature

$$F_1 = \left[\frac{Y_0}{Z_0 + HI + HR} \right] \quad (115)$$

$$F_2 = \left[\frac{UM}{UM + HO} \right] \quad (116)$$

$$F_3 = \left[\frac{UM}{UM + HA + HS + HG} \right] \quad (117)$$

Case1: Y_0 small, simple conductance, no movable insulation:

From Equation (107)

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_{t-1} + HO \cdot T_a}{X_0 + HO} \right] \quad (118)$$

Case2: Y_0 not small, simple conductance, no movable insulation:

From Equations (107) and (114)

$$TO_t = \left[\frac{KOP_t + QSO + HO \cdot T_a + F_1 \cdot (KIP_t + QSI + HI \cdot TZ + HR \cdot TR)}{X_0 + HO - F_1 \cdot Y_0} \right] \quad (119)$$

Case3: Y_0 small, detailed conductance, no movable insulation:

From Equation (108)

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_{t-1} + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g}{X_0 + HA + HS + HG} \right] \quad (120)$$

Case4: Y_0 not small, detailed conductance, no movable insulation:

From Equations (108) and (114)

$$TO_t = \left[\frac{KOP_t + QSO + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g + F_1 \cdot (KIP_t + QSI + HI \cdot TZ + HR \cdot TR)}{X_0 + HA + HS + HG - F_1 \cdot Y_0} \right] \quad (121)$$

Case5: Y_0 small, simple conductance, with movable insulation:

From Equations (111) and (112)

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_{t-1} + F_2 \cdot (QSM + HO \cdot TM)}{X_0 + UM - F_2 \cdot UM} \right] \quad (122)$$

Case6: Y_0 not small, simple conductance, with movable insulation:

From Equations (111), (112) and (114)

$$TO_t = \left[\frac{KOP_t + QSO + F_2 \cdot (QSM + HO \cdot T_a) + F_1 \cdot (KIP_t + QSI + HI \cdot TZ + HR \cdot TR)}{X_0 + UM - F_2 \cdot UM - F_1 \cdot Y_0} \right] \quad (123)$$

Case7: Y_0 small, detailed conductance, with movable insulation:

From Equations (111) and (113)

$$TO_t = \left[\frac{KOP_t + QSO + Y_0 \cdot TI_{t-1} + F_3 \cdot (QSM + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g)}{X_0 + UM - F_3 \cdot UM} \right] \quad (124)$$

Case8: Y_0 not small, detailed conductance, with movable insulation:

From Equations (111), (113) and (114)

$$TO_t = \left[\frac{KOP_t + QSO + F_1 \cdot (KIP_t + QSI + HI \cdot TZ + HR \cdot TR) + F_3 \cdot (QSM + HA \cdot T_a + HS \cdot T_s + HG \cdot T_g)}{X_0 + UM - F_3 \cdot UM - F_1 \cdot Y_0} \right] \quad (125)$$

FORTRAN Algorithm Examples**Case5: Y_0 small, simple conductance, with movable insulation:**

From Equation (122)

```

! Outside heat balance case: Movable insulation, slow conduction, simple
convection
F2 = DBLE(HmovInsul) / ( DBLE(HmovInsul) + DBLE(HExtSurf(SurfNum)) )
TH(SurfNum,1,1) = (-CTFConstOutPart(SurfNum) &
                  +DBLE(QRadSWOutAbs(SurfNum)) &
                  +Construct(ConstrNum)%CTFCross(0)*TempSurfIn(SurfNum) &
                  +F2*( DBLE(QRadSWOutMvIns(SurfNum)) &
                  + DBLE(HExtSurf(SurfNum))*DBLE(TempExt) ) &
                  / ( Construct(ConstrNum)%CTFOutside(0) + DBLE(HmovInsul) &
                  - F2*DBLE(HMovInsul))

```

Case6: Y_0 not small, simple conductance, with movable insulation:

From Equation (123)

```

! Outside heat balance case: Movable insulation, quick conduction, simple
convection
F2 = DBLE(HmovInsul) / ( DBLE(HmovInsul) + DBLE(HExtSurf(SurfNum)) )
TH(SurfNum,1,1) = (-CTFConstOutPart(SurfNum)                &
+ DBLE(QRadSWOutAbs(SurfNum))                                &
+F2*( DBLE(QRadSWOutMvIns(SurfNum))                          &
+DBLE(HExtSurf(SurfNum))*DBLE(TempExt) )                    &
+F1*( CTFConstInPart(SurfNum)                                &
+ DBLE(QRadSWInAbs(SurfNum))                                  &
+ DBLE(QRadThermInAbs(SurfNum))                              &
+ DBLE(HConvIn(SurfNum))*MAT(ZoneNum)                        &
+ DBLE(NetLWRadToSurf(SurfNum)) ) )                          &
/( Construct(ConstrNum)%CTFOutside(0) + DBLE(HmovInsul)      &
-F2*DBLE(HMovInsul) - F1*Construct(ConstrNum)%CTFCross(0) )

```

Case7: Y_0 small, detailed conductance, with movable insulation:

From Equation (124)

```

! Outside heat balance case: Movable insulation, slow conduction, detailed
convection
F2 = DBLE(HMovInsul) / ( DBLE(HMovInsul) + DBLE(HExtSurf(SurfNum)) &
+DBLE(HSky) + DBLE(HGround) )
TH(SurfNum,1,1) = (-CTFConstOutPart(SurfNum)                &
+DBLE(QRadSWOutAbs(SurfNum))                                &
+Construct(ConstrNum)%CTFCross(0)*TempSurfIn(SurfNum)      &
+F2*( DBLE(QRadSWOutMvIns(SurfNum))                          &
+DBLE(HExtSurf(SurfNum))*DBLE(TempExt)                      &
+DBLE(HSky)*DBLE(SkyTemp)                                  &
+DBLE(HGround)*DBLE(OutDryBulbTemp) ) )                    &
/( Construct(ConstrNum)%CTFOutside(0)                       &
+DBLE(HMovInsul) - F2*DBLE(HMovInsul) )

```

Case8: Y_0 not small, detailed conductance, with movable insulation:

From Equation (125)

```

! Outside heat balance case: Movable insulation, quick conduction, detailed
convection
F2 = DBLE(HMovInsul) / ( DBLE(HMovInsul) + DBLE(HExtSurf(SurfNum)) &
+DBLE(HSky) + DBLE(HGround) )
TH(SurfNum,1,1) = (-CTFConstOutPart(SurfNum)                &
+DBLE(QRadSWOutAbs(SurfNum))                                &
+F1*( CTFConstInPart(SurfNum)                                &
+DBLE(QRadSWInAbs(SurfNum))                                  &
+DBLE(QRadThermInAbs(SurfNum))                              &
+DBLE(HConvIn(SurfNum))*MAT(ZoneNum)                        &
+DBLE(NetLWRadToSurf(SurfNum)) ) )                          &
+F2*( DBLE(QRadSWOutMvIns(SurfNum))                          &
+DBLE(HExtSurf(SurfNum))*DBLE(TempExt)                      &
+DBLE(HSky)*DBLE(SkyTemp)                                  &
+DBLE(HGround)*DBLE(OutDryBulbTemp) ) )                    &
/( Construct(ConstrNum)%CTFOutside(0)                       &
+DBLE(HMovInsul) - F2*DBLE(HMovInsul)                      &
-F1*Construct(ConstrNum)%CTFCross(0) )

```

Fortran Variable Descriptions

Table 14. Fortran Variables and Descriptions

FORTTRAN Variable	Description	Tarp	Units	Description
-------------------	-------------	------	-------	-------------

		Variable		
TH(SurfNum,1,1)	Temperature History(SurfNum,Hist Term,In/Out), where: Hist Term (1 = Current Time, 2- MaxCTFTerms = previous times), In/Out (1 = Outside, 2 = Inside)	TO_t	C	Temperature of outside of surface I at time t
Construct(ConstrNum)%CTFCross(0)	Cross or Y term of the CTF equation	$Y0$	W/m^2K	Cross CTF term
Construct(ConstrNum)%CTFInside(0)	Inside or Z terms of the CTF equation	$Z0$	W/m^2K	Inside CTF term
Construct(ConstrNum)%CTFOutside(0)	Outside or X terms of the CTF equation	$X0$	W/m^2K	Outside CTF term
CTFConstInPart(SurfNum)	Constant inside portion of the CTF calculation	KIP_t	W/m^2	Portion of inward conductive flux based on previous temperature and flux history terms
CTFConstOutPart(SurfNum)	Constant Outside portion of the CTF calculation	KOP_t	W/m^2	Portion of outward conductive flux based on previous temperature and flux history terms
F1, F2, F3	Intermediate calculation variables	F1, F2, F3		Radiation interchange factor between surfaces
GroundTemp	Ground surface temperature	T_g	C	Temperature of ground at the surface exposed to the outside environment
HConvIn(SurfNum)	Inside convection coefficient	HI	W/m^2K	Inside convection coefficient
HExtSurf(SurfNum)	Outside Convection Coefficient	HO, HA	W/m^2K	Overall outside surface conductance

HGround	Radiant exchange (linearized) coefficient	HG	W/m^2K	Radiative conductance (outside surface to ground temperature)
HmovInsul	Conductance or "h" value of movable insulation	UM	W/m^2K	Conductance of Movable insulation
HSky	Radiant exchange (linearized) coefficient	HS	W/m^2K	Radiative conductance (outside surface to sky radiant temperature)
MAT(ZoneNum)	Zone temperature	TZ	C	Temperature of zone air
NetLWRadToSurf(SurfNum)	Net interior long wave radiation to a surface from other surfaces	HR*TR	W/m^2	Net surface to surface radiant exchange
QRadSWInAbs(SurfNum)	Short-wave radiation absorbed on inside of opaque surface	QSI	W/m^2	Short wave radiant flux absorbed at inside of surface
QRadSWOutAbs(SurfNum)	Short wave radiation absorbed on outside opaque surface	QSO	W/m^2	Short wave radiant flux absorbed at outside of surface
QRadSWOutMvIns(SurfNum)	Short wave radiation absorbed on outside of movable insulation	QSM	W/m^2	Short wave radiant flux absorbed at surface of movable insulation
QRadTherminAbs(SurfNum)	Thermal Radiation absorbed on inside surfaces		W/m^2	Long wave radiant flux from internal gains
SkyTemp	Sky temperature	T_s	C	Sky temp
TempExt	Exterior surface temperature or exterior air temperature	TM, T_a	C	Temperature of external surface of movable insulation or outside ambient air

				temperature
TempSurfIn(SurfNum)	Temperature of inside surface for each heat transfer surface	$T_{I,t-1}$	C	Temperature of inside of surface I at time t-1

References

Walton, G.N. 1983. "The Thermal Analysis Research Program Reference Manual Program (TARP)", National Bureau of Standards (now National Institute of Standards and Technology).

Advanced Surface Concepts

Exterior Naturally Vented Cavity

The input object “SurfaceProperty:ExteriorNaturalVentedCavity” allows modeling a special case for the outside boundary conditions of heat transfer surfaces with a multi-skin exterior that is opaque. From the thermal envelope’s point of view, the presence of a vented cavity on the outside of the surface modifies the conditions experienced by the underlying heat transfer surfaces. This exterior cavity acts as a radiation and convection baffle situated between the exterior environment and the outside face of the underlying heat transfer surface. The actual outer surface is referred to as the “baffle”. The modeling here assumes that the heat capacity in the outer baffle can be neglected since it is much lower than the underlying mass surface. This object is used with the BuildingSurface:Detailed object where the heat transfer surfaces are referred to as the underlying surfaces. The constructions and materials for the heat transfer surfaces should reflect the construction of just the underlying surface. The SurfaceProperty:ExteriorNaturalVentedCavity object is used to describe the detached layer, or baffle, and the characteristics of the cavity and openings for natural ventilation. This model uses the SurfaceProperty:OtherSideConditionsModel object to pass boundary conditions to the heat transfer modeling for the underlying surfaces.

Baffle Heat Balance

The baffle is assumed to be sufficiently thin and high-conductivity so that it can be modeled using a single temperature (for both sides and along its area). This temperature $T_{s,baff}$ is determined by formulating a heat balance on a control volume that just encapsulates the baffle surface. The baffle is assumed to completely cover the underlying surface such that it is opaque to shortwave and longwave radiation. This assumption means that even though the baffle will have some open area for ventilation, no solar energy passes through these openings. The heat balance is diagrammed in the following figure.

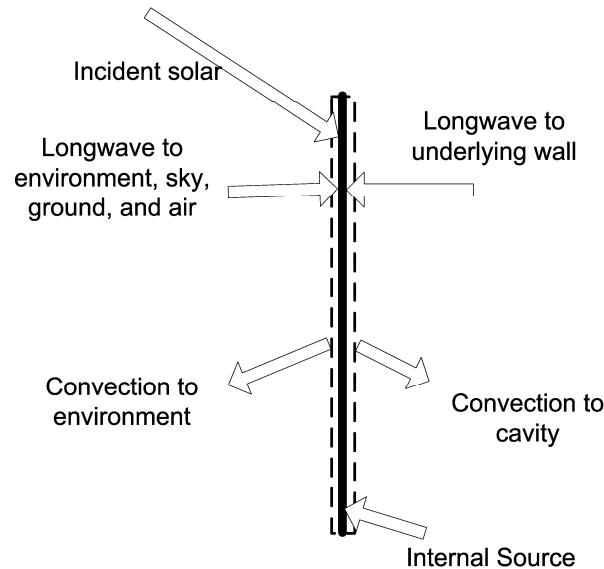


Figure 31. Baffle Surface Heat Balance

The heat balance on the baffle surface’s control volume is:

$$q''_{asol} + q''_{LWR,Env} + q''_{conv,Env} + q''_{LWR,cav} + q''_{conv,cav} + q''_{source} = 0 \quad (126)$$

where:

$q''_{\alpha sol}$ is absorbed direct and diffuse solar (short wavelength) radiation heat flux.

$q''_{LWR,Env}$ is net long wavelength (thermal) radiation flux exchange with the air and surroundings.

$q''_{conv,Env}$ = surface convection flux exchange with outside air.

$q''_{LWR,cav}$ is net long wavelength (thermal) radiation flux exchange with the outside face of the underlying surface(s).

$q''_{conv,cav}$ = surface convection flux exchange with cavity air.

q''_{source} is a source/sink term that accounts for energy exported out of the control volume when the baffle is a hybrid device such as a photovoltaic panel.

All terms are positive for net flux to the baffle. Each of these heat balance components is introduced briefly below.

External SW Radiation

$q''_{\alpha sol}$ is calculated using procedures presented elsewhere in this manual and includes both direct and diffuse incident solar radiation absorbed by the surface face. This is influenced by location, surface facing angle and tilt, shading surfaces, surface face material properties, weather conditions, etc. The baffle blocks all shortwave radiation from reaching the underlying surface.

External LW Radiation

$q''_{LWR,Env}$ is a standard radiation exchange formulation between the surface, the sky, the ground, and the atmosphere. The radiation heat flux is calculated from the surface absorptivity, surface temperature, sky, air, and ground temperatures, and sky and ground view factors. Radiation is modeled using linearized coefficients. The baffle blocks all longwaver radiation.

External Convection

$q''_{conv,Env}$ is modeled using the classical formulation: $q''_{conv} = h_{co}(T_{air} - T_o)$ where h_{co} , is the convection coefficient. The h_{co} is treated in the same way as an outside face with ExteriorEnvironment conditions. In addition, when it is raining outside, we assume the baffle gets wet and model the enhanced surface heat transfer using a large value for h_{co} .

Cavity LW Radiation

$q''_{LWR,cav}$ is a standard radiation exchange formulation between the baffle surface and the underlying heat transfer surface located across the cavity. Radiation is modeled using linearized coefficients.

Cavity Convection

$q''_{conv,cav}$ is modeled using the classical formulation: $q''_{conv} = h_{cp}(T_{air} - T_o)$ where h_{cp} , is the convection coefficient. The value for h_{cp} is obtained from correlations used for window gaps from ISO (2003) standard 15099.

Substituting models into (113) and solving for $T_{s,baff}$ yields the following equation:

$$T_{s,baff} = \frac{(I_s \alpha + h_{co} T_{amb} + h_{r,atm} T_{amb} + h_{r,sky} T_{sky} + h_{r,gnd} T_{amb} + h_{r,cav} T_{so} + h_{c,cav} T_{a,cav} + q''_{source})}{(h_{co} + h_{r,air} + h_{r,sky} + h_{r,gnd} + h_{r,cav} + h_{c,cav})} \quad (127)$$

where,

I_s is the incident solar radiation of all types [W/m^2],

α is the solar absorptivity of the baffle [dimensionless],

$h_{r,atm}$ is the linearized radiation coefficient for the surrounding atmosphere [$W/m^2 \cdot K$],

T_{amb} is the outdoor drybulb from the weather data, also assumed for ground surface [$^{\circ}C$],

$h_{r,sky}$ is the linearized radiation coefficient for the sky [$W/m^2 \cdot K$],

T_{sky} is the effective sky temperature [$^{\circ}C$],

$h_{r,gnd}$ is the linearized radiation coefficient for the ground [$W/m^2 \cdot K$],

$h_{r,cav}$ is the linearized radiation coefficient for the underlying surface [$W/m^2 \cdot K$],

T_{so} is the temperature of the outside face of the underlying heat transfer surface [$^{\circ}C$],

h_{co} is the convection coefficient for the outdoor environment [$W/m^2 \cdot K$],

$h_{c,cav}$ is the convection coefficient for the surfaces facing the plenum [$W/m^2 \cdot K$], and

$T_{a,cav}$ is the drybulb temperature for air in the cavity [$^{\circ}C$].

Cavity Heat Balance

The *cavity* is the volume of air located between the baffle and the underlying heat transfer surface. The cavity air is modeled as well-mixed. The uniform temperature of the cavity air, $T_{a,cav}$, is determined by formulating a heat balance on a control volume of air as diagrammed below.

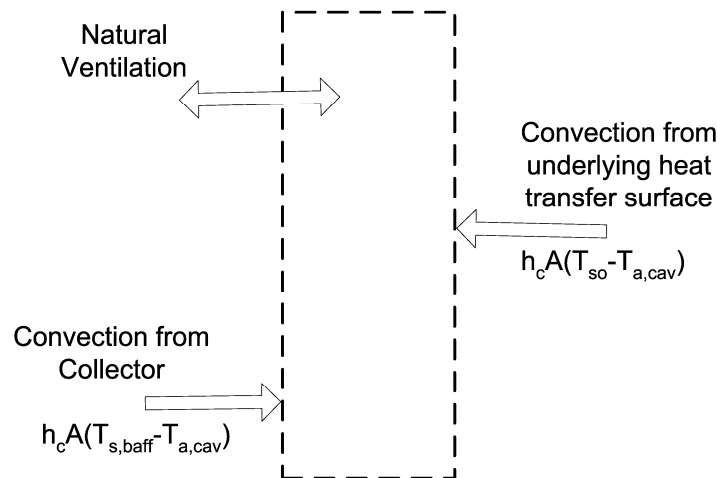


Figure 32. Cavity Air Heat Balance

The heat balance on the cavity air control volume is:

$$\dot{Q}_{vent} + \dot{Q}_{co} + \dot{Q}_{c,baff} = 0 \quad (128)$$

where,

\dot{Q}_{vent} is the net rate of energy added from natural ventilation – where outdoor ambient air exchanges with the cavity air.

\dot{Q}_{co} is the net rate of energy added by surface convection heat transfer with the underlying surface.

$\dot{Q}_{c,baff}$ is the net rate of energy added by surface convection heat transfer with the collector.

And substituting into (128) yields the following equation:

$$T_{a,cav} = \frac{(h_{c,cav}AT_{so} + \dot{m}_{vent}c_pT_{amb} + h_{c,cav}AT_{s,baff})}{(h_{c,cav}A + \dot{m}_{vent}c_p + h_{c,cav}A)}$$

where,

\dot{m}_{vent} is the air mass flow from natural forces [kg/s]

Modeling natural ventilation air exchanges in a general way is challenging. Simplistic engineering models are used to model \dot{m}_{vent} resulting from natural buoyancy and wind forces. Reasoning that the configuration is similar to single-side natural ventilation, we elect to use correlations for natural ventilation presented as equations (29) and (30) in Chapter 26. of ASHRAE HOF (2001).

$$\dot{m}_{vent} = \rho \dot{V}_{tot}$$

where,

ρ is the density of air [kg/m³], and

$\dot{V}_{tot} = \dot{V}_{wind} + \dot{V}_{thermal}$ is the total volumetric flow rate of air ventilating in and out of the cavity.

$$\dot{V}_{wind} = C_v A_m U_\infty$$

$$\dot{V}_{thermal} = C_D A_{in} \sqrt{2g\Delta H_{NPL} (T_{a,cav} - T_{amb}) / T_{a,cav}} \quad (\text{if } T_{a,cav} > T_{amb})$$

$$\dot{V}_{thermal} = C_D A_{in} \sqrt{2g\Delta H_{NPL} (T_{amb} - T_{a,cav}) / T_{amb}} \quad (\text{if } T_{amb} > T_{a,cav} \text{ and baffle is vertical})$$

C_v is the effectiveness of the openings that depends on opening geometry and the orientation with respect to the wind. ASHRAE HoF (2001) indicates values ranging from 0.25 to 0.6. This value is available for user input.

C_D is the discharge coefficient for the opening and depends on opening geometry. This value is available for user input.

Mass continuity arguments lead to modeling the area of the openings as one half of the total area of the openings, so we have:

$$A_{in} = \frac{A}{2}$$

g is the gravitational constant taken as 9.81 [m/s²].

ΔH_{NPL} is the height from midpoint of lower opening to the Neutral Pressure Level. This is value is available for user input.

If the cavity is horizontal and $T_{amb} > T_{a,cav}$ then $\dot{V}_{thermal} = 0$ because this is a stable situation.

Underlying Heat Transfer Surface

The exterior baffle and cavity are applied to the outside of a heat transfer surface. This surface is modeled using the usual EnergyPlus methods for handling heat capacity and transients – typically the CTF method. These native EnergyPlus heat balance routines are used to calculate T_{so} . The exterior baffle and cavity system is coupled to the underlying surface using the SurfaceProperty:OtherSideConditionsModel mechanism. The exterior naturally vented cavity model provides values for $h_{r,cav}$, $T_{s,baff}$, $h_{c,cav}$, and $T_{a,cav}$ for use with the heat balance model calculations for the outside face of the underlying surface (described elsewhere in this manual).

Solar and Shading Calculations

The exterior vented cavity model uses standard EnergyPlus surfaces in order to take advantage of the detailed solar and shading calculations. Solar radiation incident on the surface includes beam and diffuse radiation, as well as radiation reflected from the ground and adjacent surfaces. Shading of the collector by other surfaces, such as nearby buildings or trees, is also taken into account.

Local Wind Speed Calculations

The outdoor wind speed affects terms used in modeling. The wind speed in the weather file is assumed to be measured at a meteorological station located in an open field at a height of 10 m. To adjust for different terrain at the building site and differences in the height of building surfaces, the local wind speed is calculated for each surface.

The wind speed is modified from the measured meteorological wind speed by the equation (ASHRAE 2001):

$$U_{\infty} = V_{met} \left(\frac{\delta_{met}}{z_{met}} \right)^{a_{met}} \left(\frac{z}{\delta} \right)^a \quad (129)$$

where z is the height of the centroid of the system, z_{met} is the height of the standard meteorological wind speed measurement, and a and δ are terrain-dependent coefficients. δ is the boundary layer thickness for the given terrain type. The values of a and δ are shown in the following tables:

Table 15. Terrain-Dependent Coefficients (ASHRAE 2001).

Terrain	Description	Exponent, a	Layer Thickness, δ (m)
1	Flat, open country	0.14	270
2	Rough, wooded country	0.22	370
3	Towns and cities	0.33	460
4	Ocean	0.10	210
5	<i>Urban, industrial, forest</i>	<i>0.22</i>	<i>370</i>

The exterior vented cavity can be defined such that it has multiple underlying heat transfer surfaces. The centroid heights for each surface are area-weighted to determine the average height for use in the local wind calculation.

Convection Coefficients

Exterior cavity modeling requires calculating up to three different coefficients for surface convection heat transfer. These coefficients are defined in the classic way by:

$$h_c = \frac{T_{air} - T_{surf}}{q_{conv}''}$$

First, h_{co} is the convection coefficient for the baffle surface facing the outdoors. It is modeled in exactly the same way as elsewhere in EnergyPlus and will depend on the user setting for Outside Convection Algorithm – Outside Surface Heat Balance entry elsewhere in this document.

Second, $h_{c,cav}$ is the convection coefficient for baffle surfaces facing the cavity. This coefficient is applied to both the baffle and the underlying surface. The convection coefficient is modeled in the same way used in EnergyPlus to model air gaps in windows. These correlations vary by Rayleigh number and surface tilt and are based on the work of various research including Hollands et. al., Elsherbiny et. al., Wright, and Arnold. The formulations are documented in ISO (2003) standard 15099. The routines were adapted from Subroutine NusseltNumber in WindowManager.f90 (by F. Winkelmann), which itself was derived from Window5 subroutine “nusselt”.

Radiation Coefficients

Exterior vented cavity modeling requires calculating up to four different linearized coefficients for radiation heat transfer. Whereas radiation calculations usually use temperature raised to the fourth power, this greatly complicates solving heat balance equations for a single temperature. Linearized radiation coefficients have the same units and are used in the same manner as surface convection coefficients and introduce very little error for the temperature levels involved.

The radiation coefficient, $h_{r,cav}$, is used to model thermal radiation between the collector surface and the outside face of the underlying heat transfer surface. We assume a view factor of unity. It is calculated using:

$$h_{r,cav} = \sigma_{SB} e_{baff} e_{so} \frac{(T_{s,baff}^4 - T_{so}^4)}{(T_{s,baff} - T_{so})} \quad (130)$$

where,

all temperatures are converted to Kelvin,

σ_{SB} is the Stefan-Boltzmann constant,

e_{baff} is the longwave thermal emittance of the baffle, and

e_{so} is the longwave thermal emittance of the underlying heat transfer surface.

The three other coefficients, $h_{r,atm}$, $h_{r,sky}$, and $h_{r,gnd}$ are used elsewhere in EnergyPlus for the outside face surface heat balance and are calculated in the same manner as equation (130). [This is accomplished by calling subroutine InitExteriorConvectionCoeffs in the file HeatBalanceConvectionCoeffs.f90.]

References

- ASHRAE HOF 2001. 2001 ASHRAE Fundamentals Handbook. American Society of Heating Refrigeration and Air-Conditioning Engineers. Atlanta GA.
- ISO. 2003. ISO 15099:2003. Thermal performance of windows, doors, and shading devices – Detailed calculations. International Organization for Standardization.

Green Roof Model (EcoRoof)

Overview

The input object Material:RoofVegetation provides a model for green roofs (aka ecoroofs or vegetated roofs) that are becoming increasingly common for both new and retrofit buildings. There is widespread recognition and a growing literature of measured data that suggest green roofs can reduce building energy consumption. Currently, however, there are few design tools available to assist developers and architects in assessing the likely magnitude of energy savings associated with various implementation options (e.g., soil type/depth, irrigation options, plant type). As a result there is a significant need for a quantitative and physically-based building energy simulation tool that represents the effects of green roof constructions. Such a tool would facilitate more rapid spread of green roof technologies and make it possible to account for green roof benefits in state energy codes and related energy efficiency standards such as LEED.

In response to the need for green roof design tools a computational model of the heat transfer processes involved on a vegetated roof has been developed. This model accounts for:

- long wave and short wave radiative exchange within the plant canopy,
- plant canopy effects on convective heat transfer,
- evapotranspiration from the soil and plants, and
- heat conduction (and storage) in the soil layer

The ability to track moisture-dependent thermal properties is not implemented yet due to stability issues in the CTF scheme, but is under development for use with the finite difference solution scheme made available in EnergyPlus starting in version 2.

As implemented in EnergyPlus the green roof module allows the user to specify “ecorooF” as the outer layer of a rooftop construction using a “Material:RoofVegetation” object. The user can then specify various aspects of the green roof construction including growing media depth, thermal properties, plant canopy density, plant height, stomatal conductance (ability to transpire moisture), and soil moisture conditions (including irrigation).

The model formulation includes the following:

- simplified moisture balance that allows precipitation, irrigation, and moisture transport between two soil layers (top and root zone).

- soil and plant canopy energy balance based on the Army Corps of Engineers' FASST vegetation models (Frankenstein and Koenig), drawing heavily from BATS (Dickenson et al.) and SiB (Sellers et al.).
- soil surface (T_g) and foliage (T_f) temperature equations are solved simultaneously each time step, inverting the CTF to extract heat flux information for the energy balance calculation.

The detailed energy balance analysis and resulting equations, being rather complicated, are summarized here. The interested reader is referred to the FASST documentation cited herein for the complete development. The end result is a set of two simultaneous equations for temperature—one for the soil surface and the other for the foliage.

Green Roof Model Description

As with a traditional roof, the energy balance of a green roof is dominated by radiative forcing from the sun. This solar radiation is balanced by sensible (convection) and latent (evaporative) heat flux from soil and plant surfaces combined with conduction of heat into the soil substrate. This energy balance is illustrated in Figure 33. The variables introduced in this figure are defined in the equations that follow.

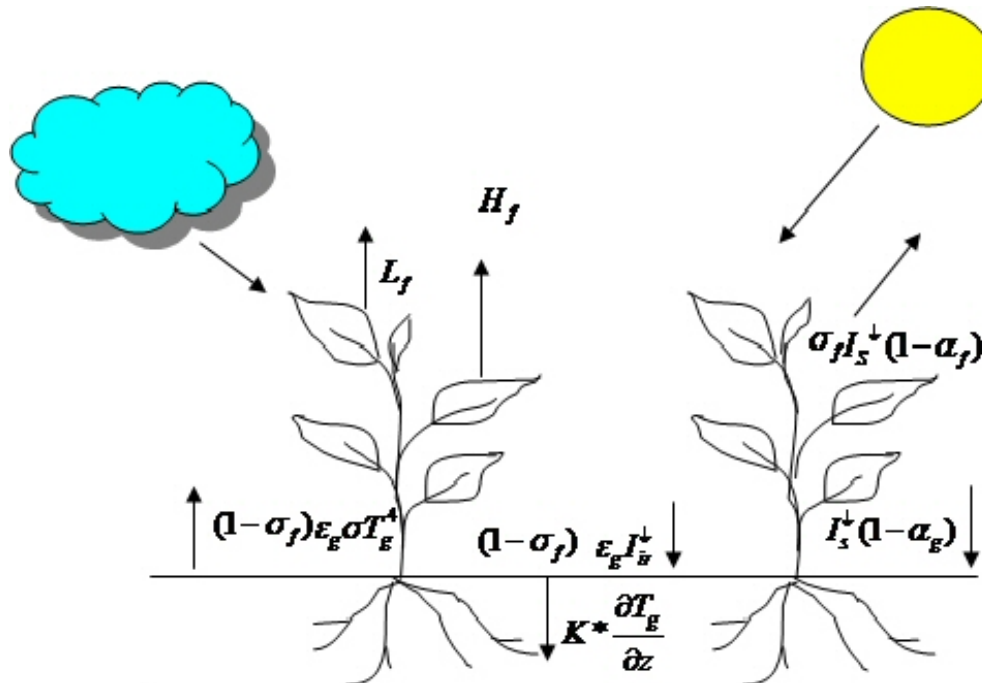


Figure 33. The Energy Balance for a Green Roof.

The energy budget analysis follows the Fast All Season Soil Strength (FASST) model developed by Frankenstein and Koenig for the US Army Corps of Engineers. FASST was developed, in part, to determine the ability of soils to support manned and unmanned vehicles and personnel movement. In order to accomplish this, however, FASST tracks the energy and moisture balance (including ice and snow) within a vegetated soil. It is a one-dimensional model that draws heavily from other plant canopy models including BATS (Dickinson et al.) and SiB (Sellers et al.). We have implemented FASST here with only a few modifications to adapt it for use with a relatively thin soil layer. The sign convention used assumes all heat fluxes are positive when energy is absorbed into the layer.

In the following discussion this energy budget is divided into a budget for the foliage layer (F_f) and a budget for the ground surface (F_g). The various parameterizations for latent and sensible heat flux are described in some detail and then the equation set is reduced to the

simultaneous solution of two equations involving the temperatures of the foliage and ground surface.

Energy budget in the foliage layer

The foliage energy balance is given by:

$$F_f = \sigma_f \left[I_S^\downarrow (1 - \alpha_f) + \varepsilon_f I_{ir}^\downarrow - \varepsilon_f \sigma T_f^4 \right] + \frac{\sigma_f \varepsilon_g \varepsilon_f \sigma}{\varepsilon_1} (T_g^4 - T_f^4) + H_f + L_f$$

In addition to convective and sensible heat transfer this equation accounts for both the short and long wave radiation absorbed by the vegetation, including the effects of multiple reflections. The sensible and latent heat flux terms (H_f and L_f) are somewhat complicated and therefore discussed in some detail below.

Sensible heat flux in the foliage layer

The sensible heat transfer between the leaf surface and near-canopy air (H_f) is influenced by the temperature difference between them, wind speed, and Leaf Area Index (LAI). The Leaf Area Index is the dimensionless ratio of the projected leaf area for a unit ground area (Oke). In contrast fractional vegetative cover (σ_f) is the ratio of shaded ground surface to total ground surface area. The sensible heat flux is given by:

$$H_f = (1.1 * LAI \rho_{af} C_{p,a} C_f W_{af}) * (T_{af} - T_f)$$

In this equation the constant 1.1 accounts for heat transfer from the stems, twigs and limbs (Deardorff). The properties of air near the foliage are modeled using the average from the foliage and instrument conditions:

$$\rho_{af} = 0.5(\rho_a + \rho_f)$$

where ρ_a is the density of air at the instrument height and ρ_f is the density of air at the leaf temperature. The air temperature within the foliage is estimated by:

$$T_{af} = (1 - \sigma_f)(T_a) + \sigma_f (0.3T_a + 0.6T_f + 0.1T_g)$$

where, T_a is the air temperature at the instrument height in Kelvin T_f , is leaf temperature in Kelvin and T_g , is the ground surface temperature in Kelvin. The foliage wind speed is estimated as:

$$W_{af} = 0.83 \sigma_f W \sqrt{C_{hn}^f} + (1 - \sigma_f) W$$

Here W is the larger of 2.0 m/s or the actual wind speed above the canopy (Hughes et al.) and C_{hn}^f is the transfer coefficient at near-neutral atmospheric stability conditions:

$$C_{hn}^f = K_v^2 \cdot \left(\ln \left(\frac{Z_a - Z_d}{Z_o^f} \right) \right)^{-2}$$

where K_v , is von Karmen's constant (0.4), Z_a is the instrument height, Z_d is the zero displacement height in meters (height above soil within which the wind speed is effectively zero), and Z_o^f is the foliage roughness length scale (m). The formulations for zero displacement height, roughness length are based on Balick et al.:

$$Z_d = 0.701Z_f^{0.979}$$

$$Z_o = 0.131Z_f^{0.997}$$

Finally, the bulk transfer coefficient as defined by Deardorff is given by:

$$C_f = 0.01 * \left(1 + \frac{0.3(m/s)}{W_{af}(m/s)} \right)$$

Latent heat flux in the foliage layer

The process of water loss through plant respiration is known as transpiration. It is controlled by the closing and opening of stomata - the intercellular openings between to epidermal (guard) cells (Gates). The resistance to the diffusion of water vapor from these spaces into the atmosphere is called stomatal resistance. It depends on factors such as light intensity, soil moisture content and vapor pressure difference between inside leaf and the outside atmosphere. It is measured in units of s/m and is formulated as:

$$r_s = \frac{r_{s,\min}}{LAI} \cdot f_1 \cdot f_2 \cdot f_3$$

Here, $r_{s,\min}$ is the minimum stomatal resistance. The actual stomatal resistance at any time is proportional to this minimum resistance and inversely proportional to LAI. The stomatal resistance is further modified by fractional multiplying factors that relate to incoming solar radiation and atmospheric moisture. As found in Frankenstein and Koenig the inverses of the multiplying factors f_1 , f_2 , and f_3 are given by:

$$\frac{1}{f_1} = \min \left[1, \frac{0.004 * I_s^\downarrow + 0.005}{0.81 * (0.004 * I_s^\downarrow + 1)} \right]$$

$$\frac{1}{f_2} = \begin{cases} 0 & \text{when } \theta_r > \bar{\theta} \\ \frac{\bar{\theta} - \theta_r}{\theta_{\max} - \theta_r} & \text{when } \theta_r \leq \bar{\theta} \leq \theta_{\max} \end{cases}$$

$$\frac{1}{f_3} = \exp \left[-g_d (e_{f,\text{sat}} - e_a) \right]$$

Here, θ_r , is the residual moisture content (defined as the amount of moisture in soil when plants begin to wilt), θ_{\max} is the maximum moisture content (defined as the maximum amount of moisture a particular type of soil can hold and above which run off occurs), and $\bar{\theta}$ is the average soil moisture in the root zone. The residual moisture content is typically around $0.01 \text{ m}^3/\text{m}^3$ (Frankenstein and Koenig). The maximum moisture content depends upon the soil, but generally varies from 0.3 to $0.6 \text{ m}^3/\text{m}^3$ (Guymon et al.). In the expression for f_3 , g_d is a plant specific characteristic that is only non-zero for trees, $e_{f,\text{sat}}$ is the saturated vapor pressure at the leaf temperature, and e_a is the air vapor pressure.

Resistance to moisture exchange offered by the boundary layer formed on the leaf surface is known as aerodynamic resistance. It is measured in units of (s/m) and is influenced by wind speed, surface roughness and stability of the atmosphere (Oke). It is formulated as:

$$r_a = \frac{1}{c_f W_{af}}$$

The combined effect of aerodynamic and stomatal resistances to vapor diffusion is integrated into a foliage surface wetness factor:

$$r'' = \frac{r_a}{r_a + r_s}$$

This surface wetness factor is simply a ratio of the aerodynamic resistance to the total resistance. When the aerodynamic resistance is small the wetness factor approaches zero (leaf surfaces remain dry as surface moisture is readily evaporated). As the aerodynamic resistance increases in importance relative to stomatal resistance the wetness factor approaches 1.0 (moisture readily travels to the leaf surfaces, but is not easily evaporated).

The latent heat flux is then given by:

$$L_f = l_f * LAI \rho_{af} C_f W_{af} r'' (q_{af} - q_{f,sat})$$

Here l_f , is the latent heat of vaporization (J/kg), $q_{f,sat}$ is the saturation mixing ratio at the leaf surface temperature, and q_{af} is the mixing ratio of the air within the canopy. As developed in Frankenstein and Koenig the mixing ratio within the canopy can be determined from:

$$q_{af} = \left[\frac{(1 - \sigma_f) q_a + \sigma_f (0.3 q_a + 0.6 q_{f,sat} r'' + 0.1 q_{f,sat} M_g)}{1 - \sigma_f [0.6(1 - r'') + 0.1(1 - M_g)]} \right]$$

where the factor M_g (ranging from 0 to 1) is the ratio of volumetric moisture content to the porosity of the soil (Koenig). The latent heat of vaporization (l_f) is the amount of energy required to convert a unit mass of water to vapor. It is measured in units of J/kg and is inversely proportional to the temperature. From Henderson-Sellers it is estimated as:

$$l_f = 1.91846 * 10^6 \left[\frac{T_f}{T_f - 33.91} \right]^2$$

Soil Energy budget

The energy budget at the soil surface is mainly influenced by the soil thermal properties, the amount of foliage coverage (σ_f) and the amount of moisture in the soil. If the soil surface is densely covered the diurnal range of surface temperature is small. In the soil energy budget the heat released or gained due to phase changes of soil water, precipitation heat flux and heat flux due to vertical transport of water in the soil are ignored. Future refinements to this model will incorporate these phenomena. The sign convention followed here is the same as above (heat flux into the soil is positive). The overall energy balance at the soil surface (as given in Frankenstein and Koenig) is:

$$F_g = (1 - \sigma_f) \left[I_s^\downarrow (1 - \alpha_g) + \varepsilon_g I_{ir}^\downarrow - \varepsilon_g T_g^4 \right] - \frac{\sigma_f \varepsilon_g \varepsilon_f \sigma}{\varepsilon_1} (T_g^4 - T_f^4) + H_g + L_g + K * \frac{\partial T_g}{\partial z}$$

As with the energy equation for the foliage this equation represents sensible heat flux (H_g), latent heat flux (L_g) and the multiple reflections associated with long and short wave radiation. The final term on the right side gives the conduction of heat into the soil substrate.

Sensible heat flux in the soil layer

Sensible heat flux between the soil surface and air in its vicinity is dependent on the temperature difference between them and the wind speed within the canopy. It is given as

$$H_g = \rho_{ag} C_{p,a} C_h^g W_{af} (T_{af} - T_g)$$

where C_h^g is the bulk transfer coefficient and ρ_{ag} is the density of air near the soil surface (kg/m^3) given by:

$$\rho_{ag} = \frac{\rho_a + \rho_g}{2}$$

Here ρ_g is the density of air at the ground surface temperature

The bulk transfer coefficient is given as the linear combination of bulk transfer coefficient near ground (C_{hn}^g) and near foliage-atmosphere interface (C_{hn}^f) multiplied by the stability factor (Γ_h) and is formulated as:

$$C_h^g = \Gamma_h \left[(1 - \sigma_f) C_{hn}^g + \sigma_f C_{hn}^f \right]$$

The ground and foliage bulk transfer coefficients, in turn, are given by:

$$C_{hn}^g = r_{ch}^{-1} \left[\frac{K_v}{\ln \left(\frac{Z_a}{Z_o^g} \right)} \right]^2$$

And

$$C_{hn}^f = \left[\frac{K_v}{\ln \left(\frac{Z_a - Z_d}{Z_o^f} \right)} \right]^2$$

where Z_o^g and Z_o^f are the ground and foliage roughness lengths, r_{ch} is turbulent Schmidt number (0.63), and K_v is the von Karman constant (0.4).

The condition of the atmosphere (Γ_h) is determined as stable or unstable based on the sign of the bulk Richardson number:

$$R_{ib} = \frac{2gZ_a (T_{af} - T_g)}{(T_{af} + T_g) W_{af}^2}$$

The atmospheric stability factor is then given by Businger and Lumley and Panofsky as:

$$\Gamma_h = \begin{cases} \frac{1.0}{(1.0 - 16.0R_{ib})^{0.5}} & \text{for } R_{ib} < 0 \\ \frac{1.0}{(1.0 - 5.0R_{ib})} & \text{for } R_{ib} > 0 \end{cases}$$

Latent heat flux in the soil layer

Removal of water vapor from the soil surface depends on the difference between the mixing ratio of the soil surface and air and the wind speed within the canopy. The resulting latent heat flux is then given by:

$$L_R = C_e^g l_g W_{af} \rho_{ag} (q_{af} - q_g)$$

Here C_e^g is the bulk transfer coefficient, l_g is the latent heat of vaporization at the ground surface temperature, q_{af} is the mixing ratio at the foliage-atmosphere interface, and q_f is the mixing ratio at the ground surface, given by:

$$q_g = M_g q_{g,sat} + (1 - M_g) q_{af}$$

The bulk transfer coefficient for latent heat exchange is analogous to that for sensible heat exchange and is given by:

$$C_e^g = \Gamma_e \left[(1 - \sigma_f) C_{en}^g + \sigma_f C_{hn}^f \right]$$

where C_{en}^g is the near ground bulk transfer coefficient for Latent heat flux and Γ_e is the latent heat exchange stability correction factor (assumed to be the same as Γ_h).

Linearization

In order to solve the foliage and soil heat budget equations, the 4th order terms T_f^4 and T_g^4 and mixing ratio terms $q_{g,sat}$ and $q_{f,sat}$ are linearized as given by Deardorff:

$$\left[T_f^{(n+1)} \right]^4 = \left[T_f^n \right]^4 + 4 \left[T_f^n \right]^3 \left[T_f^{n+1} - T_f^n \right]$$

$$\left[T_g^{(n+1)} \right]^4 = \left[T_g^n \right]^4 + 4 \left[T_g^n \right]^3 \left[T_g^{n+1} - T_g^n \right]$$

Here T_f^{n+1} and T_g^{n+1} are the current time step leaf and ground surface temperatures in Kelvin. T_f^n and T_g^n are the corresponding temperatures at the previous time step.

The saturation mixing ratio at the ground and leaf surface temperatures are given as:

$$q_{g,sat} \left(T_g^{n+1} \right) = q_{sat} \left(T_g^n \right) + \left(\frac{\partial q_{sat}}{\partial T} \right)_{T_g^n} * \left(T_g^{n+1} - T_g^n \right)$$

$$q_{f,sat}(T_f^{n+1}) = q_{sat}(T_f^n) + \left(\frac{\partial q_{sat}}{\partial T} \right)_{T_f^n} * (T_f^{n+1} - T_f^n)$$

where $q_{sat}(T_g^n)$ is the saturation mixing ratio at the previous time step and is formulated as given in Garratt:

$$q_{sat}(T_g^n) = \frac{0.622e^*(T_g^n)}{P - e^*(T_g^n)}$$

Here the saturation vapor pressure e^* (Pa) is evaluated at the ground temperature from the previous time step (T_g^n) as:

$$e^* = 611.2 \exp \left[17.67 \left(\frac{T_g^n - 273.15}{T_g^n - 29.65} \right) \right]$$

The derivative of saturation mixing ratio at the previous time step is given by:

$$\frac{dq^*}{dT_g^n} = \left[\frac{0.622 * P}{(P - 0.378 * e^*)^2} \right] \left(\frac{de^*}{dT_g^n} \right)$$

Here, the derivative of the saturation vapor pressure can be calculated from the Clausius-Clapeyron equation:

$$\frac{de^*}{dT_g^n} = \frac{l_g * e^*(T_g^n)}{(R_v * (T_g^n)^2)}$$

Where R_v is the gas constant for water vapor and l_g is the latent heat of vaporization at the soil surface temperature.

The corresponding saturation mixing ratio relations for the leaf surfaces can be obtained by replacing T_g with T_f in the above relations.

Final Equations

After linearization the final equations are of the form:

$$C_1^f + C_2^f T_g + C_3^f T_f = 0$$

$$C_1^g + C_2^g T_g + C_3^g T_f = 0$$

The coefficients in these equations result from the direct combination of the equations from the above development. The interested reader is directed to the papers by Frankenstein and Koenig for the complete and somewhat complicated expressions.

This final set of equations is then solved simultaneously to obtain T_g and T_f . One key difference in our implementation of the FASST algorithm is that the conduction terms in the

equations for C_1^g and C_2^g are solved by inverting the Conduction Transfer Functions (CTF) within the EnergyPlus solution scheme.

Green Roof Nomenclature

C_1, C_2, C_3	=	coefficients in linearized temperature equations
C_e^g	=	latent heat flux bulk transfer coefficient at ground layer
C_f	=	bulk heat transfer coefficient
C_h^g	=	sensible heat flux bulk transfer coefficient at ground layer
C_{hn}^f	=	near-neutral transfer coefficient at foliage layer
C_{hn}^g	=	near-neutral transfer coefficient at ground layer
$C_{p,a}$	=	specific heat of air at constant pressure (1005.6 J/kg k)
e^*	=	saturation vapor pressure (Pa)
f_1	=	multiplying factor for radiation effect on stomatal resistance
f_2	=	multiplying factor for moisture effect on stomatal resistance
f_3	=	additional multiplying factor for stomatal resistance
F_f	=	net heat flux to foliage layer (W/m^2)
F_g	=	net heat flux to ground surface (W/m^2)
g_d	=	plant specific characteristic related to stomatal resistance
H_f	=	foliage sensible heat flux (W/m^2)
H_g	=	ground sensible heat flux (W/m^2)
I_s^\downarrow	=	total incoming short wave radiation (W/m^2)
$I_{l\gamma}^\downarrow$	=	total incoming long wave radiation (W/m^2)
K_v	=	von Karmen constant (0.4)
l_f	=	latent heat of vaporization at foliage temperature (J/kg)
l_g	=	latent heat of vaporization at ground temperature (J/kg)
L_f	=	foliage latent heat flux (W/m^2)
L_g	=	ground latent heat flux (W/m^2)
LAI	=	leaf area index (m^2/m^2)
M_g	=	moisture saturation factor
q_a	=	mixing ratio for air
q_{af}	=	mixing ratio for air within foliage canopy
$q_{f,sat}$	=	saturation mixing ratio at foliage temperature
$q_{g,sat}$	=	saturation mixing ratio at ground temperature
r_a	=	aerodynamic resistance to transpiration (s/m)
r_s	=	foliage leaf stomatal resistance (s/m)
$r_{s,min}$	=	minimal leaf stomatal resistance (s/m)
r''	=	surface wetness factor
R_{ib}	=	bulk Richardson number
R_v	=	gas constant for water vapor (461.53 J/kgK)
T_a	=	the air temperature at the instrument height (Kelvin)
T_{af}	=	air temperature with in the canopy (Kelvin)
T_f	=	leaf temperature (Kelvin)
T_g	=	ground surface temperature (Kelvin)

W	=	wind speed above canopy (m/s)
W_{af}	=	wind speed within the canopy (m/s)
z	=	height or depth (m)
Z_a	=	instrument height (m)
Z_d	=	displacement height (m)
Z_o^f	=	foliage roughness length scale (m)

Greek letters

α_f	=	albedo (short wave reflectivity) of the canopy
α_g	=	albedo (short wave reflectivity) of ground surface
ε_l	=	$\varepsilon_g + \varepsilon_f - \varepsilon_f \varepsilon_g$
ε_f	=	emissivity of canopy
ε_g	=	emissivity of the ground surface
Γ_h	=	stability factor
ρ_a	=	density of air at instrument height (kg/m^3)
ρ_f	=	density of air at foliage temperature (kg/m^3)
ρ_{af}	=	density of air at foliage temperature (kg/m^3)
ρ_{ag}	=	density of air at ground surface temperature (kg/m^3)
σ	=	the Stefan-Boltzmann constant ($5.699 \times 10^{-8} \text{ W/m}^2 \text{ }^\circ\text{K}^4$)
σ_f	=	fractional vegetation coverage

Subscripts and superscripts

a	=	air
af	=	air within the foliage layer
e	=	latent heat flux term
f	=	foliage surface
g	=	ground surface
h	=	sensible heat flux term
n	=	current time step
n+1	=	future time step
ir	=	infrared (or long-wave)
sat	=	saturation value
S	=	short-wave

References

- ASHRAE. 2005. 2005 ASHRAE Handbook – Fundamentals. Chapter 16, Air flow Around Buildings, Atlanta: American Society of Heating Refrigerating and Air-Conditioning Engineers, Inc.
- Balick, L. R., R. K. Scoggins, and L. E. Link. 1981. Inclusion of a simple vegetation layer in terrain temperature models for thermal IR signature prediction. *IEEE Geoscience and Remote Sensing GE-19(3)*, pp.143-152.
- Businger, J. A. 1966. In 'Arctic Heat Budget and Atmospheric Circulation', Symposium Proceedings, pp. 305-332. The Rand Corporation.
- Deardorff, J.W. 1978. "Efficient Prediction of ground surface temperature and moisture with inclusion of a layer of vegetation", *Journal Geophysical Research*, pp. 1889-1902.

- Dickinson, R.E., A. Henderson-Sellers, P.J. Kennedy, and M.F. Wilson. 1986. Biosphere-Atmosphere Transfer Scheme (BATS) for the NCAR community climate model. NCAR Technical Note, TN-275+STR.
- ECMWF. 2002. European Centre for Medium-Range Weather Forecasts, Integrated Forecast System. Documentation, CY25R1 (Operational implementation 9 April 2002). <http://www.ecmwf.int/research/ifdocs/CY25r1/Physics/Physics-08-03.html>.
- Frankenstein, S., and G. Koenig. 2004. FASST Vegetation Models. U. S. Army Engineer Research and Development Center, Cold regions Research and Engineering Laboratory, ERDC/CRREL Technical Report TR-04-25.
- Frankenstein, S., and G. Koenig. 2004. Fast All-season Soil Strength (FASST). U.S. Army Engineer Research and Development Center, Cold regions Research and Engineering Laboratory, ERDC/CRREL Special Report SR-04-1.
- Garratt, J.R. 1992. *The Atmospheric Boundary Layer*, Cambridge university press.
- Gates, D.M. 1980. *Biophysical Ecology*. New York: Springer-Verlag
- Guymon, G.L., R.L. Berg, and T.V. Hromadka. 1993. *Mathematical Model of Frost Heave and Thaw Settlement in Pavements*. U.S. Army Cold Regions Research and Engineering Laboratory, CRREL Report 93-2.
- Henderson-Sellers, B. 1984. "A New Formula for Latent Heat of Vaporization of water as function of temperature", *Quarterly Journal Royal Meteorological Society*, 10 pp. 1186-1190.
- Hughes, P.A., T.J.L. McComb, A.B. Rimmer, and K.E. Turver. 1993. "A mathematical model for the prediction of temperature of man-made and natural surfaces", *International Journal of Remote Sensing* 14 (7), pp. 1383-1412.
- Koenig, G.G. 1994. *Smart Weapons Operability Enhancement (SWOE) Joint Test and Evaluation (JT and E) Program: Final Report*. Dr. James P. Welch, Joint Test Director, SWOE JT and E, SWOE Report 94-10, Annex D.
- Lumley, J. L. and Panofsky, H. A. 1964. 'The structure of Atmospheric Turbulence'. *Interscience Monographs and Texts in Physics and Astronomy*, Vol. XII. Wiley, New York.
- Oke, T.R. 1987. *Boundary Layer Climates*, University Press, Cambridge
- Sellers, P.J., Y. Mintz, Y.C. Sud, and A. Dalcher. 1986. A simple biosphere model (SiB) for use within general circulation models. *Journal of Atmospheric Science*, 43 (6), pp. 505-532.

Climate, Sky and Solar/Shading Calculations

Climate Calculations

The location of the facility under analysis is critical for the determination of energy consumption, heating/cooling loads, daylighting potential, and a host of other calculations. In EnergyPlus, both external (i.e., weather files supplied from others) and internal (i.e., solar position, design day temperature/humidity/solar profiles) data is used during simulations.

The “Site:Location” input object includes parameters (Latitude, Longitude, Elevation, Timezone) that allow EnergyPlus to calculate the solar position (using Latitude, Longitude and Timezone) for any day of the year as well as supply the standard barometric pressure (using elevation). Solar position modeling is discussed in more detail in both the Sky Radiance and Shading Calculation sections that directly follow this section.

Weather files have hourly or sub-hourly data for each of the critical elements needed during the calculations (i.e., Dry-Bulb Temperature, Dew-Point Temperature, Relative Humidity, Barometric Pressure, Direct Normal Radiation, Diffuse Horizontal Radiation, Total & Opaque Sky Cover, Wind Direction, Wind Speed) as well as some auxiliary data such as Rain or Snow that assist in certain calculational aspects. Weather file excerpts such as might be used in sizing calculations also have this breadth of data. The input object “SizingPeriod:DesignDay” describes design days (meant to mimic ASHRAE design conditions but in a whole day profile) using certain characteristics for the day and then EnergyPlus supplies the remaining portions to complete outdoor conditions needed for program execution. SizingPeriod:DesignDay are perhaps the best objects for sizing equipment as the ASHRAE specified design conditions can be input AND weather files may or may not have the conditions necessary to size equipment properly.

Two other objects, however, can be used at times: SizingPeriod:WeatherFileDays and SizingPeriod:WeatherFileType. With the first of these, one specifies a set of weather file days similar to a RunPeriod (but will be used for the Sizing calculations). In the second, the extreme or typical conditions that are calculated for the weather file during processing can be used by name. Of course, either of these can be used as a measure of usage over small periods.

The ASHRAE Handbook of Fundamentals describes their criteria in creating design condition synopses. “Design data based on dry-bulb temperature represent peak occurrences of the sensible component of ambient outdoor conditions. Design values based on wet-bulb temperature are related to the enthalpy of the outdoor air. Conditions based on dewpoint relate to the peaks of the humidity ratio. The designer, engineer, or other user must decide which set(s) of conditions and probability of occurrence apply to the design situation under consideration.”

In EnergyPlus, the typical design day input includes a “high” and a “low” dry-bulb temperature for the day. As these two temperatures are insufficient to represent a full 24 hour period, the program uses a “range multiplier” profile to represent the full day’s temperatures:

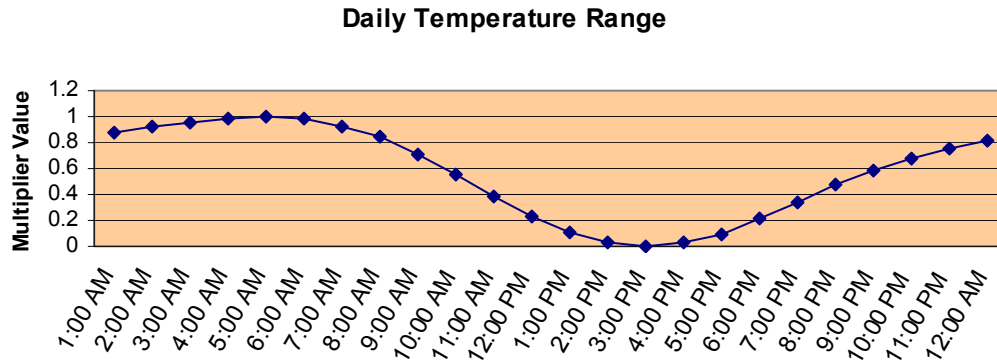


Figure 34. Default Daily Temperature Range Profile

Since this default temperature profile may not be applicable to all locations, the user can give a different profile as part of the design day definition.

Similarly, calculating solar irradiance (solar modeling) is one of the important effects to be accomplished. Several solar models exist with varying complexity. The default model used is the ASHRAE Clear Sky model. The ASHRAE clear sky model described in ASHRAE HOF Chapter 31 can be used to estimate hourly clear-day solar radiation for any month of the year in U.S. or similar temperate climates in the northern hemisphere. EnergyPlus calculations extend the clear sky application to both northern and southern hemispheres.

ASHRAE Clear Sky Solar Model

At the earth's surface on a clear day, direct normal irradiation is represented by

$$\text{Direct Normal Irradiation} = \frac{A}{\exp\left(\frac{B}{\sin \beta}\right)} \quad (131)$$

where

A = apparent solar irradiation at air mass $m = 0$ (Table 16)

B = atmospheric extinction coefficient Table 16)

Values of A and B vary during the year because of seasonal changes in the dust and water vapor content of the atmosphere and because of the changing earth-sun distance. Equation (131) does not give the maximum value of direct normal irradiation that can occur in each month but yields values that are representative of conditions on cloudless days for a relatively dry and clear atmosphere. For very clear atmospheres, direct normal irradiation can be 15% higher than indicated by Equation (131), using values of A and B in Table 16 below.

Table 16. Extraterrestrial Solar Irradiance and Related Data

Note: Data are for 21st day of each month during the base year of 1964.

	I_0 {W/m ² }	Equation of Time {minutes}	Declination {degrees}	A {W/m ² }	B {}	C {}
Jan	1416	-11.2	-20.0	1202	0.141	0.103
Feb	1401	-13.9	-10.8	1187	0.142	0.104
Mar	1381	-7.5	0.0	1164	0.149	0.109

Apr	1356	1.1	11.6	1130	0.164	0.120
May	1336	3.3	20.0	1106	0.177	0.130
Jun	1336	-1.4	23.45	1092	0.185	0.137
Jul	1336	-6.2	20.6	1093	0.186	0.138
Aug	1338	-2.4	12.3	1107	0.182	0.134
Sep	1359	7.5	0.0	1136	0.165	0.121
Oct	1380	15.4	-10.5	1166	0.152	0.111
Nov	1405	13.8	-19.8	1190	0.144	0.106
Dec	1417	1.6	-23.45	1204	0.141	0.103

For locations where clear, dry skies predominate (e.g., at high elevations) or, conversely, where hazy and humid conditions are frequent, values found by using Equation (131) and Table 16 should be multiplied by the clearness numbers in Threlkeld and Jordan (1958), reproduced as Figure 5 in Chapter 33 of the 2007 ASHRAE Handbook—HVAC Applications.

The Clear Sky model usually over estimates the amount of solar radiation available to the building.

Zhang-Huang Solar Model

The Zhang-Huang solar model was developed for initial use in modeling typical meteorological years for China weather sites. This model seems to be good for other locations as well. Using total cloud cover, dry-bulb temperature, relative humidity, and wind speed as the independent variables, the total (global horizontal) solar radiation is estimated by:

$$I = \frac{\left[I_0 \cdot \sin(h) \cdot (c_0 + c_1 \cdot CC + c_2 \cdot CC^2 + c_3 (T_n - T_{n-3}) + c_4 \cdot \varphi + c_5 \cdot V_w) + d \right]}{k}$$

Where

I = estimated hourly solar radiation, W/m^2

I_0 = global solar constant, $1367 W/m^2$

h = solar altitude angle, i.e, the angle between the horizontal and the line to the sun

CC = cloud cover

φ = relative humidity, %

T_n, T_{n-3} = dry-bulb temperature at hours n (current) and $n-3$, respectively

V_w = Wind speed, m/s

$c_0, c_1, c_2, c_3, c_4, c_5, d, k$ = regression coefficients

The constants were determined by analysis from measured data and are as follows:

$c_0 = 37.6865$, $c_1 = 13.9263$, $c_2 = -20.2354$, $c_3 = 0.9695$, $c_4 = -0.2046$, $c_5 = -0.0980$,
 $d = -10.8568$, $k = 49.3112$.

Design Week Specification

A design week (or longer) may be specified in EnergyPlus by selecting a RunPeriod of one week. One-week custom weather files may be constructed if extreme design week

conditions are desired or one can use the specific dates from the EPW weather files that designate the extreme conditions on the weather file. The design week simulation requires that the simulation reach a 'steady-periodic' state for the one week time period. This is achieved manually by increasing the 'number of years of simulation' until the weekly profile does not change from one year to the next. The 'steady-periodic' output for the last week of the simulation represents the design week simulation results.

Using the SizingPeriod:WeatherFileDays object, one could size equipment from a longer period.

Sky Radiance Model

In EnergyPlus the calculation of diffuse solar radiation from the sky incident on an exterior surface takes into account the anisotropic radiance distribution of the sky. For this distribution, the diffuse sky irradiance on a surface is given by

$$AnisoSkyMultiplier_{surface} \cdot DiffuseSolarIrradiance$$

Where

DiffuseSolarIrradiance is the diffuse solar irradiance from the sky on the ground surface is the surface being analyzed

AnisoSkyMultiplier is determined by surface orientation and sky radiance distribution, and accounts for the effects of shading of sky diffuse radiation by shadowing surfaces such as overhangs. It does not account for *reflection* of sky diffuse radiation from shadowing surfaces.

The sky radiance distribution is based on an empirical model based on radiance measurements of real skies, as described in Perez et al., 1990. In this model the radiance of the sky is determined by three distributions that are superimposed (see Figure 35)

- (1) An isotropic distribution that covers the entire sky dome;
- (2) A circumsolar brightening centered at the position of the sun;
- (3) A horizon brightening.

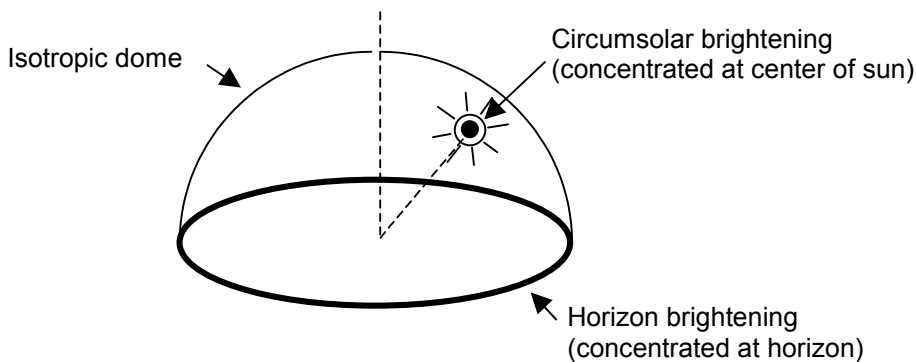


Figure 35. Schematic view of sky showing solar radiance distribution as a superposition of three components: dome with isotropic radiance, circumsolar brightening represented as a point source at the sun, and horizon brightening represented as a line source at the horizon.

The proportions of these distributions depend on the sky condition, which is characterized by two quantities, *clearness factor* and *brightness factor*, defined below, which are determined from sun position and solar quantities from the weather file.

The *circumsolar brightening* is assumed to be concentrated at a point source at the center of the sun although this region actually begins at the periphery of the solar disk and falls off in intensity with increasing angular distance from the periphery.

The *horizon brightening* is assumed to be a linear source at the horizon and to be independent of azimuth. In actuality, for clear skies, the horizon brightening is highest at the horizon and decreases in intensity away from the horizon. For overcast skies the horizon brightening has a negative value since for such skies the sky radiance increases rather than decreases away from the horizon.

Table 17. Variables in Anisotropic Sky Model and Shadowing of Sky Diffuse Radiation

Mathematical variable	Description	Units	FORTTRAN variable
I_{sky}	Solar irradiance on surface from sky	W/m ²	-
I_{horizon}	Solar irradiance on surface from sky horizon	W/m ²	-
I_{dome}	Solar irradiance on surface from sky dome	W/m ²	-
$I_{\text{circumsolar}}$	Solar irradiance on surface from circumsolar region	W/m ²	-
I_h	Horizontal solar irradiance	W/m ²	-
S	Surface tilt	radians	Surface(SurfNum)%Tilt*DegTo Radians
a, b	intermediate variables	-	-
F_1, F_2	Circumsolar and horizon brightening coefficients	-	F1, F2
α	Incidence angle of sun on surface	radians	IncAng
Z	Solar zenith angle	radians	ZenithAng
Δ	Sky brightness factor	-	Delta
ϵ	Sky clearness factor	-	Epsilon
m	relative optical air mass	-	AirMass
I_0	Extraterrestrial solar irradiance	W/m ²	-
I	Direct normal solar irradiance	W/m ²	Material%Thickness
κ	constant = 1.041 for Z in radians	radians ⁻³	-
F_{ij}	Brightening coefficient factors	-	F11R, F12R, etc.
$R_{\text{circumsolar}}$	Shadowing factor for circumsolar radiation	-	SunLitFrac
R_{dome}	Shadowing factor for sky dome radiation	-	DifShdgRatioIsoSky
R_{horizon}	Shadowing factor for horizon radiation	-	DifShdgRatioHoriz
E	Sky radiance	W/m ²	-
θ	Azimuth angle of point in sky	radians	Theta
ϕ	Altitude angle of point in sky	radians	Phi

I_i	Irradiance on surface from a horizon element	W/m^2	-
I_{ij}	Irradiance on surface from a sky dome element	W/m^2	-
SF	Sunlit fraction	-	FracIlluminated
I'	Sky solar irradiance on surface with shadowing	W/m^2	-

Sky Diffuse Solar Radiation on a Tilted Surface

The following calculations are done in subroutine AnisoSkyViewFactors in the SolarShading module.

In the absence of shadowing, the sky formulation described above gives the following expression for sky diffuse irradiance, I_{sky} , on a tilted surface:

$$I_{sky} = I_{horizon} + I_{dome} + I_{circumsolar}$$

where

$$\begin{aligned} I_{horizon} &= \text{irradiance on surface from sky horizon} &&= I_h F_2 \sin S \\ I_{dome} &= \text{irradiance on surface from sky dome} &&= I_h (1 - F_1)(1 + \cos S) / 2 \\ I_{circumsolar} &= \text{irradiance on surface from circumsolar region} &&= I_h F_1 a / b \end{aligned}$$

AnisoSkyMult is then $I_{sky}/\text{DifSolarRad}$.

In the above equations:

I_h = horizontal solar irradiance (W/m^2)

S = surface tilt (radians)

$a = \max(0, \cos \alpha)$

$b = \max(0.087, \cos Z)$

F_1 = circumsolar brightening coefficient

F_2 = horizon brightening coefficient

where

α = incidence angle of sun on the surface (radians)

Z = solar zenith angle (radians).

The brightening coefficients are a function of sky conditions; they are given by

$$F_1 = F_{11}(\epsilon) + F_{12}(\epsilon)\Delta + F_{13}(\epsilon)Z$$

$$F_2 = F_{21}(\epsilon) + F_{22}(\epsilon)\Delta + F_{23}(\epsilon)Z$$

Here the sky brightness factor is

$$\Delta = I_h m / I_o$$

where

m = relative optical air mass

I_o = extraterrestrial irradiance (taken to have an average annual value of $1353 W/m^2$);

and the sky *clearness factor* is

$$\varepsilon = \frac{(I_h + I) / I_h + \kappa Z^3}{1 + \kappa Z^3}$$

where

I = direct normal solar irradiance

κ = 1.041 for Z in radians

The factors F_{ij} are shown in the following table.¹

Table 18. F_{ij} Factors as a Function of Sky Clearness Range.

ε Range	1.000-1.065	1.065-1.230	1.230-1.500	1.500-1.950	1.950-2.800	2.800-4.500	4.500-6.200	> 6.200
F_{11}	-0.0083117	0.1299457	0.3296958	0.5682053	0.8730280	1.1326077	1.0601591	0.6777470
F_{12}	0.5877285	0.6825954	0.4868735	0.1874525	-0.3920403	-1.2367284	-1.5999137	-0.3272588
F_{13}	-0.0620636	-0.1513752	-0.2210958	-0.2951290	-0.3616149	-0.4118494	-0.3589221	-0.2504286
F_{21}	-0.0596012	-0.0189325	0.0554140	0.1088631	0.2255647	0.2877813	0.2642124	0.1561313
F_{22}	0.0721249	0.0659650	-0.0639588	-0.1519229	-0.4620442	-0.8230357	-1.1272340	-1.3765031
F_{23}	-0.0220216	-0.0288748	-0.0260542	-0.0139754	0.0012448	0.0558651	0.1310694	0.2506212

Shadowing of Sky Diffuse Solar Radiation

Sky diffuse solar shadowing on an exterior surface is calculated as follows in subroutine SkyDifSolarShading in the SolarShading module. The sky is assumed to be a superposition of the three Perez sky components described above.

For the horizon source the following ratio is calculated by dividing the horizon line into 24 intervals of equal length:

$$R_{horiz} = \frac{\text{Irradiance from horizon with obstructions}}{\text{Irradiance from horizon without obstructions}} = \frac{\sum_{i=1}^{24} I_i SF_i}{\sum_{i=1}^{24} I_i}$$

where I_i is the unobstructed irradiance on the surface from the i^{th} interval, SF_i is the sunlit fraction from radiation coming from the i^{th} interval, and the sums are over intervals whose center lies in front of the surface. SF_i is calculated using the beam solar shadowing method as though the sun were located at the i^{th} horizon point. Here

$$I_i = E(\theta_i) d\theta \cos \alpha_i$$

where

$E(\theta_i)$ = radiance of horizon band (independent of θ)

$d\theta = 2\pi/24$ = azimuthal extent of horizon interval (radians)

$\theta_i = 0^\circ, 15^\circ, \dots, 345^\circ$

α_i = incidence angle on surface of radiation from θ_i

The corresponding ratio for the isotropic sky dome is given by

¹ The F_{ij} values in this table were provided by R. Perez, private communication, 5/21/99. These values have higher precision than those listed in Table 6 of Perez et al., 1990.

$$R_{dome} = \frac{\text{Irradiance from dome with obstructions}}{\text{Irradiance from dome without obstructions}} = \frac{\sum_{i=1}^{24} \sum_{j=1}^6 I_{ij} SF_{ij}}{\sum_{i=1}^{24} \sum_{j=1}^6 I_{ij}}$$

where (i,j) is a grid of 144 points (6 in altitude by 24 in azimuth) covering the sky dome, I_{ij} is the unobstructed irradiance on the surface from the sky element at the ij^{th} point, SF_{ij} is the sunlit fraction for radiation coming from the ij^{th} element, and the sum is over points lying in front of the surface. Here

$$I_{ij} = E(\theta_i, \phi_j) \cos \phi_j d\theta d\phi \cos \alpha_{ij}$$

where

$E(\theta_i, \phi_j)$ = sky radiance (independent of θ and ϕ for isotropic dome)

$d\theta = 2\pi/24$ = azimuthal extent of sky element (radians)

$d\phi = (\pi/2)/6$ = altitude extent of sky element (radians)

$\theta_i = 0^\circ, 15^\circ, \dots, 345^\circ$

$\phi_j = 7.5^\circ, 22.5^\circ, \dots, 82.5^\circ$

α_{ij} = incidence angle on surface of radiation from (θ_i, ϕ_j)

Because the circumsolar region is assumed to be concentrated at the solar disk, the circumsolar ratio is

$$R_{circumsolar} = \frac{\text{Irradiance from circumsolar region with obstructions}}{\text{Irradiance from circumsolar without obstructions}} = SF_{sun}$$

where SF_{sun} is the beam sunlit fraction. The total sky diffuse irradiance on the surface with shadowing is then

$$I'_{sky} = R_{horizon} I_{horizon} + R_{dome} I_{dome} + R_{circumsolar} I_{circumsolar}$$

$R_{horizon}$ and R_{dome} are calculated once for each surface since they are independent of sun position.

With shadowing we then have:

$$\text{AnisoSkyMult} = I'_{sky} / \text{DifSolarRad}.$$

Shadowing of Sky Long-Wave Radiation

EnergyPlus calculates the sky long-wave radiation incident on exterior surfaces assuming that the sky long-wave radiance distribution is isotropic. If obstructions such as overhangs are present the sky long-wave incident on a surface is multiplied by the isotropic shading factor, R_{dome} , described above. The long-wave radiation from these obstructions is added to the long-wave radiation from the ground; in this calculation both obstructions and ground are assumed to be at the outside air temperature and to have an emissivity of 0.9.

Shading Module

Shading and Sunlit Area Calculations

When assessing heat gains in buildings due to solar radiation, it is necessary to know how much of each part of the building is shaded and how much is in direct sunlight. As an example, the figure below shows a flat roofed, L-shaped structure with a window in each of the visible sides. The sun is to the right so that walls 1 and 3 and windows a and c are completely shaded, and wall 4 and window d are completely sunlit. Wall 2 and window b are partially shaded. The sunlit area of each surface changes as the position of the sun changes during the day. The purpose of the EnergyPlus shadow algorithm is to compute such sunlit areas. Predecessors to the EnergyPlus shadowing concepts include the BLAST and TARP shadowing algorithms.

The shadow algorithm is based on coordinate transformation methods similar to Groth and Lokmanhekim and the shadow overlap method of Walton.

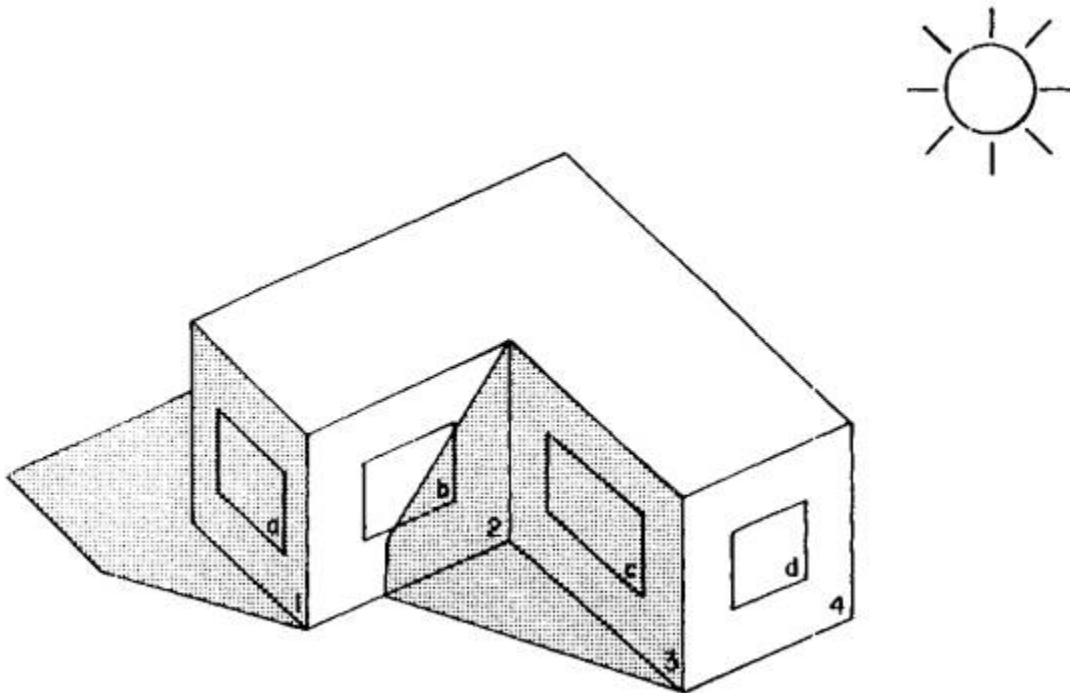


Figure 36. Overall Shadowing Scheme Depiction

Solar Position

Current solar position is described in terms of three direction cosines that are convenient for determining the angle of incidence of the sun's rays on a building surface. The following procedure is used to determine the direction cosines. The values of the solar declination angle, δ , and the equation of time, ϵ , are based on *Astronomical Algorithms*, Meeus. Solar declination is a function of local/site latitude.

The fractional year is calculated, in radians:

$$\gamma = \frac{2\pi}{366} (\text{day_of_year})$$

From this fractional year, the equation of time and solar declination angle are calculated. For each time step (time value = fractional hour), the hour angle is calculated from:

$$\text{HourAngle} = \left(15 \cdot \left(12 - (\text{TimeValue} + \text{EquationOfTime}) \right) \right) + (\text{TimeZoneMeridian} - \text{Longitude})$$

TimeZoneMeridian is the standard meridian for the location's time zone {GMT +/-}.

Solar HourAngle (H) gives the apparent solar time for the current time period (degrees); HourAngle is positive before noon, negative after noon. It is common astronomical practice to express the hour angle in hours, minutes and seconds of time rather than in degrees. You can convert the hour angle displayed from EnergyPlus to time by dividing by 15. (Note that 1 hour is equivalent to 15 degrees; 360° of the Earth's rotation takes place every 24 hours.) The relationship of angles in degrees to time is shown in the following table:

Table 19. Relationship of Angles (degrees) to Time

Unit of Angle	Equivalent time
1 radian	3.819719 hours
1 degree	4 minutes
1 arcmin	4 seconds
1 arcsec	0.066667 seconds

The Solar Altitude Angle (β) is the angle of the sun above the horizontal (degrees). The Solar Azimuth Angle (ϕ) is measured from the North (clockwise) and is expressed in degrees. This is shown more clearly in the following figure.

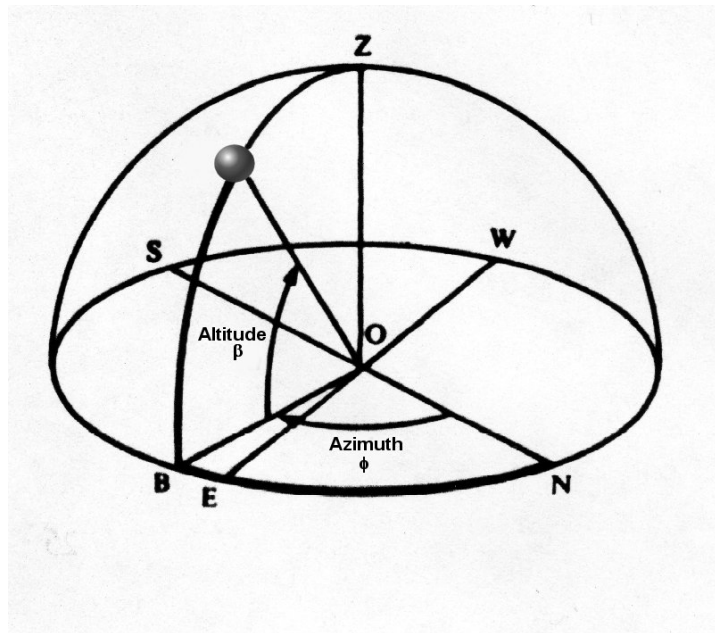


Figure 37. Solar Position Illustration

Surface Geometry

Shadow calculations first require that the building surfaces be described geometrically. Surfaces are described by the coordinates of their vertices in a three dimensional Cartesian coordinate system. This Right-hand coordinate system has the X-axis pointing east, the Y-axis pointing north, and the Z-axis pointing up (see figure below). The azimuth angle (ψ) of a surface is the angle from the north axis to the projection onto the X-Y plane of a normal to the surface (clockwise positive). The surface tilt angle (ϕ) is the angle between the Z-axis and the

normal to the surface. The vertices are recorded in counter-clockwise sequence (as the surface is viewed from outside its zone).

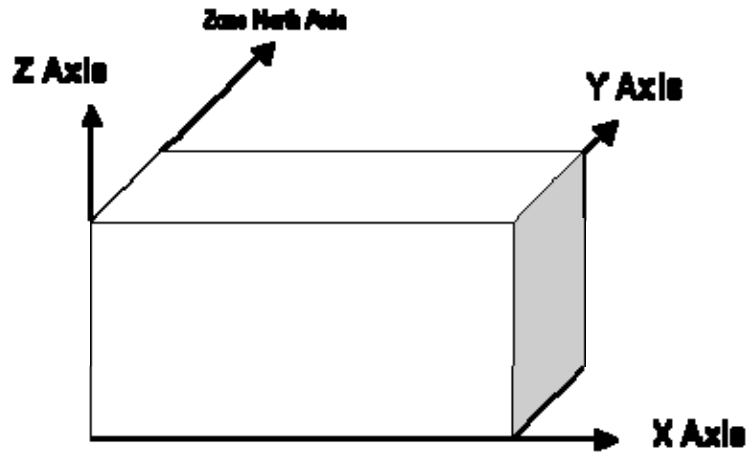


Figure 38. EnergyPlus Coordinate System

The GlobalGeometryRules object specifies to EnergyPlus how the surface vertices will be presented in the input file. Of pertinent interest here is that the user may specify the vertices in either “relative” or “world” coordinates. Regardless of input specifications, when vertices are reported, they are reported in world coordinates, starting at the upper-left-corner (4-sided surface) and are listed counter-clockwise.

Relative Coordinate Transformation

When vertices are specified in “relative” coordinates, there can be a “building” north axis as well as a “zone” north axis. The building north axis/coordinate system is a rotation of ψ_b degrees from the global/world coordinate system. The global coordinates of zone origins are related to the building relative coordinates by:

$$X_{zo} = X_{br} \cdot \cos \psi_b - Y_{br} \cdot \sin \psi_b \quad (132)$$

$$Y_{zo} = Y_{br} \cdot \sin \psi_b - X_{br} \cdot \cos \psi_b \quad (133)$$

$$Z_{zo} = Z_{br} \quad (134)$$

Where

zo – represents Zone Origin

br – represents the Zone Origin as input (relative to building origin)

The zone may also be rotated ψ_z degrees relative to the building coordinates. Origins of zone surfaces are then given relative to the zone coordinate system. The global coordinates of the surface origins are calculated by:

$$X_{so} = X_{zo} + X_{zr} \cdot \cos \psi_z - Y_{zr} \cdot \sin \psi_z \quad (135)$$

$$Y_{so} = Y_{zo} + X_{zr} \cdot \sin \psi_z - Y_{zr} \cdot \cos \psi_z \quad (136)$$

$$X_{so} = X_{zo} + X_{zr} \cdot \cos \psi_z - Y_{zr} \cdot \sin \psi_z \quad (137)$$

A surface azimuth angle relative to the zone coordinate system (ψ_s) is converted to a global azimuth by:

$$\psi = \psi_s + \psi_z + \psi_b \quad (138)$$

The surface tilt angle (ϕ) is not changed by these rotations about the Z-axis.

The coordinates of the surface vertices are given in a coordinate system in the plane of the surface relative to the second vertex as shown for surfaces in Figure 38. The X-axis of the surface coordinate system is a horizontal line through the second vertex. The global coordinates of the surface vertices are given by:

$$X = X_{so} + X_{sr} \cdot \cos \psi - Y_{sr} \cdot \sin \psi \cdot \cos \phi \quad (139)$$

$$Y = Y_{so} + X_{sr} \cdot \sin \psi - Y_{sr} \cdot \cos \psi \cdot \cos \phi \quad (140)$$

$$Z = Z_{so} + Y_{sr} \cdot \sin \phi \quad (141)$$

World Coordinates → Relative Coordinates

Vertices in the global coordinate system can be transformed to the coordinate system relative to a given surface by

$$X' = X - X_{so} \quad (142)$$

$$Y' = Y - Y_{so} \quad (143)$$

$$Z' = Z - Z_{so} \quad (144)$$

$$X_{sr} = -X' \cdot \cos \psi + Y' \cdot \sin \psi \quad (145)$$

$$Y_{sr} = -X' \cdot \sin \psi \cdot \cos \phi + Y' \cdot \cos \psi \cdot \cos \phi + Z' \cdot \sin \phi \quad (146)$$

$$Z_{sr} = -X' \cdot \sin \psi \cdot \sin \phi + Y' \cdot \cos \psi \cdot \sin \phi + Z' \cdot \cos \phi \quad (147)$$

Shadow Projection

All architectural forms are represented by plane polygons. This can give good accuracy even for curved surfaces: a sphere can be approximated by the 20 nodes of an icosahedron with only 3 percent error in the shadow area cast by the sphere. Consider how a solid object, which is composed of a set of enclosing plane polygons, casts a shadow. Figure 39 shows a box shaped structure on a horizontal surface. The structure consists of a top (surface 1) and four vertical surfaces (2 and 3 visible to the observer and 4 and 5 not visible). The sun is positioned behind and to the right of the structure and a shadow is cast onto the horizontal surface (the ground).

Surfaces 1, 4, and 5 are in sunlight; 2 and 3 are in shade. It is possible to think of the structure's shadow as the combination of shadows cast by surfaces 1, 2, 3, 4 and 5 or by 1, 4 and 5, or by surfaces 2 and 3. This last combination of shadow casting surfaces is the

simplest. In the EnergyPlus shadow algorithm every surface is considered to be one of the surfaces that enclose a solid, and only those surfaces that are not sunlit at a given hour are considered shadowing surfaces.

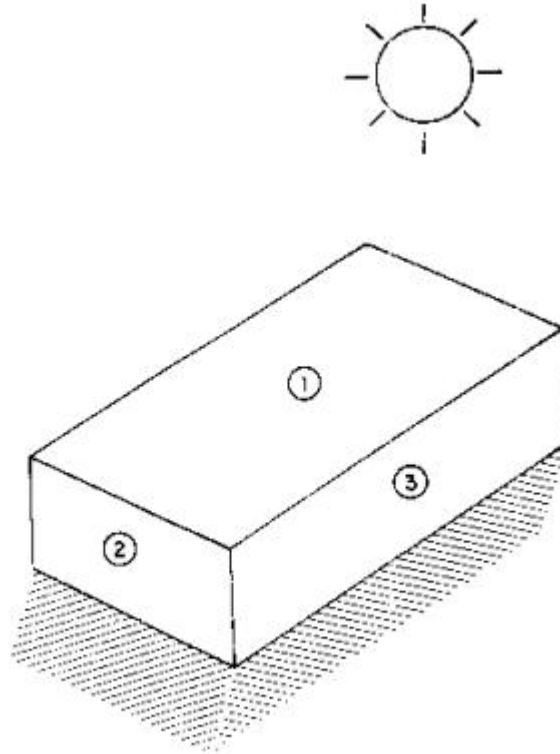


Figure 39. Basic shadowing concept structure

The expressions in equation (147) are the direction cosines of the surface:

$$CW_1 = \sin \psi \cdot \cos \phi \quad (148)$$

$$CW_2 = \cos \psi \cdot \sin \phi \quad (149)$$

$$CW_3 = \cos \phi \quad (150)$$

The cosine of the angle of incidence of the sun's rays on the surface are given by the dot product of surface and sun direction cosines.

$$\cos \theta = CS_1 \cdot CW_1 + CS_2 \cdot CW_2 + CS_3 \cdot CW_3 \quad (151)$$

If $\cos \theta$ is less than zero, the sun is behind the surface.

A shadow is projected from the vertices of the shadowing polygon (SP) along the direction of the sun's rays to the plane of the shadow receiving polygon (RP). If any vertices of the SP are below the plane of the RP ($z < 0$), a false shadow is cast as in Figure 40. The "submerged" portion of the SP must be clipped off before projection.

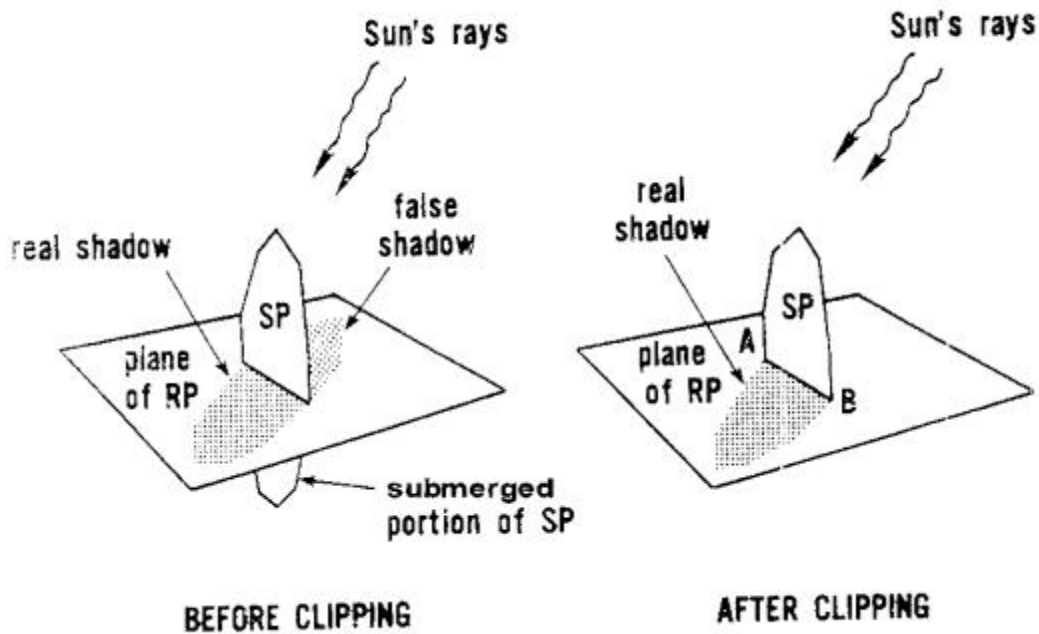


Figure 40. Illustration of Shadow Clipping

This is done by finding, through linear interpolation, the points on the perimeter of the SP, which intersect the plane of the RP. These points become new vertices of the SP, which together with the other positive vertices define a clipped SP that casts only a real shadow.

A vertex located at (x, y, z) relative to the RP coordinate system casts a shadow to a point in the plane of the RP given by

$$x' = x - \frac{z \cdot a}{\cos \theta} \quad (152)$$

$$y' = y - \frac{z \cdot b}{\cos \theta} \quad (153)$$

where

$$a = \sin \psi \cdot CS_1 - \cos \psi \cdot CS_2$$

and

$$b = -\cos \psi \cdot \cos \phi \cdot CS_1 - \sin \psi \cdot \cos \phi \cdot CS_2 + \sin \phi \cdot CS_3$$

Homogeneous Coordinates

Two-dimensional homogeneous coordinate techniques are used to determine the vertices of shadow overlaps. In homogeneous coordinates, points and lines are represented by a single form that allows simple vector operations between those forms [Newman-Sproul]. A point (X, Y) is represented by a three element vector (x, y, w) where $x = w \cdot X$, $y = w \cdot Y$, and w is any real number except zero. A line is also represented by a three element vector (a, b, c) . The directed line (a, b, c) from point (x_1, y_1, w_1) to point (x_2, y_2, w_2) is given by:

$$(a, b, c) = (x_1, y_1, z_1) \otimes (x_2, y_2, z_2) \quad (154)$$

The sequence in the cross product is a convention to determine sign. The condition that a point (x, y, w) lie on a line (a, b, c) is that

$$(a, b, c) \bullet (x, y, w) = 0 \quad (155)$$

The point is normalized by dividing by w . Then if

$$(a, b, c) \bullet (x/w, y/w, 1) > 0 \quad (156)$$

the point is to the left of the line. If it is less than zero, the point is to the right of the line. The intercept (x, y, w) of line (a_1, b_1, c_1) and line (a_2, b_2, c_2) is given by:

$$(x, y, w) = (a_1, b_1, c_1) \otimes (a_2, b_2, c_2) \quad (157)$$

Note that the use of homogeneous coordinates as outlined above provides a consistent method and notation for defining points and lines, for determining intercepts, and for determining whether a point lies to the left, to the right, or on a line. Normalization provides the means for transforming to and from homogeneous notation and Cartesian coordinates. Thus, if (X, Y) is a Cartesian coordinate pair, its homogeneous coordinates are $(X, Y, 1)$. Similarly, the homogeneous coordinates (x, y, w) can be transformed to the Cartesian point with coordinates $(x/w, y/w)$.

Overlapping Shadows

After transforming the shadows onto the plane of the receiving surface, the basic job of the shadow algorithm is to determine the area of the overlap between the polygons representing the shadows and the polygon representing the receiving surface.

There is considerable simplification if only convex (no interior angle $> 180^\circ$) polygons are considered. The overlap between two convex polygons is another convex polygon. Coordinate and projection transformations of a convex polygon produce another convex polygon. Any non-convex polygon can be constructed as a sum of convex ones.

The vertices that define the overlap between two convex polygons, A and B, consist of:

- the vertices of A enclosed by B
- the vertices of B enclosed by A
- and the intercepts of the sides of A with the sides of B

In Figure 41, point a is the result of rule 1, point c is the result of rule 2, and points b and d result from rule 3. The overlap of A and B is the polygon a-b-c-d. Figure 42 shows an overlap where all of the vertices of B are enclosed by A. Figure 43 shows an overlap defined only by the intercepts of A and B. Figure 44 shows a more complex overlap.

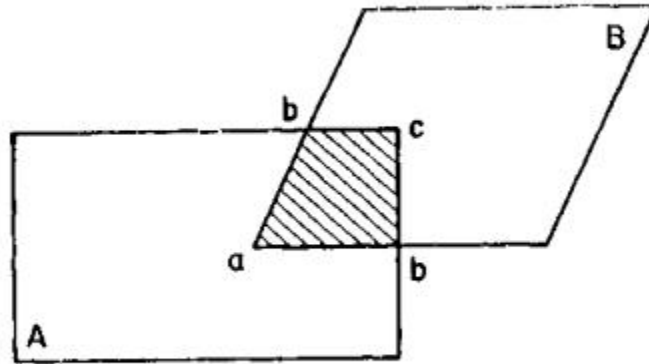


Figure 41. Point a – Vertex of A Enclosed by B

Coordinate transformation retains the order of the vertices of a polygon, while a projection reverses the order. The sequence of vertices of the receiving polygons should be reversed so it and all shadow polygons will have the same sequence.

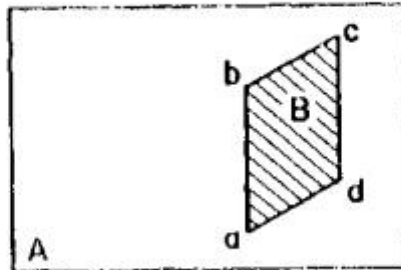


Figure 42. Surface A Totally Overlaps Surface B.

A point is enclosed by a clockwise, convex polygon if the point lies to the right of all sides (or does not lie to the left of any side) of the polygon. The intercept of two sides may not lie beyond the ends of either side. These are "line segments" rather than "lines". It is possible to tell if line segments A and B intercept within their end points by noting that the ends of A must lie on both sides of B, and the ends of B must lie on both sides of A. This should be done before the intercept is calculated.

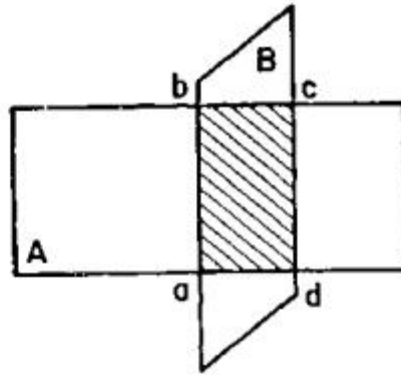


Figure 43. Figure Formed from Intercept Overlaps Between A and B

Once the vertices are determined, they must be sorted into clockwise order for the area to be computed. Given a closed, planar polygon of n sequential vertices $(x_1, y_1), (x_2, Y_2) \dots, (x_n, Y_n)$, its **area** is given:

$$Area = \frac{1}{2} \sum_{i=1}^n (x_i y_{i+1} - x_{i+1} y_i) \quad (158)$$

where $(x_{n+1}, y_{n+1}) = (x_1, y_1)$

The area is positive if the vertices are counter-clockwise and negative if they are clockwise.

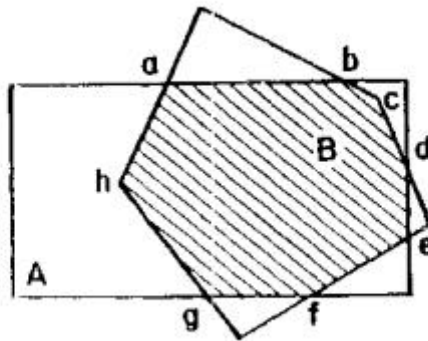


Figure 44. Complex Overlapping Condition

If two shadows overlap the receiving surface, they may also overlap each other as in Figure 45. The vertices of this overlap can be computed. The areas of all overlaps can be computed. The total sunlit area can be expressed as the sum of all polygon areas given a proper sign on each of the areas.

The following convention was adopted:

Table 20. Surface / Area Characteristic / Convention

Surface Characteristic	Area Convention
receiving surface	positive (A)
overlap between shadow and receiving	negative (B & C)
overlap between two shadows	positive (D)

and so on through multiple overlaps where the sign of the overlap area is the product of the signs of the overlapping areas.

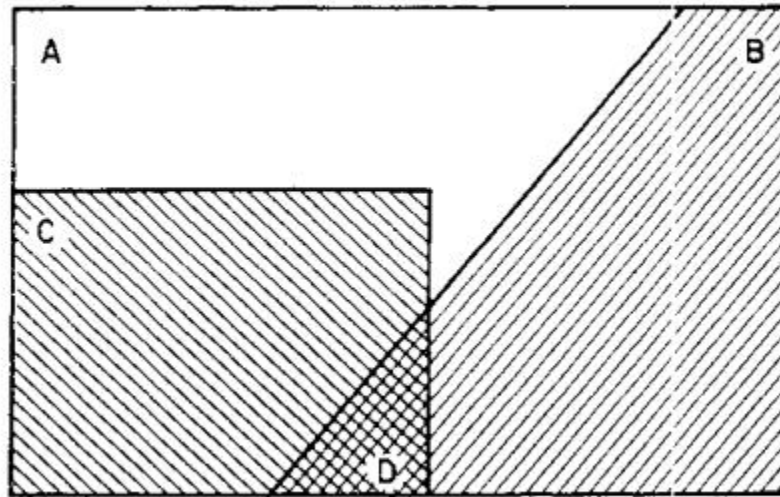


Figure 45. Multiple Shadow Overlaps

Partially transparent shadowing surfaces can also be modeled by giving a transparency (τ) to every shadowing polygon. Let τ of the receiving polygon be one. Then the τ of every overlap of polygons i and j is the product of τ_i and τ_j . The shaded area is then computed by summing $A_i \cdot (1 - \tau_i)$ for all overlap polygons.

It is easy to determine the sunlit area of a window once all the shadow and overlap vertices on the wall have been computed. Consider wall 2 of Figure 36. First, the wall is considered a simple rectangle and the window on it is ignored. The shadow overlapping is performed and the sunlit portion of the gross wall area is computed. Then the window rectangle is overlapped with the shadow to determine its sunlit area. The sunlit area of the window is subtracted from the gross wall sunlit area to determine the net wall sunlit area. During this calculation it is not necessary to recompute the shadows, because they were precisely determined on the wall.

Solar Gains

The total solar gain on any exterior surface is a combination of the absorption of direct and diffuse solar radiation given by

$$Q_{so} = \alpha \cdot \left(I_b \cdot \cos \theta \cdot \frac{S_s}{S} + I_s \cdot F_{ss} + I_g \cdot F_{sg} \right) \quad (159)$$

where

α =solar absorptance of the surface

θ =angle of incidence of the sun's rays

S =area of the surface

S_s = sunlit area

I_b =intensity of beam (direct) radiation

I_s =intensity of sky diffuse radiation

I_g =intensity of ground reflected diffuse radiation

F_{ss} = angle factor between the surface and the sky

F_{sg} = angle factor between the surface and the ground

For the surface of a building located on a featureless plain

$$F_{ss} = \frac{1 + \cos \phi}{2} \quad (160)$$

and

$$F_{sg} = \frac{1 - \cos \phi}{2} \quad (161)$$

If the surface is shaded the program modifies F_{ss} by a correction factor that takes into account the radiance distribution of the sky (see “Shadowing of Sky Diffuse Solar Radiation”). Shading of ground diffuse solar radiation is not calculated by the program. It is up to the user to estimate the effect of this shading and modify the input value of F_{sg} accordingly.

Solar Distribution

As discussed in the Input Output Reference (Object: Building), the field Solar Distribution, in the “Building” input object, determines how EnergyPlus will treat beam solar radiation entering a zone through exterior windows. There are five choices: **MinimalShadowing**, **FullExterior**, **FullInteriorAndExterior**, **FullExteriorWithReflections**, and **FullInteriorAndExteriorWithReflections**.

MinimalShadowing

In this case, there is no exterior shadowing except from window and door reveals. All beam solar radiation entering the zone is assumed to fall on the floor, where it is absorbed according to the floor's solar absorptance. Any reflected by the floor is added to the transmitted diffuse radiation, which is assumed to be uniformly distributed on all interior surfaces. If no floor is present in the zone, the incident beam solar radiation is absorbed on all interior surfaces according to their absorptances. The zone heat balance is then applied at each surface and on the zone's air with the absorbed radiation being treated as a flux on the surface.

FullExterior

In this case, shadow patterns on exterior surfaces caused by detached shading, wings, overhangs, and exterior surfaces of all zones are computed. As for MinimalShadowing, shadowing by window and door reveals is also calculated. Beam solar radiation entering the zone is treated as for MinimalShadowing.

FullExteriorWith Reflections

This case is the same interior distribution as the preceding option but uses exterior reflections as well (see Solar Radiation Reflected from Exterior Surfaces for further explanation).

FullInteriorAndExterior

This is the same as FullExterior except that instead of assuming all transmitted beam solar falls on the floor the program calculates the amount of beam radiation falling on each surface in the zone, including floor, walls and windows, by projecting the sun's rays through the exterior windows, taking into account the effect of exterior shadowing surfaces and window shading devices.

If this option is used, you should be sure that the surfaces of the zone totally enclose a space. This can be determined by viewing the **eplusout.dxf** file with a program like AutoDesk's Volo View Express. You should also be sure that the zone is **convex**. Examples of convex and non-convex zones are shown in Figure 46. The most common non-convex zone is an L-shaped zone. (A formal definition of convex is that any straight line passing through the zone intercepts at most two surfaces.) If the zone's surfaces do not enclose a space or if the zone is not convex you should use Solar Distribution = **FullExterior** instead of **FullInteriorAndExterior**.

If you use **FullInteriorAndExterior** the program will calculate how much beam radiation falling on an interior window is absorbed by the window, how much is reflected back into the zone, and how much is transmitted into the adjacent zone. (Interior windows are assumed to have no shading device).

If you use **FullInteriorAndExterior** the program will also calculate how much beam radiation falling on the inside of an exterior window (from other windows in the zone) is absorbed by the window, how much is reflected back into the zone, and how much is transmitted to the outside. In this calculation the effect of an interior or exterior shading device, if present, is accounted for.

FullInteriorAndExteriorWith Reflections

This case is the same interior distribution as the preceding option but uses exterior reflections as well (see Solar Radiation Reflected from Exterior Surfaces for further explanation).

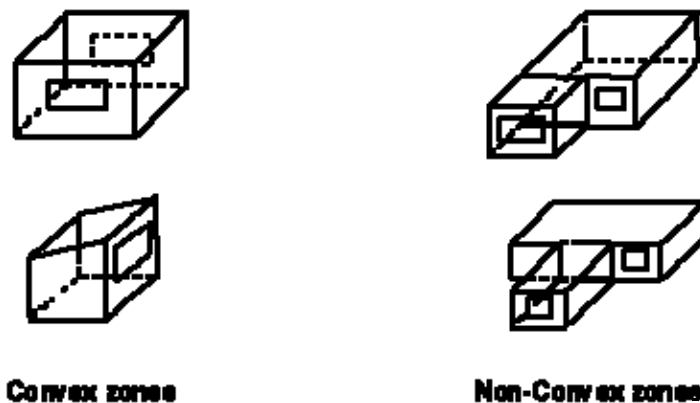


Figure 46. Illustration of Convex and Non-convex Zones

Details of the Interior Solar Distribution Calculation

EnergyPlus calculates the distribution of short-wave radiation in the interior of each thermal zone. This radiation consists of beam solar radiation, diffuse solar radiation, and short-wave radiation from electric lights. The program determines the amount of this radiation that is (1) absorbed on the inside face of opaque surfaces, (2) absorbed in the glass and shading

device layers of the zone's exterior and interior windows, (3) transmitted through the zone's interior windows to adjacent zones, and (4) transmitted back out of the exterior windows. The effects of movable shading devices on the exterior windows are taken into account.² Most of this calculation is done in subroutine CalcInteriorSolarDistribution in the SolarShading module.

Initial Distribution of Diffuse Solar Transmitted through Exterior and Interior Windows

As of Version 2.1 the treatment of diffuse solar transmitted first through exterior windows and subsequently through interior windows has been improved. Diffuse solar (from sky and ground sources) transmitted through exterior windows is first distributed to the interior heat transfer surfaces in the zone containing the exterior windows. This initial distribution apportions the transmitted diffuse solar to interior surfaces using the approximate view factors described above in "LW Radiation Exchange Among Zone Surfaces." The amount of this initially distributed diffuse solar absorbed by each interior surface, and each window material layer, is calculated and later added to the "short-wave radiation absorbed" values described below. The amount of this initially distributed diffuse solar that is reflected is accumulated for each zone and redistributed uniformly as part of the QD calculation described below. The amount of this initially distributed diffuse solar that is transmitted by interior windows to adjacent zones is initially distributed to the interior heat transfer surfaces in the adjacent zone in the same manner as just described.

This new treatment of diffuse solar is intended to more accurately account for the initial absorption, transmittance, and reflection of short-wave radiation prior to the uniform distribution described below.

Interior Solar Radiation Absorbed by Opaque Surfaces

The short-wave radiation absorbed on the inside face of an opaque surface (floor, wall or ceiling) is given by

$$Q_{RadSWInAbs}(SurfNum) = QS(ZoneNum) * AbsIntSurf(SurfNum) + AISurf(SurfNum) * BeamSolarRad \quad [W/m^2] \quad (162)$$

where

$SurfNum$ = surface number

$ZoneNum$ = number of zone that surface belongs to

$QS(ZoneNum)$ = short-wave diffuse irradiance in the zone $[W/m^2]$

$AbsIntSurf(SurfNum)$ = inside solar absorptance of the surface

$AISurf(SurfNum)$ = inside beam solar irradiance factor for the surface [-]

$BeamSolarRad$ = outside beam normal solar irradiance $[W/m^2]$

Interior Diffuse Radiation

QS is assumed to be uniformly distributed throughout the zone. It is calculated as follows. Let Q_{sw} be the total diffuse short-wave radiation entering the zone or originating in the zone. Since Q_{sw} is ultimately absorbed or transmitted by zone heat transfer surfaces, summing over these surfaces gives the following energy balance equation:

$$\sum_{i=1}^{N_{surf}} QS(ZoneNum) * \alpha_i A_i = Q_{sw}(ZoneNum)$$

where

² The program does not allow shading devices on interior windows.

i = zone surface number counter

N_{surf} = number of heat transfer surfaces in zone

A_i = surface area [m^2]

α_i = inside solar absorptance for an opaque surface, or, for a window, = back diffuse transmittance plus back diffuse system absorptance of glass layers and shading device layers (if present)

Solving this equation for QS gives:

$$QS(ZoneNum) = \frac{Q_{SW}(ZoneNum)}{\sum_{i=1}^{N_{surf}} AbsInsSurf_i A_i} = Q_{SW}(ZoneNum) * VMULT(ZoneNum) \quad (163)$$

where

$$VMULT(ZoneNum) = \frac{1}{\sum_{i=1}^{N_{surf}} AbsIntSurf_i * A_i} \quad [m^{-2}]$$

Q_{sw} is given by

$$Q_{SW} = QD(ZoneNum) + \\ ZoneIntGain(ZoneNum)\%QLTSW + \\ ZoneIntGain(ZoneNum)\%T_QLTSW \quad [W]$$

where

$ZoneIntGain(ZoneNum)\%QLTSW$ = short-wave radiation into zone from general (overhead) electric lighting [W]

$ZoneIntGain(ZoneNum)\%T_QLTSW$ = short-wave radiation into zone from task electric lighting [W]

$QD(ZoneNum)$ = diffuse solar radiation entering or originating in zone [W]

$QD(ZoneNum)$ is given by:

$$QD(ZoneNum) = DBZone(ZoneNum) * BeamSolarRad + InitialDifSolDistReflectedW(ZoneNum)$$

where

$BeamSolarRad$ is the outside beam normal solar irradiance [W/m^2]

$DBZone(ZoneNum)$ is the diffuse solar radiation originating from beam solar that passes through the exterior windows in the zone and reflects diffusely from inside zone surfaces plus beam solar entering the zone as diffuse radiation from windows with shading devices or diffusing glass (all divided by $BeamSolarRad$) [m^2]

$InitialDifSolDistReflectedW(ZoneNum)$ is the diffuse solar radiation originating from sky and sun related diffuse solar transmitted through the zone's exterior or interior windows into the zone, and reflected diffusely from inside zone surfaces.

$DBZone(ZoneNum)$ is calculated as:

$$DBZone(ZoneNum) = BTOTZone - BABSZone \quad [m^2]$$

where

$BTOTZone$ = total beam solar incident on the zone's exterior windows that is transmitted as beam or diffuse.³

$BABSZone$ = total beam solar absorbed inside the zone.

$BTOTZone$ is given by:

$$BTOTZone = \sum_{i=1}^{N_{extwin}} TBmAll_i * SunlitFract_i * CosInc_i * Area_i * InOutprojSLFracMult_i$$

+ Diffuse entering zone from beam reflected by window inside reveal surfaces⁴

+ Diffuse transmitted by windows from beam reflected by outside reveal surfaces

– Beam absorbed by window inside reveal surfaces

Here,

$TBmAll$ = beam-to-beam plus beam-to-diffuse transmittance of window

$SunlitFract$ = fraction of window irradiated by sun

$CosInc$ = cosine of solar incidence angle on window

$Area$ = glazed area of window [m^2]

$InOutProjSLFracMult$ = shadowing factor due to inside and outside projections of window frame and/or divider (= 1.0 if there is no frame or divider).

$BABSZone$ is given by the following sum (see Figure 47):

$$BABSZone = \text{Beam absorbed by opaque inside surfaces}^5$$

- + Beam transmitted through the zone's interior windows +
- + Beam transmitted back out of the zone's exterior windows +
- + Beam absorbed by the zone's exterior and interior windows +
- + Beam absorbed by inside daylighting shelves

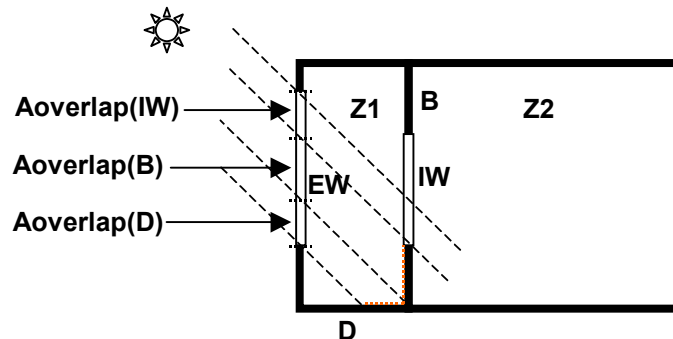


Figure 47. Vertical section through a two-zone building showing where transmitted beam solar falls. Some of the beam solar from exterior window EW is absorbed by the floor, D, interior wall, B, and interior window, IW. Some is transmitted by IW to the adjacent zone, Z2. Aoverlap is the irradiated area of a

³ For beam incident on an exterior window we have the following: For transparent glass with no shade or blind there is only beam-to-beam transmission. For diffusing glass, or if a window shade is in place, there is only beam-to-diffuse transmission. If a window blind is in place there is beam-to-diffuse transmission, and, depending on slat angle, solar profile angle, etc., there can also be beam-to-beam transmission.

⁴ See "Beam Solar Reflection from Window Reveal Surfaces."

⁵ If Solar Distribution = FullInteriorAndExterior in the Building object, the program calculates where beam solar from exterior windows falls inside the zone. Otherwise, all beam solar is assumed to fall on the floor.

surface projected back onto the plane of EW. Beam reflected by D, B and IW contributes to the interior short-wave radiation flux in Z1.

If zone $ZoneNum$ shares interior windows with other zones, $QS(ZoneNum)$ is modified to take into account short-wave radiation received from the other zones through these windows:

$$QS(ZoneNum) \rightarrow QS(ZoneNum) + \sum_{\substack{\text{other} \\ \text{zones}}} FractDifShortZtoZ(OtherZoneNum, ZoneNum) * [QD(OtherZoneNum) + ZoneIntGain(OtherZoneNum)\%QLTSW + ZoneIntGain(OtherZoneNum)\%T_QLTSW]$$

where

$FractDifShortZtoZ(OtherZoneNum, ZoneNum)$ = “diffuse solar exchange factor” = fraction of short-wave radiation in $OtherZoneNum$ that is transmitted to $ZoneNum$. This factor is calculated in subroutine `ComputeDifSolExcZonesWIZWindows` taking into account multiple reflection between zones. For example, for two zones means that some of the radiation transmitted from Zone1 to Zone2 is reflected back to Zone1, and some of this is in turn reflected back to Zone2, etc.

Interior Beam Radiation

The inside beam solar irradiance factor⁶ in (162) is given by:

$$AISurf(SurfNum) = \frac{AbsIntSurf(SurfNum)}{A(SurfNum)} \sum_{i=1}^{N_{extwin}} TBm_i * Aoverlap_i(SurfNum) * CosInc_i$$

where

i = exterior window number

N_{extwin} = number of exterior windows in zone

$CosInc_i$ = cosine of angle of incidence of beam on exterior window i

TBm_i = beam-to-beam transmittance of exterior window i at $CosInc_i$ ⁷

$Aoverlap_i(SurfNum)$ = beam solar irradiated area of surface $SurfNum$ projected back onto the plane of exterior window i (the $Aoverlap$'s for an exterior window sum up to the glazed area of the window). These overlap areas (Figure 47) are determined with the EnergyPlus shadowing routines by considering a zone's exterior window as a “sending” surface and the inside faces of the zone's other surfaces as “receiving” surfaces (see “Shading Module”). The overlap areas for a particular exterior window depend on the sun position, the geometry of the window, the geometry of the interior surfaces, and the location of the window with respect to the interior surfaces.

$AbsIntSurf(SurfNum)$ = inside face solar absorptance of surface $SurfNum$

$A(SurfNum)$ = area of surface $SurfNum$ [m²]

⁶ For the purposes of the surface heat balance calculation, any beam solar radiation absorbed by a surface is assumed to be uniformly distributed over the surface even though in reality it is likely to be concentrated in one or more discrete patches on the surface.

⁷ TBm_i is zero if the window has diffusing glass or a shade. TBm_i can be > 0 if a blind is present and the slat angle, solar profile angle, etc., are such that some beam passes between the slats.

Interior Solar Radiation Absorbed by Windows

The interior short-wave radiation absorbed by layer l (glass, shade or blind) of a window is equal to:

$$QS(ZoneNum) * \alpha_{l,back}^{dif}(SurfNum) + \\ BeamSolarRad * \frac{\alpha_{l,back}^{beam}(SurfNum)}{A(SurfNum)} \sum_{i=1}^{N_{extwin}} TBM_i * Aoverlap_i(SurfNum) * CosInc_i \quad [W/m^2]$$

where

$\alpha_{l,back}^{dif}$ = the system diffuse solar absorptance of layer l for irradiance from the back side

$\alpha_{l,back}^{beam}$ = the system beam solar absorptance of layer l for irradiance from the back side

$A(SurfNum)$ = glazing area [m^2]

Note that as of Version 2.1, the initially distributed diffuse solar absorbed by each surface (as described above under "Initial Distribution of Diffuse Solar Transmitted through Exterior and Interior Windows") is added to this uniformly distributed short-wave radiation.

Interior Solar Radiation Transmitted by Interior Windows

Interior Diffuse Radiation Transmitted by Interior Windows

The interior diffuse short-wave radiation transmitted by an interior window to the adjacent zone is given by

$$QS(ZoneNum) * \tau^{dif}(SurfNum) * A(SurfNum) \quad [W]$$

where

$\tau^{dif}(SurfNum)$ = diffuse transmittance of the interior window

Interior Beam Radiation Transmitted by Interior Windows

The interior beam solar radiation transmitted by an interior window to the adjacent zone is

$$BeamSolarRad * \tau^{beam}(SurfNum) \sum_{i=1}^{N_{extwin}} TBM_i * Aoverlap_i(SurfNum) * CosInc_i \quad [W]$$

where $\tau^{beam}(SurfNum)$ is the beam-to-beam transmittance of the interior window at the angle of incidence of beam solar from the exterior window on the interior window. The program does not track where this radiation falls in the adjacent zone: it is counted as diffuse radiation in that zone. Therefore,

$$QS(ZoneNum) \rightarrow QS(ZoneNum) + [beam\ solar\ from\ adjacent\ zones] * VMULT(ZoneNum)$$

Ground Reflectances

Ground reflectance values (Ref Object: Site:GroundReflectance) are used to calculate the ground reflected solar amount. This fractional amount (entered monthly) is used in the following equation:

$$\text{GroundReflectedSolar} = (\text{BeamSolar} \cdot \text{COS}(\text{SunZenithAngle}) + \text{DiffuseSolar}) \cdot \text{GroundReflectance}$$

Of course, the Ground Reflected Solar is never allowed to be negative. The Snow Ground Reflectance Modifier can further modify the ground reflectance when snow is on the ground. If the user enters 0.0 for each month, no ground reflected solar is used.

Ground Reflectances (Snow)

When snow is on the ground, ground reflectances may change. (Ref Object: Site:GroundReflectance:SnowModifier). This object allows the user to specify two values, Ground Reflected Solar Modifier and Daylighting Ground Reflected Solar Modifier.

Ground Reflected Solar Modifier is used to modified the basic monthly ground reflectance when snow is on the ground (from design day input or weather data values). Values can range from 0.0 to 1.0.

$$\text{GroundReflectance}_{\text{used}} = \text{GroundReflectance} \cdot \text{SolarModifier}_{\text{Snow}}$$

Daylighting Ground Reflected Solar Modifier is used to modified the basic monthly ground reflectance when snow is on the ground (from design day input or weather data values). Values can range from 0.0 to 1.0.

$$\text{DaylightingGroundReflectance}_{\text{used}} = \text{GroundReflectance} \cdot \text{DaylightingModifier}_{\text{Snow}}$$

References

- ASHRAE. 2005. Handbook of Fundamentals, Chapter 31, Atlanta: ASHRAE.
- ASHRAE. 2007. HVAC Applications, Chapter 33, Atlanta, ASHRAE.
- Zhang, Qingyuan, Joe Huang, and Siwei Lang. 2002. "Development of Typical Year Weather Data for Chinese Locations", American Society of Heating Refrigeration and Air-Conditioning Engineers, ASHRAE Transactions, Vol 108, Part 2.
- Threlkeld, J.L. and R.C. Jordan. 1958. Direct solar radiation available on clear days. ASHRAE Transactions 64:45.
- Groth, C. C., and Lokmanhekim, M. 1969. "Shadow - A New Technique for the Calculation of Shadow Shapes and Areas by Digital Computer," Second Hawaii International Conference on System Sciences, Honolulu, HI, January 22-24, 1969.
- Walton, G.N. 1983. "The Thermal Analysis Research Program Reference Manual Program (TARP)", National Bureau of Standards (now National Institute of Standards and Technology).
- Walton, G. N. 1978. "The Application of Homogeneous Coordinates to Shadowing Calculations", American Society of Heating Refrigeration and Air-Conditioning Engineers, ASHRAE Transactions, Vol 84, Part I.
- Meeus, Jean. 2000. Astronomical Algorithms, Willmann-Bell.
- Newman, M. W., and Sproul, R. F. 1973. Principles of Interactive Computer Graphics, McGraw-Hill.
- Polygon area derived from Green's Theorem. Graphic Gems repository.

Solar Radiation Reflected from Exterior Surfaces

EnergyPlus has an option to calculate beam and sky solar radiation that is reflected from exterior surfaces and then strikes the building. This calculation occurs if “withReflections” is used on the SolarDistribution option in the Building object. For zones with detailed daylighting, these reflections are also considered in the daylight illuminance calculations.⁸

The reflecting surfaces fall into three categories:

- **Shadowing surfaces.** These are surfaces like overhangs or neighboring buildings entered with objects Shading:Site:Detailed, Shading:Building:Detailed, or Shading:Zone:Detailed. Examples are shown in Figure 48.

These surfaces can have diffuse and/or specular (beam-to-beam) reflectance values that are specified with the ShadingProperty:Reflectance object.

- **Exterior building surfaces.** In this case one section of the building reflects solar radiation onto another section (and vice-versa). See Figure 49.

Opaque building surfaces (walls, for example) are assumed to be diffusely reflecting. Windows and glass doors are assumed to be specularly reflecting. The reflectance values for opaque surfaces are calculated by the program from the Solar Absorptance and Visible Absorptance values of the outer material layer of the surface’s construction. The reflectance values for windows and glass doors are calculated by the program from the reflectance properties of the individual glass layers that make up surface’s construction assuming no shading device is present and taking into account inter-reflections among the layers.

- **The ground surface.** Beam solar and sky solar reflection from the ground is calculated even if “withReflections” is not used (the default). But in this case the ground plane is considered unobstructed, i.e., the shadowing of the ground by the building itself or by obstructions such as neighboring buildings is ignored. This shadowing is taken into account only if “WithReflections” is used in the Solar Distribution field (in “Building” input object) (Figure 50). In this case the user-input value of ground view factor is not used.

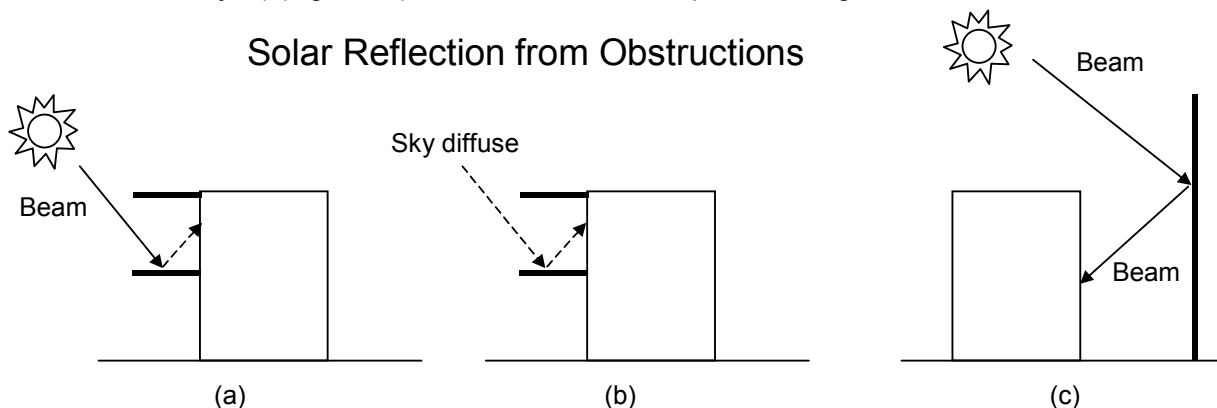


Figure 48. Examples of solar reflection from shadowing surfaces in the Shading series of input objects. Solid arrows are beam solar radiation; dashed arrows are diffuse solar radiation. (a) Diffuse reflection of beam solar radiation from the top of an overhang. (b) Diffuse reflection of sky solar radiation from the top of an overhang. (c) Beam-to-beam (specular) reflection from the façade of an adjacent highly-glazed building represented by a vertical shadowing surface.

⁸ A different method from that described here is used for calculating reflections from daylighting shelves (see “Daylighting Shelves”).

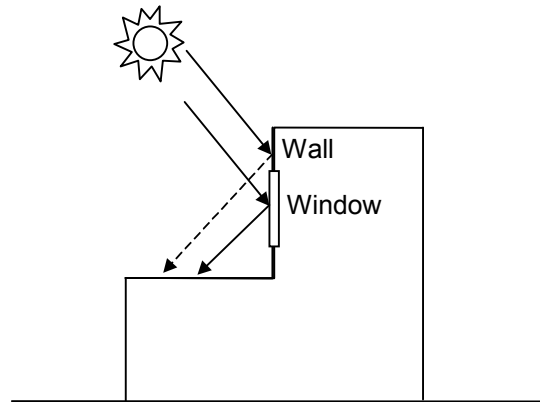


Figure 49. Solar reflection from building surfaces onto other building surfaces. In this example beam solar reflects from a vertical section of the building onto a roof section. The reflection from the window is specular. The reflection from the wall is diffuse.

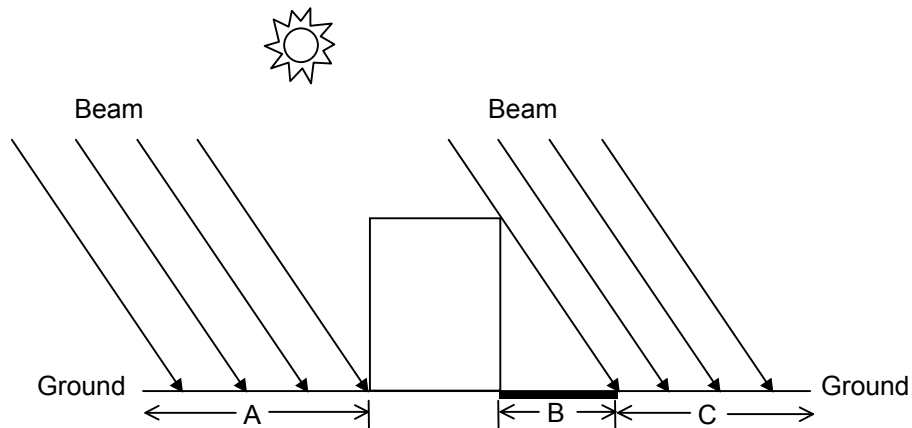


Figure 50. Shadowing by the building itself affects beam solar reflection from the ground. Beam-to-diffuse reflection from the ground onto the building occurs only for sunlit areas, A and C, not for shaded area, B. Shadowing by the building also affects sky solar reflection from ground (not shown).

Diffuse Reflection of Beam Solar and Sky Solar Radiation

A ray-tracing method is used to calculate beam solar and sky solar radiation that is diffusely reflected onto each of a building's exterior surfaces (walls, roofs, windows and doors), called here "receiving surfaces." The calculation begins by generating a set of rays proceeding into the outward hemisphere at each receiving point on a receiving surface. Then it determines whether each ray hits the sky, ground or an obstruction. The radiance at the hit point from reflection of incident beam or sky solar is determined and the contribution of this radiance to the receiving surface is calculated, added to the contribution from other hit points, and averaged over the receiving points. Separate calculations are done for beam-to-diffuse and sky solar reflection from all obstructions and beam-to-diffuse and sky solar reflection from the ground. (For beam-to-beam reflection see "Beam Solar Radiation Specularly Reflected from Obstructions," below.)

Receiving points

An N -sided surface is assigned N receiving points with the following coordinates, expressed in terms of the surface vertex coordinates:

$$P_{ij} = \sum_{k=1}^N a_{ki} v_{kj}, \quad i = 1, 3; \quad j = 1, 3$$

where

$P_{ij} = j^{\text{th}}$ coordinate of the i^{th} receiving point

$v_{kj} = j^{\text{th}}$ coordinate of the k^{th} surface vertex

If $N = 3$: $a_{kj} = 3/5$ if $k = i$; $= 1/5$ otherwise

If $N > 3$: $a_{kj} = \frac{N+1}{2N}$ if $k = i$; $= \frac{1}{2N}$ otherwise

For example, for a vertical 3m by 5m rectangle with vertices (0,0,3), (0,0,0), (5,0,0) and (5,0,3), this expression gives receiving points at (1.25,0,2.25), (1.25,0,0.75), (3.75,0,0.75) and (3.75,0,2.25), as shown in Figure 51.

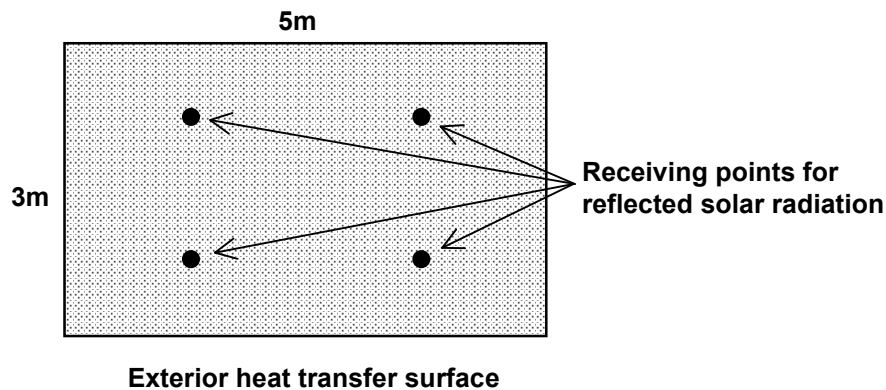


Figure 51. Vertical rectangular exterior heat transfer surface showing location of receiving points for calculating incident solar radiation reflected from obstructions.

Rays

A total of 90 rays are sent out into the exterior hemisphere surrounding each receiving point. An upgoing ray may hit an obstruction or the sky. A downgoing ray may hit an obstruction or the ground. See Figure 52.

In subroutine InitSolReflRecSurf, the following is determined for each ray, i :

1. Unit vector in direction of ray
2. Cosine of angle between ray and plane of receiving surface ($\cos \alpha_i$)
3. Element of solid angle associated with ray ($d\Omega_i$)
4. If the ray hits one or more obstructions, the coordinates of the hit point on the obstruction nearest the receiving point

5. For the surface containing the hit point: the surface number, the solar reflectance ($\rho_{obs,i}$ if an obstruction), and the surface unit vector at the hit point pointing into the hemisphere containing the receiving point
6. If the ray is downgoing and hits the ground, the coordinates of the ground hit point
7. Distance from receiving point to hit point

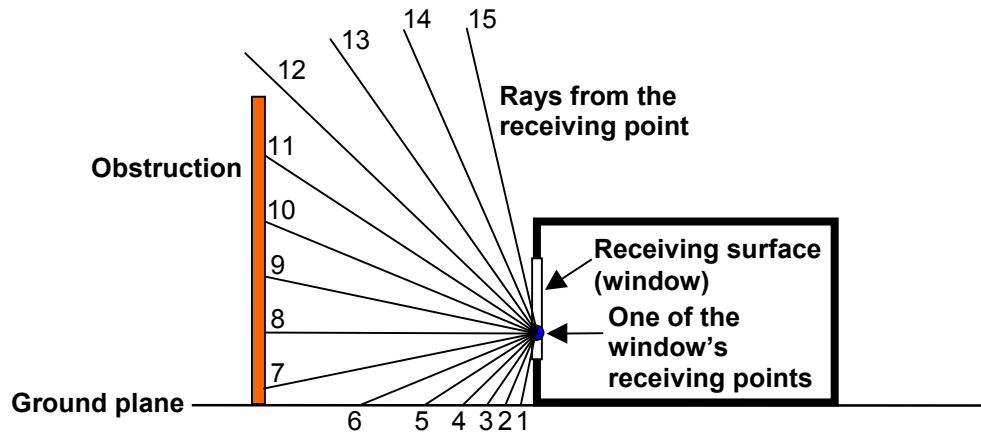


Figure 52. Two-dimensional schematic showing rays going outward from a point on a receiving surface. Rays 1-6 hit the ground, rays 7-11 hit an obstruction, and rays 12-15 hit the sky.

Sky Solar Radiation Diffusely Reflected from Obstructions

The factor for reflection of sky radiation from obstructions onto a receiving surface is calculated in subroutine CalcSkySolDiffuseReffFactors. It is given by:

$$\text{ReflFacSkySolObs}(\text{RecSurfNum}) = \frac{1}{N_{rec}} \sum_{i=1}^{N_{rec}} \sum_{j=1}^{N_{ray}} \text{Hit}_{obs,i} \text{ViewFacSky}_{obs,i} \text{DifShdgRatioIsoSky}_{obs,i} \rho_{obs,i} \cos \alpha_i / \pi$$

where

RecSurfNum is the receiving surface number,

N_{rec} is the number of receiving points,

N_{ray} is the number of rays,

“ obs,i ” denotes the obstruction hit by ray i ,

$\text{Hit}_{obs,i} = 1$ if ray i hits an obstruction, = 0 otherwise,

$\text{ViewFacSky}_{obs,i} = \text{unobstructed sky view factor of the obstruction} = (1 + \cos \text{tilt}_{obs}) / 2$,

$\text{DifShdgRatioIsoSky}_{obs,i} = (\text{obstructed sky irradiance on obstruction}) / (\text{unobstructed sky irradiance on obstruction})$

In this equation the product $\text{ViewFacSky} * \text{DifShdgRatioIsoSky}$ is the sky irradiance at the hit point divided by the horizontal sky irradiance taking into account shadowing of sky diffuse radiation on the obstruction by other obstructions, and assuming that the radiance of the sky is uniform. Note that we ignore secondary reflections here and in the following sections. In

the present case this means that the irradiance at the hit point due to reflection of sky radiation from the ground or from other obstructions is not considered.

The above reflection factor is used in the timestep calculation to find the irradiance on a receiving surface due to sky radiation reflected from obstructions:

$$Q_{RadSWOutIncSkyDiffReflObs}(RecSurfNum) = DifSolarRad * ReflFacSkySolObs(RecSurfNum) \quad (W/m^2)$$

where *DifSolarRad* is the horizontal sky irradiance on an unobstructed horizontal plane (W/m^2).

Sky Solar Radiation Diffusely Reflected from the Ground

If a downgoing ray from a receiving point hits the ground (for example, rays 1-6 in Figure 52), the program calculates the radiance at the ground hit point due to sky diffuse solar reaching that point. To do this, rays are sent upward from the ground hit point and it is determined which of these rays are unobstructed and so go to the sky (for example, rays 6-10 in Figure 53). For this calculation it is assumed that the radiance of the sky is uniform.

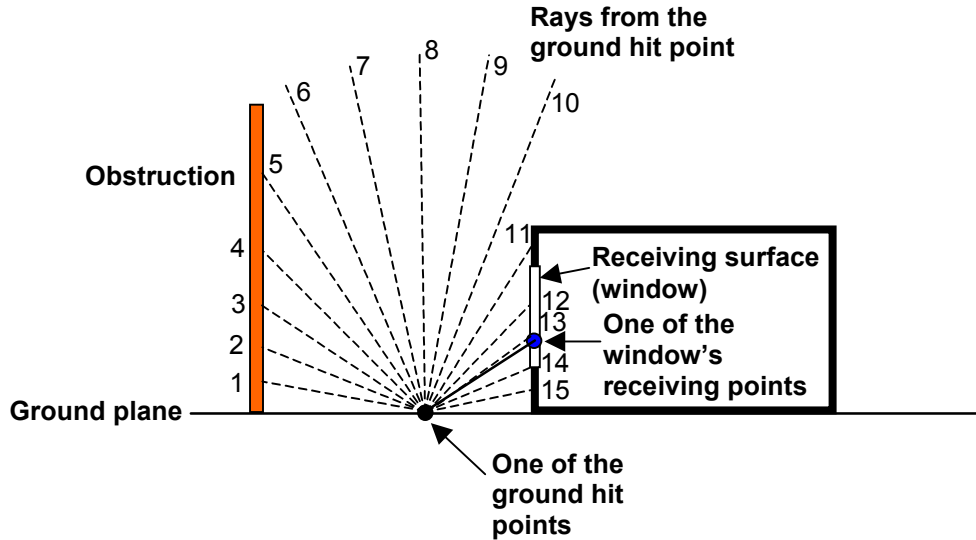


Figure 53. Two-dimensional schematic showing rays going upward from a ground hit point.

The factor for reflection of sky radiation from the ground onto a receiving surface is calculated in subroutine CalcSkySolDiffuseReflFactors. It is given by:

$$ReflFacSkySolGnd(RecSurfNum) = \frac{1}{N_{rec}} \sum_{i=1}^{N_{rec}} \sum_{j(i)=1}^{N_{ray}} \left[(Hit_{gnd,i} d\Omega_i \cos \alpha_i / \pi) \sum_{j(i)} Hit_{sky,j(i)} \cos \alpha_{j(i)} d\Omega_{j(i)} / \pi \right]$$

where

j(i) denotes an upgoing ray from the ground point hit by ray *i* from the receiving point,

$Hit_{sky,j(i)} = 1$ if ray $j(i)$ hits the sky, = 0 otherwise,

$\alpha_{j(i)}$ is the angle of incidence of ray $j(i)$ with respect to the ground plane,

$d\Omega_{j(i)}$ is the solid angle element associated with ray $j(i)$.

This factor is used in the timestep calculation to find the irradiance on a receiving surface due to sky radiation reflected from the ground:

$$\begin{aligned} \text{QRadSWOutIncSkyDiffReflGnd}(\text{RecSurfNum}) &= \\ \text{DifSolarRad} * \rho_{gnd} * \text{ReflFacSkySolGnd}(\text{RecSurfNum}) & \quad (\text{W/m}^2) \end{aligned}$$

where ρ_{gnd} is the solar reflectance of the ground, which is assumed to be uniform over the ground plane but may vary monthly (because of snow cover, for example).

Beam Solar Radiation Diffusely Reflected from Obstructions

This calculation is similar to that for sky solar reflected from obstructions. However, to find the radiance at a hit point on an obstruction a line is drawn from the hit point to center of the sun. From this line it is determined (1) if there is an obstruction between the hit point and the sun, in which case it is assumed that no beam solar reaches the hit point; and (2) if beam solar does reach the hit point, what the solar angle of incidence at that point is.

The calculation is done for the hourly sun positions on each of the design days. It is also done for hourly sun positions on selected days during the weather file run period (the same days for which the shadowing calculations are done).

The factor for diffuse reflection of beam solar radiation from obstructions onto a receiving surface is calculated in subroutine CalcBeamSolDiffuseReflFactors. It is given by:

$$\text{ReflFacBmToDiffSolObs}(\text{RecSurfNum}, \text{Ihr}) = \frac{1}{N_{rec}} \sum_1^{N_{rec}} \sum_{i=1}^{N_{ray}} Hit_{obs,i} Hit_{obs,i,sun} d\Omega_i \cos \alpha_i \rho_{obs,i} \cos \alpha_{sun,obs,i}$$

where

Ihr = hour number

$Hit_{obs,i} = 1$ if ray i from the receiving point hits an obstruction, = 0 otherwise,

$Hit_{obs,i,sun} = 1$ if the line from ray i 's hit point to the sun is unobstructed, = 0 otherwise,

$\alpha_{sun,obs,i}$ is the angle of incidence of the sun on the obstruction.

This factor is used in the timestep calculation to find the diffuse irradiance on a receiving surface due to beam solar diffusely reflected from obstructions:

$$\begin{aligned} \text{QRadSWOutIncBmToDiffReflObs}(\text{RecSurfNum}) &= \text{BeamSolarRad} * \\ (\text{WeightNow} * \text{ReflFacBmToDiffSolObs}(\text{RecSurfNum}, \text{HourOfDay}) + \\ \text{WeightPreviousHour} * \text{ReflFacBmToDiffSolObs}(\text{RecSurfNum}, \text{PreviousHour})) \end{aligned}$$

where $BeamSolarRad$ is the timestep value of beam normal solar intensity (W/m^2), and $WeightNow$ and $WeightPreviousHour$ are time-averaging factors.

Beam Solar Radiation Diffusely Reflected from the Ground

This calculation is the same as that for beam solar diffusely reflected from obstructions except that only rays from a receiving point that hit the ground are considered. The factor for

diffuse reflection of beam solar from the ground onto a receiving surface is calculated in subroutine CalcBeamSolDiffuseReflFactors. It is given by:

$$\text{ReflFacBmToDiffSolGnd}(\text{RecSurfNum}, \text{IHr}) = \frac{1}{N_{\text{rec}}} \sum_{i=1}^{N_{\text{ray}}} \text{Hit}_{\text{gnd},i} \text{Hit}_{\text{gnd},i,\text{sun}} d\Omega_i \cos \alpha_{\text{gnd},i} \cos \alpha_{\text{sun},\text{gnd}}$$

where

$\text{Hit}_{\text{gnd},i} = 1$ if ray i hits the ground, = 0 otherwise,

$\text{Hit}_{\text{gnd},i,\text{sun}} = 1$ if the line from ray i 's hit point of the sun is unobstructed, = 0 otherwise,

$\alpha_{\text{sun},\text{gnd}} =$ angle of incidence of sun on ground (= solar zenith angle).

This factor is used in the timestep calculation to find the diffuse irradiance on a receiving surface due to beam solar diffusely reflected from the ground:

```
QRadSWOutIncBmToDiffReflGnd(RecSurfNum) = BeamSolarRad *  $\rho_{\text{gnd}}$  *
(WeightNow * ReflFacBmToDiffSolGnd(RecSurfNum, HourOfDay) +
WeightPreviousHour * ReflFacBmToDiffSolGnd(RecSurfNum, PreviousHour))
```

Beam Solar Radiation Specularly Reflected from Obstructions

Figure 54 shows schematically how specular (beam-to-beam) reflection from an obstruction is calculated.⁹

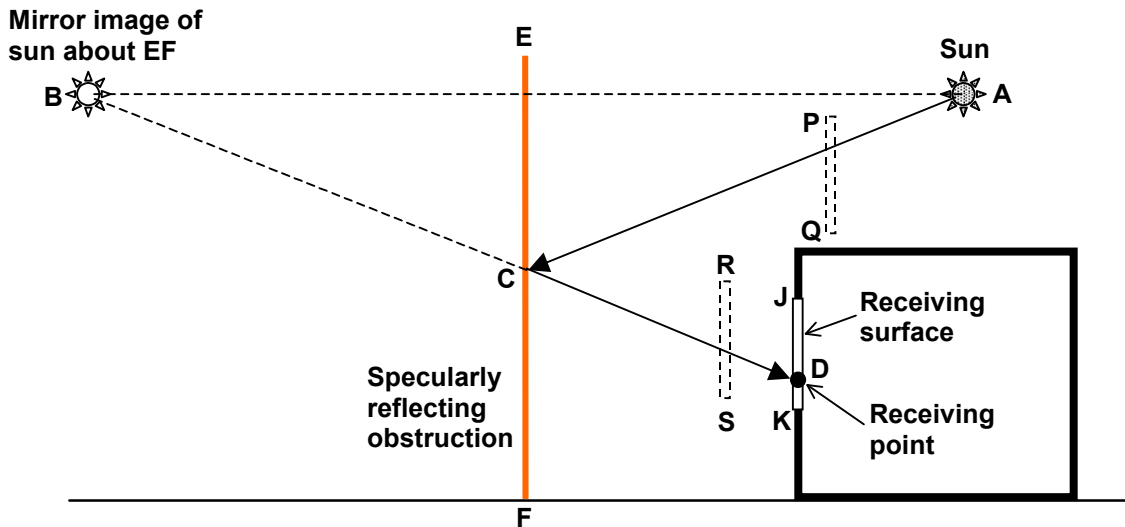


Figure 54. Two-dimensional schematic showing specular reflection from an obstruction such as the glazed façade of a neighboring building. The receiving point receives specularly reflected beam solar radiation if (1) DB passes through specularly reflecting surface EF, (2) CD does not hit any obstructions (such as RS), and (3) AC does not hit any obstructions (such as PQ).

The calculation procedure is as follows:

1. Select receiving point D on receiving surface JK.

⁹ The ground surface is assumed to be diffusely reflecting so there is no specular reflection from the ground. The program could be improved by adding a ground surface specular component, which could be important for snow-cover conditions.

2. Select specularly reflecting surface EF.
3. Find the mirror image, B, of the sun with respect to the plane of EF and construct ray DB.
4. Check if DB passes through EF; if yes, find intersection point C and construct ray CD.
5. Check if CD is obstructed.
6. If no, construct ray AC and check if it is obstructed.
7. If no, find reflected beam irradiance (W/m^2) at D:

$$I_{D,refl}^{bm} = BeamSolarRad * \rho_{spec}(\alpha_C) \cos \alpha_D$$

where

α_C = angle of incidence of beam solar at point C of the obstruction,

$\rho_{spec}(\alpha_C)$ = reflectance of obstruction as a function of the angle of incidence,

α_D = angle of incidence of ray CD on JK.

The factor for specular reflection of beam solar from obstruction onto a receiving surface is calculated in subroutine CalcBeamSolSpecularReflFactors. It is given by:

$$ReflFacBmToBmSolObs(RecSurfNum, IHr) = \sum_{\substack{\text{specularly} \\ \text{reflecting} \\ \text{surfaces}}} \left[\frac{1}{N_{rec}} \sum_1^{N_{rec}} f_{C,glazed} \rho_{spec}(\alpha_C) \cos \alpha_D \right]$$

The program assumes that specular reflection from a surface is due to glazing. If the reflecting surface is a window belonging to the building itself (as in Figure 49), then $f_{C,glazed}$ is the fraction of the window that is glazed (which is 1.0 unless the window has dividers).

If the surface is a shading surface (that represents, for example, the glazed façade of a neighboring building) the surface reflection information is entered with the Shading Surface Reflectance object. This object contains values for:

1. Diffuse solar reflectance of the unglazed part of the shading surface
2. Diffuse visible reflectance of the unglazed part of the shading surface
3. Fraction of shading surface that is glazed
4. Name of glazing construction

In this case $f_{C,glazed}$ is "Fraction of shading surface that is glazed" and $\rho_{spec}(\alpha_C)$ is the front reflectance of the indicated glazing construction as a function of beam solar incidence angle.

The above specular reflection factor is used in the timestep calculation to find the beam irradiance on a receiving surface due to beam-beam reflection from obstructions:

$Q_{RadSWOutIncBmToBmReflObsRecSurfNum} = BeamSolarRad * (WeightNow * ReflFacBmToBmSolObs(RecSurfNum, HourOfDay) + WeightPreviousHour * ReflFacBmToBmSolObs(RecSurfNum, PreviousHour))$

Daylighting and Window Calculations

Daylighting Calculations

The EnergyPlus daylighting model, in conjunction with the thermal analysis, determines the energy impact of daylighting strategies based on analysis of daylight availability, site conditions, window management in response to solar gain and glare, and various lighting control strategies.

The daylighting calculation has three main steps:

1. *Daylight factors*, which are ratios of interior illuminance or luminance to exterior horizontal illuminance, are calculated and stored. The user specifies the coordinates of one or two reference points in each daylit zone. EnergyPlus then integrates over the area of each exterior window in the zone to obtain the contribution of direct light from the window to the illuminance at the reference points, and the contribution of light that reflects from the walls, floor and ceiling before reaching the reference points. Window luminance and window background luminance, which are used to determine glare, are also calculated. Taken into account are such factors as sky luminance distribution, window size and orientation, glazing transmittance, inside surface reflectances, sun control devices such as movable window shades, and external obstructions. Dividing daylight illuminance or luminance by exterior illuminance yields daylight factors. These factors are calculated for the hourly sun positions on sun-paths for representative days of the run period.
2. A daylighting calculation is performed each heat-balance time step when the sun is up. In this calculation the illuminance at the reference points in each zone is found by interpolating the stored daylight factors using the current time step's sun position and sky condition, then multiplying by the exterior horizontal illuminance. If glare control has been specified, the program will automatically deploy window shading, if available, to decrease glare below a specified comfort level. A similar option uses window shades to automatically control solar gain.
3. The electric lighting control system is simulated to determine the lighting energy needed to make up the difference between the daylighting illuminance level and the design illuminance. Finally, the zone lighting electric reduction factor is passed to the thermal calculation, which uses this factor to reduce the heat gain from lights.

The EnergyPlus daylighting calculation is derived from the daylighting calculation in DOE-2.1E, which is described in [Winkelmann, 1983] and [Winkelmann and Selkowitz, 1985]. There are two major differences between the two implementations: (1) In EnergyPlus daylight factors are calculated for four different sky types—clear, clear turbid, intermediate, and overcast; in DOE-2 only two sky types are used—clear and overcast. (2) In EnergyPlus the clear-sky daylight factors are calculated for hourly sun-path sun positions several times a year whereas in DOE-2 these daylight factors are calculated for a set of 20 sun positions that span the annual range of sun positions for a given geographical location.

Daylight Factor Calculation

Table 21. Variables in Daylighting Calculations

Mathematical variable	Description	Units	FORTTRAN variable
$E_{h,sky}$	Exterior horizontal illuminance due to light from the sky	lux	GILSK

$E_{h,sun}$	Exterior horizontal illuminance due to light from the sun	lux	GILSU
d_{sky}, d_{sun}	Interior illuminance factor due to sky, sun related light	-	DFACSK, DFACSU
w_{sky}, w_{sun}	Window luminance factor due to sky, sun related light	cd/lm	SFACSK, SFACSU
b_{sky}, b_{sun}	Window background luminance factor due to sky, sun related light	cd/lm	BFACSK, BFACSU
N	Number of exterior windows in a zone	-	NWD
$\theta_{sky}, \varphi_{sky}$	Azimuth and altitude angles of a point in the sky	radians	THSKY, PHSKY
ψ_{cs}	Clear sky luminance distribution	cd/m ²	-
ψ_{ts}	Clear turbid sky luminance distribution	cd/m ²	-
ψ_{is}	Intermediate sky luminance distribution	cd/m ²	-
ψ_{os}	Overcast sky luminance distribution	cd/m ²	-
φ_{sun}	Altitude angle of the sun	radians or degrees	PHSUN
γ	Angle between point in the sky and the sun; or angle between vertical and ray from reference point to window element	radians	G
L_z	Sky zenith luminance	cd/m ²	ZENL
m	Optical air mass of the atmosphere	m	AM
h	Building altitude	m	Elevation
$E_{h,k}$	Exterior horizontal illuminance for sky type k	lux	-
N_θ, N_φ	Number of azimuth, altitude steps for sky integration		NTH, NPH
\vec{R}_{ref}	Vector from zone origin to reference point	m	RREF
\vec{R}_{win}	Vector from zone origin to window element	m	RWIN
$d\Omega$	Solid angle subtended by window element	steradians	DOMEGA
L_w	Luminance of a window element as seen from reference point	cd/m ²	WLUMSK, WLUMSU
$L_{w,shade}$	Luminance of window element with shade in place	cd/m ²	WLUMSK, WLUMSU

dE_h	Horizontal illuminance at reference point from window element	lux	-
dx, dy	Size of window element	m	DWX, DWY
D	Distance from reference point to window element	m	DIS
B	Angle between window element's outward normal and ray from reference point to window element	radians	-
\hat{R}_{ray}	Unit vector from reference point to window element	-	RAY
\hat{W}_n	Unit vector normal to window element, pointing away from zone	-	WNORM
\hat{W}_{21}	Unit vector along window y-axis	-	W21
\hat{W}_{23}	Unit vector along window x-axis	-	W23
τ_{vis}	Glass visible transmittance	-	TVISB
L	Luminance of sky or obstruction	cd/m ²	ELUM, -
Φ_{FW}	Downgoing luminous flux from a window	lm	FLFW--
Φ_{CW}	Upgoing luminous flux from a window	lm	FLCW--
F_1	First-reflected flux	lm	-
ρ_{FW}	Area-weighted reflectance of floor and upper part of walls	-	SurfaceWindow%RhoFloorWall
ρ_{CW}	Area-weighted reflectance of ceiling and upper part of walls	-	SurfaceWindow%RhoCeilingWall
E_r	Average internally-reflected illuminance	lux	EINTSK, EINTSU
A	Total inside surface area of a zone	m ²	ATOT
ρ	Area-weighted average reflectance of zone interior surfaces	-	ZoneDaylight%AveVisDiffR effect
θ, ϕ	Azimuth and altitude angle of a sky or ground element	radians	TH, PH
$L(\theta, \phi)$	Luminance of sky or ground element at (θ, ϕ)	cd/m ²	HitPointLum--
A_w	Area of glazed part of window	m ²	Surface%Area
β	Angle of incidence, at center of window, of light from a sky or ground element	radians	-
$T(\beta)$	Glazing visible transmittance at incidence angle β		TVISBR

$d\Phi_{inc}$	Luminous flux incident on window from sky or ground element	lm	-
$d\Phi$	Luminous flux from sky or ground element transmitted through window	lm	-
$d\Phi_{FW}, d\Phi_{CW}$	Luminous flux from sky or ground element transmitted through window and going downward, upward	lm	-
$\theta_{min}, \theta_{max}$	Azimuth angle integration limits	radians	THMIN, THMAX
ϕ_w	Window normal altitude angle	radians	-
Φ_{sh}, Φ_{unsh}	Transmitted flux through window with shade, without shade	lm	-
$\Phi_{CW,sh}, \Phi_{FW,sh}$	Upgoing and downgoing portions of transmitted flux through window with shade	lm	-
$\Phi_{CW,unsh}, \Phi_{FW,unsh}$	Upgoing and downgoing portions of transmitted flux through window without shade	lm	-
f	Fraction of hemisphere seen by the inside of window that lies above the window midplane	-	SurfaceWindow%FractionUpgoing
Φ_{inc}	Flux incident on glazing from direct sun	lm	-
f_{sunlit}	Fraction of glazing that is sunlit	-	SunLitFrac
Φ	Transmitted flux from direct sun	-	-
L_{sh}	Luminance of window with shade	cd/m ²	-
L_b	Window background luminance	cd/m ²	BLUM
G	Discomfort glare constant	-	GTOT
G_i	Discomfort glare constant from window i	-	-
ω	Solid angle subtended by window with respect to reference point	steradians	SolidAngAtRefPt
Ω	Solid angle subtended by window with respect to reference point, modified to take direction of occupant view into account	steradians	SolidAngAtRefPtWtd

N_x, N_y	Number of elements in x and y direction that window is divided into for glare calculation	-	NWX, NWY
$p(x_R, y_R)$	Position factor for horizontal and vertical displacement ratios x_R and y_R	-	DayltgGlarePositionFactor
p_H	Hopkinson position factor	-	DayltgGlarePositionFactor
L_b	Window background luminance	cd/m^2	BLUM
E_b	Illuminance on window background	lm	-
E_r	Total internally-reflected component of daylight illuminance	lm	-
E_s	Illuminance setpoint	lm	IllumSetPoint
G_l	Glare index	-	GLINDX

Overview

There are three types of daylight factors: interior illuminance factors, window luminance factors, and window background luminance factors. To calculate these factors the following steps are carried out for each hourly sun position on the sun paths for the design days and for representative days¹⁰ during the simulation run period:

1. Calculate exterior horizontal daylight illuminance from sky and sun for standard (CIE) clear and overcast skies.
2. Calculate interior illuminance, window luminance and window background luminance for each window/reference-point combination, for bare and for shaded window conditions (if a shading device has been specified), for overcast sky and for standard clear sky.
3. Divide by exterior horizontal illuminance to obtain daylight factors.

Interior Illuminance Components

To calculate daylight factors, daylight incident on a window is separated into two components: (1) light that originates from the *sky* and reaches the window directly or by reflection from exterior surfaces; and (2) light that originates from the *sun* and reaches the window directly or by reflection from exterior surfaces. Light from the window reaches the workplane directly or via reflection from the interior surfaces of the room.

For fixed sun position, sky condition (clear or overcast) and room geometry, the sky-related interior daylight will be proportional to the exterior horizontal illuminance, $E_{h,sky}$, due to light from the sky. Similarly, the sun-related interior daylight will be proportional to the exterior horizontal solar illuminance, $E_{h,sun}$.

Daylight Factors

The following daylight factors are calculated:

$$d_{sky} = \frac{\text{Illuminance at reference point due to sky-related light}}{E_{h,sky}}$$

¹⁰ The sun positions for which the daylight factors are calculated are the same as those for which the solar shadowing calculations are done.

$$d_{sun} = \frac{\text{Illuminance at reference point due to sun-related light}}{E_{h,sun}}$$

$$w_{sky} = \frac{\text{Average window luminance due to sky-related light}}{E_{h,sky}}$$

$$w_{sun} = \frac{\text{Average window luminance due to sun-related light}}{E_{h,sun}}$$

$$b_{sky} = \frac{\text{Window background luminance due to sky-related light}}{E_{h,sky}}$$

$$b_{sun} = \frac{\text{Window background luminance due to sun-related light}}{E_{h,sun}}$$

For a daylit zone with N windows these six daylight factors are calculated for each of the following combinations of reference point, window, sky-condition/sun-position and shading device:

$\left[\begin{array}{l} \text{Ref. pt. \#1} \\ \text{Ref. pt. \#2} \end{array} \right]$	$\left[\begin{array}{l} \text{Window \#1} \\ \text{Window \#2} \\ \dots \\ \text{Window \#N} \end{array} \right]$	Clear sky, first sun-up hour	$\left[\begin{array}{l} \text{Unshaded window} \\ \text{Shaded window} \\ \text{(if shade assigned)} \end{array} \right]$
		Clear/turbid sky, first sun-up hour	
		Intermediate sky, first sun-up hour	
		Overcast sky, first sun-up hour	
		...	
		Clear sky, last sun-up hour	
		Clear/turbid sky, last sun-up hour	
		Intermediate sky, last sun-up hour	
		Overcast sky, last sun-up hour	

Sky Luminance Distributions

The luminance distribution of the sky is represented as a superposition of four standard CIE skies using the approach described in (Perez et al. 1990). The standard skies are as follows.

Clear Sky

The clear sky luminance distribution has the form (Kittler, 1965; CIE, 1973)

$$\psi_{cs}(\theta_{sky}, \phi_{sky}) = L_z \frac{(0.91 + 10e^{-3\gamma} + 0.45 \cos^2 \gamma)(1 - e^{-0.32 \text{cosec} \phi_{sky}})}{0.27385(0.91 + 10e^{-3(\frac{\pi}{2} - \phi_{sun})} + 0.45 \sin^2 \phi_{sun})}$$

Here, L_z is the zenith luminance (i.e., the luminance of the sky at a point directly overhead). In the calculation of daylight factors, which are ratios of interior and exterior illumination

quantities that are both proportional to L_z , the zenith luminance cancels out. For this reason we will use $L_z = 1.0$ for all sky luminance distributions.

The various angles, which are defined in the building coordinate system, are shown in Figure 48. The angle, γ , between sun and sky element is given by

$$\gamma = \cos^{-1} \left[\sin \phi_{sky} \sin \phi_{sun} + \cos \phi_{sky} \cos \phi_{sun} \cos(\theta_{sky} - \theta_{sun}) \right]$$

The general characteristics of the clear-sky luminance distribution are a large peak near the sun; a minimum at a point on the other side of the zenith from the sun, in the vertical plane containing the sun; and an increase in luminance as the horizon is approached.

Clear Turbid Sky

The clear turbid sky luminance distribution has the form [Matsuura, 1987]

$$\psi_{ts}(\theta_{sky}, \phi_{sky}) = L_z \frac{(0.856 + 16e^{-3\gamma} + 0.3 \cos^2 \gamma)(1 - e^{-0.32 \operatorname{cosec} \phi_{sky}})}{0.27385(0.856 + 10e^{-3(\frac{\pi}{2} - \phi_{sun})} + 0.3 \sin^2 \phi_{sun})}$$

Intermediate Sky

The intermediate sky luminance distribution has the form [Matsuura, 1987]

$$\psi_{is}(\theta_{sky}, \phi_{sky}) = L_z Z_1 Z_2 / (Z_3 Z_4)$$

where

$$Z_1 = \left[1.35(\sin(3.59\phi_{sky} - 0.009) + 2.31) \sin(2.6\phi_{sun} + 0.316) + \phi_{sky} + 4.799 \right] / 2.326$$

$$Z_2 = \exp \left[-0.563\gamma \{ (\phi_{sun} - 0.008)(\phi_{sky} + 1.059) + 0.812 \} \right]$$

$$Z_3 = 0.99224 \sin(2.6\phi_{sun} + 0.316) + 2.73852$$

$$Z_4 = \exp \left[-0.563 \left(\frac{\pi}{2} - \phi_{sun} \right) \{ 2.6298(\phi_{sun} - 0.008) + 0.812 \} \right]$$

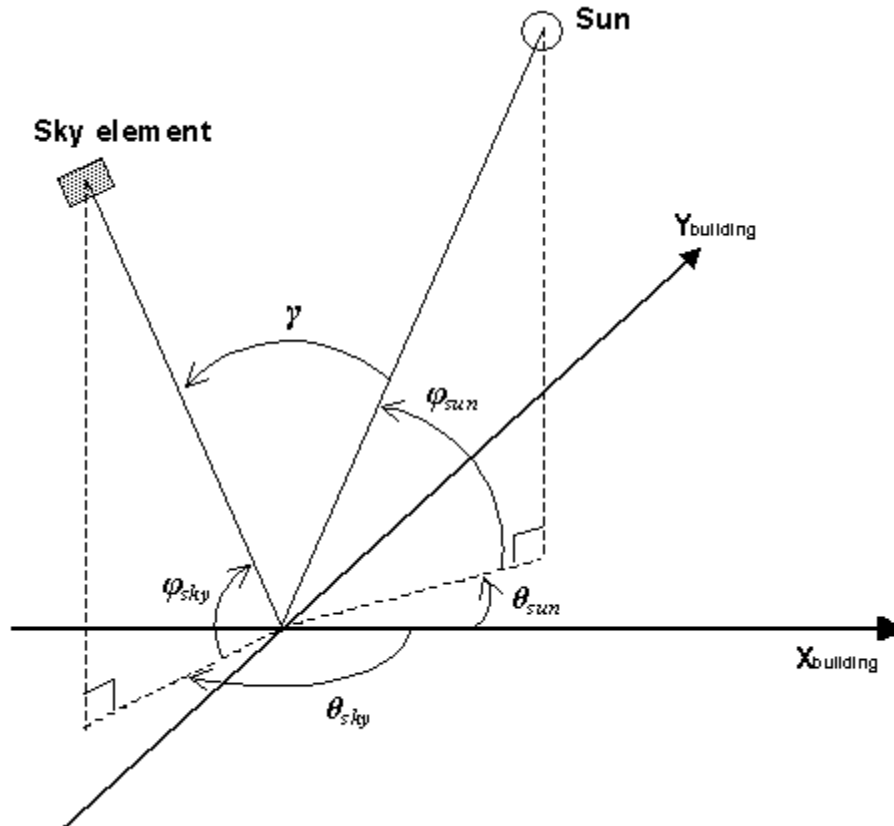


Figure 55. Angles appearing in the expression for the clear-sky luminance distribution.

Overcast Sky

The overcast sky luminance distribution has the form [Moon & Spencer, 1942]

$$\psi_{os}(\phi_{sky}) = L_z \frac{1 + 2 \sin \phi_{sky}}{3}$$

Unlike the clear sky case, the overcast sky distribution does not depend on the solar azimuth or the sky azimuth. Note that at fixed solar altitude the zenith ($\phi_{sky} = \pi/2$) is three times brighter than the horizon ($\phi_{sky} = 0$).

Direct Normal Solar Illuminance

For purposes of calculating daylight factors associated with beam solar illuminance, the direct normal solar illuminance is taken to be 1.0 W/m^2 . The actual direct normal solar illuminance, determined from direct normal solar irradiance from the weather file and empirically-determined luminous efficacy, is used in the time-step calculation.

Exterior Horizontal Illuminance

The illuminance on an unobstructed horizontal plane due to diffuse radiation from the sky is calculated for each of the four sky types by integrating over the appropriate sky luminance distribution:

$$E_{h,k} = \int_0^{2\pi} \int_0^{\pi/2} \psi_k(\theta_{sky}, \phi_{sky}) \sin \phi_{sky} \cos \theta_{sky} d\theta_{sky} d\phi_{sky}$$

where $k = cs, ts, is$ or os . The integral is evaluated as a double summation:

$$E_{h,k} = \sum_{i=1}^{N_\theta} \sum_{j=1}^{N_\phi} \psi_k(\theta_{sky}(i), \phi_{sky}(j)) \sin \phi_{sky}(j) \cos \theta_{sky}(j) \Delta\theta_{sky} \Delta\phi_{sky}$$

where

$$\theta_{sky}(i) = (i - 1/2) \Delta\theta_{sky}$$

$$\phi_{sky}(j) = (j - 1/2) \Delta\phi_{sky}$$

$$\Delta\theta_{sky} = 2\pi / N_\theta$$

$$\Delta\phi_{sky} = \pi / 2N_\phi$$

$N_\theta = 18$ and $N_\phi = 8$ were found to give a $\pm 1\%$ accuracy in the calculation of $E_{h,k}$.

Direct Component of Interior Daylight Illuminance

The direct daylight illuminance at a reference point from a particular window is determined by dividing the window into an x-y grid and finding the flux reaching the reference point from each grid element. The geometry involved is shown in Figure 56. The horizontal illuminance at the reference point, \vec{R}_{ref} , due to a window element is

$$dE_h = L_w d\Omega \cos \gamma$$

where L_w is the luminance of the window element as seen from the reference point.

The subtended solid angle is approximated by

$$d\Omega = \frac{dxdy}{D^2} \cos B \quad (164)$$

where

$$D = \left| \vec{R}_{win} - \vec{R}_{ref} \right|$$

$\cos B$ is found from

$$\cos B = \hat{R}_{ray} \cdot \hat{W}_n$$

where

$$\vec{R}_{ray} = (\vec{R}_{win} - \vec{R}_{ref}) / D$$

$$\hat{W}_n = \text{window outward normal} = \hat{W}_{21} \times \hat{W}_{23} = \frac{\vec{W}_1 - \vec{W}_2}{|\vec{W}_1 - \vec{W}_2|} \times \frac{\vec{W}_3 - \vec{W}_2}{|\vec{W}_3 - \vec{W}_2|}$$

Equation (164) becomes exact as dx/D and $dy/D \rightarrow 0$ and is accurate to better than about 1% for $dx \leq D/4$ and $dy \leq D/4$.

The net illuminance from the window is obtained by summing the contributions from all the window elements:

$$E_h = \sum_{\text{window elements}} L_w d\Omega \cos \gamma \quad (165)$$

In performing the summation, window elements that lie below the workplane ($\cos \gamma < 0$) are omitted since light from these elements cannot reach the workplane directly.

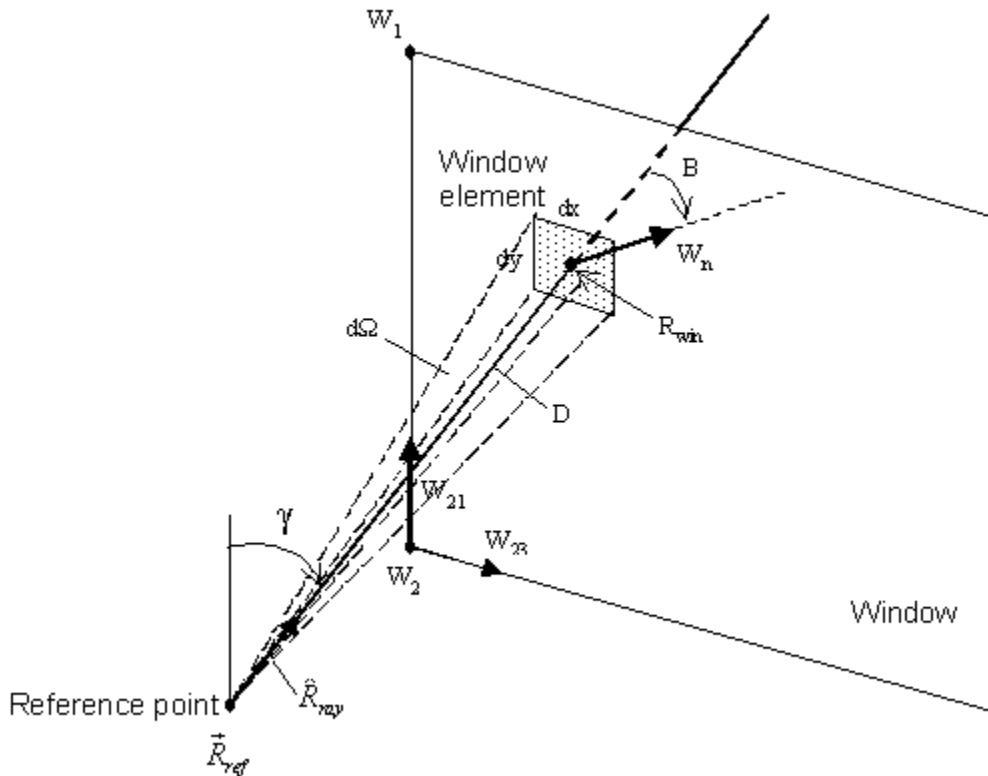


Figure 56. Geometry for calculation of direct component of daylight illuminance at a reference point. Vectors R_{ref} , W_1 , W_2 , W_3 and R_{win} are in the building coordinate system.

Unshaded Window

For the unshaded window case, the luminance of the window element is found by projecting the ray from reference point to window element and determining whether it intersects the sky or an exterior obstruction such as an overhang. If L is the corresponding luminance of the sky or obstruction, the window luminance is

$$L_w = L \tau_{\text{vis}} (\cos B)$$

where τ_{vis} is the visible transmittance of the glass for incidence angle B .

Exterior obstructions are generally opaque (like fins, overhangs, neighboring buildings, and the building's own wall and roof surfaces) but can be transmitting (like a tree or translucent awning). Exterior obstructions are assumed to be non-reflecting. If L_{sky} is the sky luminance and τ_{obs} is the transmittance of the obstruction (assumed independent of incidence angle), then $L = L_{sky}\tau_{obs}$. Interior obstructions are assumed to be opaque ($\tau_{obs} = 0$).

Shaded Window

For the window-plus-shade case the shade is assumed to be a perfect diffuser, i.e., the luminance of the shade is independent of angle of emission of light, position on shade, and angle of incidence of solar radiation falling on the shade. Closely-woven drapery fabric and translucent roller shades are closer to being perfect diffusers than Venetian blinds or other slatted devices, which usually have non-uniform luminance characteristics.

The calculation of the window luminance with the shade in place, $L_{w,sh}$, is described in [Winkelmann, 1983]. The illuminance contribution at the reference point from a shaded window element is then given by Eq. (152) with $L_w = L_{w,sh}$.

Internally-Reflected Component of Interior Daylight Illuminance

Daylight reaching a reference point after reflection from interior surfaces is calculated using the *split-flux* method [Hopkinson et al., 1954], [Lynes, 1968]. In this method the daylight transmitted by the window is split into two parts—a downward-going flux, Φ_{FW} (lumens), which falls on the floor and portions of the walls below the imaginary horizontal plane passing through the center of the window (*window midplane*), and an upward-going flux, Φ_{CW} , that strikes the ceiling and portions of the walls above the window midplane. A fraction of these fluxes is absorbed by the room surfaces. The remainder, the first-reflected flux, F_1 , is approximated by

$$F_1 = \Phi_{FW}\rho_{FW} + \Phi_{CW}\rho_{CW}$$

where ρ_{FW} is the area-weighted average reflectance of the floor and those parts of the walls below the window midplane, and ρ_{CW} is the area-weighted average reflectance of the ceiling and those parts of the walls above the window midplane.

To find the final average internally-reflected illuminance, E_r , on the room surfaces (which in this method is uniform throughout the room) a flux balance is used. The total reflected flux absorbed by the room surfaces (or lost through the windows) is $AE_r(1-\rho)$, where A is the total inside surface area of the floor, walls, ceiling and windows in the room, and ρ is the area-weighted average reflectance of the room surfaces, including windows. From conservation of energy

$$AE_r(1-\rho) = F_1$$

or

$$E_r = \frac{\Phi_{FW}\rho_{FW} + \Phi_{CW}\rho_{CW}}{A(1-\rho)}$$

This procedure assumes that the room behaves like an integrating sphere with perfectly diffusing interior surfaces and with no internal obstructions. It therefore works best for rooms

that are close to cubical in shape, have matte surfaces (which is usually the case), and have no internal partitions. Deviations from these conditions, such as would be the case for rooms whose depth measured from the window-wall is more than three times greater than ceiling height, can lead to substantial inaccuracies in the split-flux calculation.

Transmitted Flux from Sky and Ground

The luminous flux incident on the center of the window from a luminous element of sky or ground at angular position (θ, ϕ) , of luminance $L(\theta, \phi)$, and subtending a solid angle $\cos \phi d\theta d\phi$ is

$$d\Phi_{inc} = A_w L(\theta, \phi) \cos \beta \cos \phi d\theta d\phi$$

The transmitted flux is

$$d\Phi = d\Phi_{inc} T(\beta)$$

where $T(\beta)$ is the window transmittance for light at incidence angle β . This transmittance depends on whether or not the window has a shade.

For an unshaded window the total downgoing transmitted flux is obtained by integrating over the part of the exterior hemisphere seen by the window that lies above the window midplane. This gives

$$\Phi_{FW,unshaded} = A_w \int_{\theta_{min}}^{\theta_{max}} \int_0^{\pi/2} L(\theta, \phi) T(\beta) \cos \beta \cos \phi d\theta d\phi \quad (166)$$

The upgoing flux is obtained similarly by integrating over the part of the exterior hemisphere that lies below the window midplane:

$$\Phi_{CW,unshaded} = A_w \int_{\theta_{min}}^{\theta_{max}} \int_{\pi/2-\phi_w}^0 L(\theta, \phi) T(\beta) \cos \beta \cos \phi d\theta d\phi \quad (167)$$

where ϕ_w is the angle the window outward normal makes with the horizontal plane.

For a window with a diffusing shade the total transmitted flux is

$$\Phi_{sh} = A_w \int_{\theta_{min}}^{\theta_{max}} \int_{\pi/2-\phi_w}^{\pi/2} L(\theta, \phi) T(\beta) \cos \beta \cos \phi d\theta d\phi \quad (168)$$

The downgoing and upgoing portions of this flux are

$$\Phi_{FW,sh} = \Phi(1 - f)$$

$$\Phi_{CW,sh} = \Phi f$$

where f , the fraction of the hemisphere seen by the inside of the window that lies above the window midplane, is given by

$$f = 0.5 - \phi_w / \pi$$

For a vertical window ($\phi_w = 0$) the up- and down-going transmitted fluxes are equal:

$$\Phi_{FW,sh} = \Phi_{CW,sh} = \Phi / 2 .$$

For a horizontal skylight ($\phi_w = \pi / 2$):

$$\Phi_{FW,sh} = \Phi, \Phi_{CW,sh} = 0$$

The limits of integration of θ in Equations (153), (154) and (155) depend on ϕ . From [Figure 12 - Winkelmann, 1983] we have

$$\sin \alpha = \sin(A - \pi / 2) = \frac{\sin \phi \tan \phi_w}{\cos \phi}$$

which gives

$$A = \cos^{-1}(\tan \phi \tan \phi_w)$$

Thus

$$\theta_{\min} = -\left| \cos^{-1}(-\tan \phi \tan \phi_w) \right|$$

$$\theta_{\max} = \left| \cos^{-1}(-\tan \phi \tan \phi_w) \right|$$

Transmitted Flux from Direct Sun

The flux incident on the window from direct sun is

$$\Phi_{inc} = A_w E_{DN} \cos \beta f_{sunlit}$$

The transmitted flux is

$$\Phi = T(\beta) \Phi_{inc}$$

where T is the net transmittance of the window glazing (plus shade, if present).

For an unshaded window all of the transmitted flux is downward since the sun always lies above the window midplane. Therefore

$$\Phi_{FW,unsh} = \Phi$$

$$\Phi_{CW,unsh} = 0$$

For a window with a diffusing shade

$$\Phi_{FW,sh} = \Phi(1 - f)$$

$$\Phi_{CW,sh} = \Phi f$$

Luminance of Shaded Window

The luminance of a shaded window is determined at the same time that the transmitted flux is calculated. It is given by

$$L_{sh} = \frac{1}{\pi} \int_{\theta_{\min}}^{\theta_{\max}} \int_{\pi/2-\phi_w}^{\pi/2} L(\theta, \phi) T(\beta) \cos \beta \cos \phi d\theta d\phi$$

Daylight Discomfort Glare

The discomfort glare at a reference point due to luminance contrast between a window and the interior surfaces surrounding the window is given by [Hopkinson, 1970] and [Hopkinson, 1972]:

$$G = \frac{L_w^{1.6} \Omega^{0.8}}{L_b + 0.07 \omega^{0.5} L_w}$$

where

G = discomfort glare constant

L_w = average luminance of the window as seen from the reference point

Ω = solid angle subtended by window, modified to take direction of occupant view into account

L_b = luminance of the background area surrounding the window

By dividing the window into N_x by N_y rectangular elements, as is done for calculating the direct component of interior illuminance, we have

$$L_w = \frac{\sum_{j=1}^{N_y} \sum_{i=1}^{N_x} L_w(i, j)}{N_x N_y}$$

where $L_w(i, j)$ is the luminance of element (i, j) as seen from the reference point.

Similarly,

$$\omega = \sum_{j=1}^{N_y} \sum_{i=1}^{N_x} d\omega(i, j)$$

where $d\omega(i, j)$ is the solid angle subtended by element (i, j) with respect to the reference point.

The modified solid angle is

$$\Omega = \sum_{j=1}^{N_y} \sum_{i=1}^{N_x} d\omega(i, j) p(x_R, y_R)$$

where p is a "position factor" [Petherbridge & Longmore, 1954] that accounts for the decrease in visual excitation as the luminous element moves away from the line of sight. This factor depends on the horizontal and vertical displacement ratios, x_R and y_R (Figure 57), given by

$$x_R(i, j) = \frac{|A^2 - (YD)^2|^{1/2}}{RR}$$

$$y_R(i, j) = |YD / RR|$$

where

$$RR = D(\hat{R}_{ray} \cdot \hat{v}_{view})$$

$$A^2 = D^2 - (RR)^2$$

$$YD = R_{win}(3) - R_{ref}(3)$$

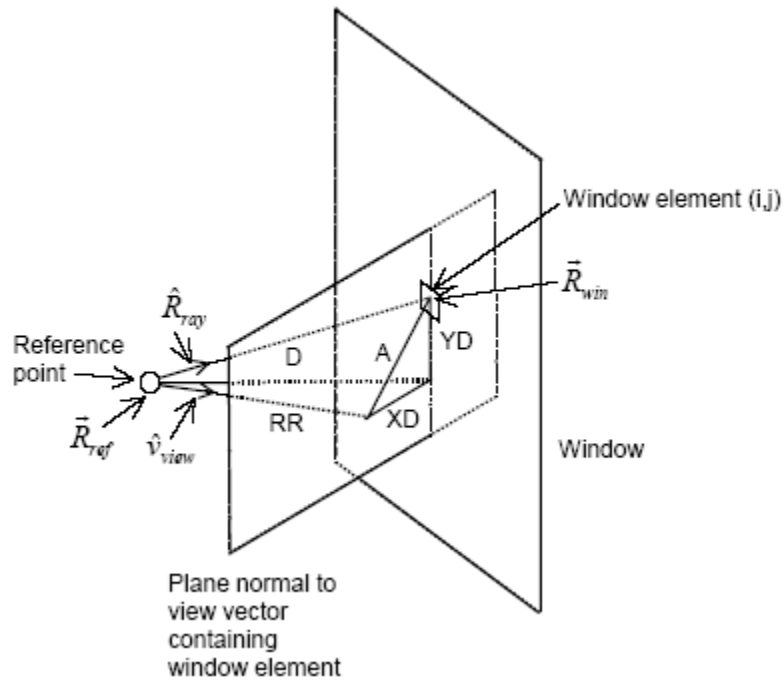


Figure 57. Geometry for calculation of displacement ratios used in the glare formula.

The factor p can be obtained from graphs given in [Petherbridge & Longmore, 1954] or it can be calculated from tabulated values of p_H , the Hopkinson position factor [Hopkinson, 1966], since $p = p_H^{1.25}$. The values resulting from the latter approach are given in Table 22. Interpolation of this table is used in EnergyPlus to evaluate p at intermediate values of x_R and y_R .

Table 22. Position factor for glare calculation

		x_R : Horizontal Displacement Factor							
		0	0.5	1.0	1.5	2.0	2.5	3.0	>3.0
y_R : Vertical Displacement Factor	0	1.00	0.492	0.226	0.128	0.081	0.061	0.057	0
	0.5	0.123	0.119	0.065	0.043	0.029	0.026	0.023	0
	1.0	0.019	0.026	0.019	0.016	0.014	0.011	0.011	0
	1.5	0.008	0.008	0.008	0.008	0.008	0.006	0.006	0

	2.0	0	0	0.003	0.003	0.003	0.003	0.003	0
	>2.0	0	0	0	0	0	0	0	0

The background luminance is

$$L_b = E_b \rho_b$$

where ρ_b is approximated by the average interior surface reflectance of the entire room and

$$E_b = \max(E_r, E_s)$$

where E_r is the total internally-reflected component of daylight illuminance produced by all the windows in the room and E_s is the illuminance setpoint at the reference point at which glare is being calculated. A precise calculation of E_b is not required since the glare index (see next section) is logarithmic. A factor of two variation in E_b generally produces a change of only 0.5 to 1.0 in the glare index.

Glare Index

The net daylight glare at a reference point due to all of the windows in a room is expressed in terms of a *glare index* given by

$$G_T = 10 \log_{10} \sum_{i=1}^{\text{number of windows}} G_i$$

where G_i is the glare constant at the reference point due to the i^{th} window

Time-Step Daylighting Calculation

Overview

A daylighting calculation is performed each time step that the sun is up for each zone that has one or two daylighting reference points specified. The exterior horizontal illuminance from the sun and sky is determined from solar irradiance data from the weather file. The interior illuminance at each reference point is found for each window by interpolating the daylight illuminance factors for the current sun position, then, for sky-related interior illuminance, multiplying by the exterior horizontal illuminance from the appropriate sky types that time step, and, for sun-related interior illuminance, multiplying by the exterior horizontal solar illuminance that time step. By summation, the net illuminance and glare due to all of the windows in a zone are found. If glare control has been specified window shading (by movable shading devices or switchable glazing) is deployed to reduce glare. Finally the illuminance at each reference point for the final window and shade configuration is used by the lighting control system simulation to determine the electric lighting power required to meet the illuminance setpoint at each reference point.

Table 23. Variables in Time-Step Calculations

Mathematical variable	Description	Units	FORTRAN variable
$S_{\text{norm,dir}}$	Direct normal solar irradiance	W/m ²	BeamSolarRad

$S_{h,dif}$	Exterior diffuse horizontal solar irradiance	W/m^2	SDIFH, DifSolarRad
$S_{h,dir}$	Exterior direct horizontal solar irradiance	W/m^2	SDIRH
Z	Solar zenith angle	radians	Zeta
m	Relative optical air mass	-	AirMass
Δ	Sky brightness	-	SkyBrightness
ϵ	Sky clearness	-	SkyClearness
k, k'	Sky type index	-	ISky
$s_{k,k'}$	Interpolation factor for skies k and k'	-	SkyWeight
$\psi_{k,k'}$	Sky luminance distribution formed from linear interpolation of skies k and k'	cd/m^2	-
f_k	Fraction of sky that is type k	-	-
$E_{h,k}$	Horizontal illuminance from sky type k	cd/m^2	HorIII Sky
$E_{h,sky}$	Exterior horizontal illuminance from sky	lux	HISKF
$E_{h,sun}$	Exterior horizontal illuminance from sun	lux	HISUNF
η_{dif}, η_{dir}	Luminous efficacy of diffuse and direct solar radiation	lm/W	DiffLumEff, DirLumEff
I_{win}	Interior illuminance from a window	lux	DayIII lum
S_{win}	Window luminance	cd/m^2	SourceLumFromWinAtRefPt
B_{win}	Window background luminance	cd/m^2	BACLUM
$d_{sun}, d_{sky,k}$	Interior illuminance factor for sun, for sky of type k	-	DayIIIFacSun, DFSUHR, DayIIIFacSky, DFSUHR
$w_{sun}, w_{sky,k}$	Window luminance factor for sun, for sky of type k	-	DayISourceFacSun, SFSUHR, DayISourceFacSky, SFSKHR
$b_{sun}, b_{sky,k}$	Window background luminance factor for sun, for sky of type k	-	DayIBackFacSun, BFSUHR, DayIBackFacSky, BFSKHR
w_j	Weighting factor for time step interpolation	-	WeightNow
i_L	Reference point index	-	IL
i_s	Window shade index	-	IS
I_{tot}	Total daylight illuminance at reference point	lux	DayIII lum

B_{tot}, B	Total window background luminance	cd/m^2	BLUM
I_{set}	Illuminance setpoint	lux	ZoneDaylight%IllumSetPoint
f_L	Fractional electric lighting output	-	FL
f_P	Fractional electric lighting input power	-	FP
N_L	Number of steps in a stepped control system	-	LightControlSteps
M_P	Lighting power multiplier	-	ZonePowerReductionFactor

Time-Step Sky Luminance

The sky luminance distribution, ψ , for a particular time step is expressed as a linear interpolation of two of the four standard skies — ψ_{cs} , ψ_{ts} , ψ_{is} and ψ_{os} — described above under “Sky Luminance Distributions.” The two sky types that are interpolated depend on the value of the sky clearness. The interpolation factors are a function of sky clearness and sky brightness (Perez et al., 1990). Sky clearness is given by

$$\varepsilon = \frac{S_{h,dif} + S_{norm,dir} + \kappa Z^3}{S_{h,dif} + 1 + \kappa Z^3}$$

where $S_{h,dif}$ is the diffuse horizontal solar irradiance, $S_{norm,dir}$ is the direct normal solar irradiance, Z is the solar zenith angle and κ is a constant equal to 1.041 for Z in radians.

Sky brightness is given by

$$\Delta = S_{h,dif} m / S_{norm,dir}^{ext}$$

where m is the relative optical air mass and $S_{norm,dir}^{ext}$ is the extraterrestrial direct normal solar irradiance.

If $\varepsilon \leq 1.2$

$$\psi_{is,os} = s_{is,os} \psi_{is} + (1 - s_{is,os}) \psi_{os}$$

where ψ_{is} is the intermediate sky luminance distribution, ψ_{os} is the overcast sky luminance distribution, and

$$s_{is,os} = \min \{1, \max[0, (\varepsilon - 1) / 0.2, (\Delta - 0.05) / 4]\}$$

If $1.2 < \varepsilon \leq 3$

$$\psi_{ts,is} = s_{ts,is} \psi_{ts} + (1 - s_{ts,is}) \psi_{is}$$

where ψ_{ts} is the clear turbid sky luminance distribution and

$$s_{ts,ts} = (\varepsilon - 1.2) / 1.8$$

If $\varepsilon > 3$

$$\psi_{cs,ts} = s_{cs,ts} \psi_{cs} + (1 - s_{cs,ts}) \psi_{ts}$$

where ψ_{cs} is the clear sky luminance distribution and

$$s_{cs,ts} = \min[1, (\varepsilon - 3) / 3]$$

Interior Illuminance

For each time step the interior illuminance, I_{win} , from a window is calculated as follows by multiplying daylight factors and exterior illuminance.

First, the sun- and sky-related daylight illuminance factors for the time step are determined by interpolation of the hourly factors:

$$\bar{d}_{sun}(i_L, i_S) = w_j d_{sun}(i_L, i_S, i_h) + (1 - w_j) d_{sun}(i_L, i_S, i_h + 1)$$

$$\bar{d}_{sky,k}(i_L, i_S) = w_j d_{sky,k}(i_L, i_S, i_h) + (1 - w_j) d_{sky,k}(i_L, i_S, i_h + 1)$$

where i_L is the reference point index (1 or 2), i_S is the window shade index (1 for unshaded window, 2 for shaded window), i_h is the hour number, and k is the sky type index. For the j th time step in an hour, the time-step interpolation weight is given by

$$w_j = 1 - \min[1, j / N_t]$$

where N_t is the number of time steps per hour.

The interior illuminance from a window is calculated as

$$I_{win}(i_L, i_S) = \bar{d}_{sun} E_{h,sun} + [\bar{d}_{sky,k}(i_L, i_S) f_k + \bar{d}_{sky,k'}(i_L, i_S) f_{k'}] E_{h,sky}$$

where $E_{h,sun}$ and $E_{h,sky}$ are the exterior horizontal illuminance from the sun and sky, respectively, and f_k and $f_{k'}$ are the fraction of the exterior horizontal illuminance from the sky that is due to sky type k and k' , respectively.

The horizontal illuminance from sun and sky are given by

$$E_{h,sun} = \eta_{dir} S_{norm,dir} \cos Z$$

$$E_{h,sky} = \eta_{dif} S_{h,dir}$$

where Z is the solar zenith angle, η_{dif} is the luminous efficacy (in lumens/Watt) of diffuse solar radiation from the sky and η_{dir} is the luminous efficacy of direct radiation from the sun. The efficacies are calculated from direct and global solar irradiance using a method described in (Perez et al, 1990).

The fractions f_k and $f_{k'}$ are given by

$$f_k = \frac{s_{k,k'} E_{h,k}}{s_{k,k'} E_{h,k} + (1 - s_{k,k'}) E_{h,k'}}$$

$$f_{k'} = \frac{(1 - s_{k,k'}) E_{h,k'}}{s_{k,k'} E_{h,k} + (1 - s_{k,k'}) E_{h,k'}}$$

where $E_{h,k}$ and $E_{h,k'}$ are the horizontal illuminances from skies k and k' , respectively (see “Exterior Horizontal Luminance,” above), and $s_{k,k'}$ is the interpolation factor for skies k and k' (see “Time-Step Sky Luminance,” above). For example, if $\varepsilon > 3$, $k = cs$ (clear sky), $k' = ts$ (clear turbid sky) and

$$s_{k,k'} = s_{cs,ts} = \min[1, (\varepsilon - 3) / 3]$$

Similarly, the window source luminance, S_{win} , and window background luminance, B_{win} , for a window are calculated from

$$S_{win}(i_L, i_S) = \bar{w}_{sun} E_{h,sun} + [\bar{w}_{sky,k}(i_L, i_S) f_k + \bar{w}_{sky,k'}(i_L, i_S) f_{k'}] E_{h,sky}$$

$$B_{win}(i_L, i_S) = \bar{b}_{sun} E_{h,sun} + [\bar{b}_{sky,k}(i_L, i_S) f_k + \bar{b}_{sky,k'}(i_L, i_S) f_{k'}] E_{h,sky}$$

The total illuminance at a reference point from all of the exterior windows in a zone is

$$I_{tot}(i_L) = \sum_{\substack{\text{windows} \\ \text{in zone}}} I_{win}(i_S, i_L)$$

where $i_S = 1$ if the window is unshaded and $i_S = 2$ if the window is shaded that time step. (Before the illuminance calculation is done the window shading control will have been simulated to determine whether or not the window is shaded.)

Similarly, the total background luminance is calculated:

$$B_{tot}(i_L) = \sum_{\substack{\text{windows} \\ \text{in zone}}} B_{win}(i_S, i_L)$$

Glare Index

The net glare index at each reference point is calculated as

$$G_I(i_L) = 10 \log_{10} \sum_{\substack{\text{windows} \\ \text{in zone}}} \frac{S_{win}(i_L, i_S)^{1.6} \Omega(i_L)^{0.8}}{B(i_L) + 0.07 \omega(i_L)^{0.5} S_{win}(i_L, i_S)}$$

where

$$B(i_L) = \max(B_{win}(i_L), \rho_b I_{set}(i_L))$$

In the last relationship, the background luminance is approximated as the larger of the background luminance from daylight and the average background luminance that would be

produced by the electric lighting at full power if the illuminance on the room surfaces were equal to the setpoint illuminance. In a more detailed calculation, where the luminance of each room surface is separately determined, $B(i_L)$ would be better approximated as an area-weighted luminance of the surfaces surrounding a window, taking into account the luminance contribution from the electric lights.

Glare Control Logic

If glare control has been specified and the glare index at either reference point exceeds a user-specified maximum value, $G_{I,max}$, then the windows in the zone are shaded one by one in attempt to bring the glare at both points below $G_{I,max}$. (Each time a window is shaded the glare and illuminance at each reference point is recalculated.) The following logic is used:

- 3) If there is only one reference point, shade a window if it is unshaded and shading it decreases the glare, even if it does not decrease the glare below $G_{I,max}$. Note that if a window has already been shaded, say to control solar gain, it will be left in the shaded state.
- 4) If there are two reference points, then:
 - If glare is too high at both points, shade the window if it decreases glare at both points.
 - If glare is too high only at the first point, shade the window if the glare at the first point decreases, and the glare at the second point stays below $G_{I,max}$.
 - If glare is too high only at the second point, shade the window if the glare at the second point decreases, and the glare at the first point stays below $G_{I,max}$.
- 5) Shades are closed in the order of window input until glare at both points is below $G_{I,max}$, or until there are no more windows left to shade.

Lighting Control System Simulation

Once the final daylight illuminance value at each reference point has been determined, the electric lighting control is simulated. The fractional electric lighting output, f_L , required to meet the setpoint at reference point i_L is given by

$$f_L(i_L) = \max \left[0, \frac{I_{set}(i_L) - I_{tot}(i_L)}{I_{set}(i_L)} \right]$$

Here, I_{set} is the illuminance setpoint and I_{tot} is the daylight illuminance at the reference point. This relationship assumes that the electric lights at full power produce an illuminance equal to I_{set} at the reference point.

The fractional electric lighting input power, f_P , corresponding to f_L is then calculated. The relationship between f_P and f_L depends on the lighting control type.

Continuous Dimming Control

For a continuously-dimmable control system, it is assumed that f_P is constant and equal to $f_{P,min}$ for $f_L < f_{L,min}$ and that f_P increases linearly from $f_{P,min}$ to 1.0 as f_L increases from $f_{L,min}$ to 1.0 (Figure 58). This gives

$$f_P = \begin{cases} f_{P,min} & \text{for } f_L < f_{L,min} \\ \frac{f_L + (1 - f_L)f_{P,min} - f_{L,min}}{1 - f_{L,min}} & \text{for } f_{L,min} \leq f_L \leq 1 \end{cases}$$

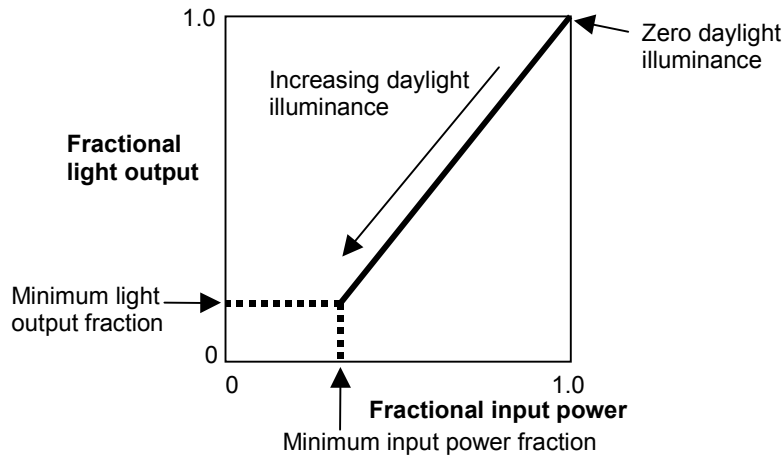


Figure 58. Control action for a continuous dimming system.

Continuous/Off Dimming Control

A “continuous/off” dimming system has the same behavior as a continuous dimming system except that the lights switch off for $f_L < f_{L,min}$ rather than staying at $f_{P,min}$.

Stepped Control

For a stepped control system, f_P takes on discrete values depending on the range of f_L and the number of steps, N_L (Figure 59). This gives

$$f_P = \begin{cases} 0, & \text{if } f_L = 0 \\ \frac{\text{int}(N_L f_L) + 1}{N_L}, & \text{for } 0 < f_L < 1 \\ 1, & \text{if } f_L = 1 \end{cases}$$

If a lighting control probability, p_L , is specified, f_P is set one level higher a fraction of the time equal to $1 - p_L$. Specifically, if $f_P < 1$, $f_P \rightarrow f_P + 1/N_L$ if a random number between 0 and 1 exceeds p_L . This can be used to simulate the uncertainty associated with manual switching of lights.

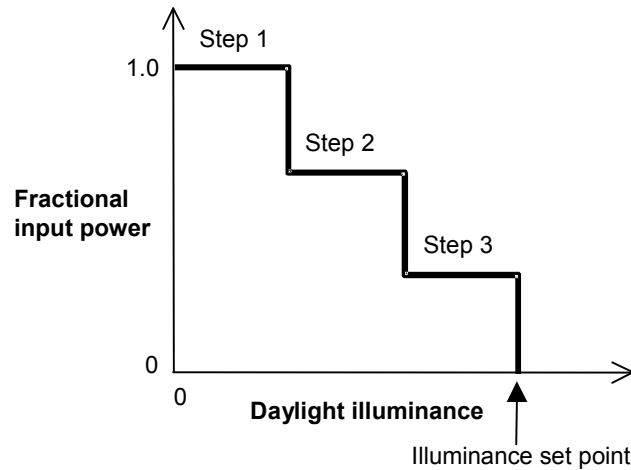


Figure 59. Stepped lighting control with three steps.

Lighting Power Reduction

Using the value of f_p at each reference point and the fraction f_z of the zone controlled by the reference point, the net lighting power multiplier, M_p , for the entire zone is calculated; this value multiplies the lighting power output without daylighting.

$$M_p = \sum_{i_L=1}^2 f_p(i_L) f_z(i_L) + \left(1 - \sum_{i_L=1}^2 f_z(i_L) \right)$$

In this expression, the term to the right in the parentheses corresponds to the fraction of the zone not controlled by either reference point. For this fraction the electric lighting is unaffected and the power multiplier is 1.0.

References

- CIE Technical Committee 4.2. 1973. Standardization of the Luminance Distribution on Clear Skies. CIE Pub. No. 22, Commission Internationale d'Eclairage, Paris.
- Hopkinson, R.G., J. Longmore and P. Petherbridge. 1954. An Empirical Formula for the Computation of the Indirect Component of Daylight Factors. Trans. Illum. Eng. Soc. (London) 19, 201.
- Hopkinson, R.G., P. Petherbridge and J. Longmore. 1966. Daylighting. Heinemann, London, p. 322.
- Hopkinson, R.G. 1970. Glare from Windows. Construction Research and Development Journal 2, 98.
- Hopkinson, R.G. 1972. Glare from Daylighting in Buildings. Applied Ergonomics 3, 206.
- Kittler, R. 1965. Standardization of Outdoor Conditions for the Calculation of the Daylight Factor with Clear Skies. Proc. CIE Inter-Session Meeting on Sunlight, Newcastle-Upon-Tyne.
- Lynes, J.A. 1968. Principles of Natural Lighting. Applied Science Publishers, Ltd., London, p. 129.
- Matsuura, K. 1987. Luminance Distributions of Various Reference Skies. CIE Technical Report of TC 3-09.
- Moon, P. and D. Spencer. 1942. Illumination from a Nonuniform Sky. Illuminating Engineering 37, 707-726.

Perez, R., P. Ineichen, R. Seals, J. Michalsky and R. Stewart. 1990. Modeling Daylight Availability and Irradiance Components from Direct and Global Irradiance. *Solar Energy* 44, 271-289.

Petherbridge, P. and J. Longmore. 1954. Solid Angles Applied to Visual Comfort Problems. *Light and Lighting* 47,173.

Winkelmann, F.C. 1983. Daylighting Calculation in DOE-2. Lawrence Berkeley Laboratory report no. LBL-11353, January 1983.

Winkelmann, F.C. and S. Selkowitz. 1985. Daylighting Simulation in the DOE-2 Building Energy Analysis Program. *Energy and Buildings* 8, 271-286.

DElight Daylighting Calculations

The Daylighting:DElight series of input objects provide an alternative daylighting model. The DElight method of analyzing daylighting applies to both simple apertures (i.e., windows and skylights) and complex fenestration systems that include geometrically complicated shading (e.g., roof monitors) and/or optically complicated glazings (e.g., prismatic or holographic glass). The DElight daylighting calculation methods are derived from the daylighting calculations in DOE-2.1E (as are the models accessed with Daylighting:Controls input object), and Superlite, with several key modifications. The engineering documentation included here focuses on the details of these differences from methods documented elsewhere. For the details of the heritage calculations, refer to the section in this documentation entitled "Daylighting Calculations" and to [Winkelmann, 1983], [Winkelmann and Selkowitz, 1985], and [Modest, 1982].

For each point in time, DElight calculates the interior daylighting illuminance at user specified reference points and then determines how much the electric lighting can be reduced while still achieving a combined daylighting and electric lighting illuminance target. The daylight illuminance level in a zone depends on many factors, including exterior light sources; location, size, and visible light transmittance of simple and complex fenestration systems; reflectance of interior surfaces; and location of reference points. The subsequent reduction of electric lighting depends on daylight illuminance level, design illuminance setpoint, fraction of zone controlled by reference point, and type of lighting control.

The DElight daylighting calculation has three main steps:

4. *Daylight Factor Calculation:* Daylight factors, which are ratios of interior illuminance to exterior horizontal illuminance, are pre-calculated and stored for later use. The user specifies the coordinates of one or more reference points in each daylit zone. DElight first calculates the contribution of light transmitted through all simple and complex fenestration systems in the zone to the illuminance at each reference point, and to the luminance at subdivided nodal patches of interior surfaces, for a given exterior luminous environment (including sky, sun, and exterior reflecting surfaces). The effect of inter-reflection of this initial light between interior reflecting surfaces is then calculated, resulting in a final total illuminance at each reference point. This total illuminance is then divided by the exterior horizontal illuminance for the given exterior environment to give a daylight factor. Daylight factors are calculated for each reference point, for a set of sun positions and sky conditions that are representative of the building location.
5. *Time-Step Interior Daylighting Calculation:* A daylighting calculation is performed for each heat-balance time step when the sun is up. In this calculation the illuminance at the reference points in each zone is found by interpolating the stored daylight factors using the current time step sun position and sky condition, then multiplying by the exterior horizontal illuminance.
6. *Electric Lighting Control Calculation:* The electric lighting control system is simulated to determine the proportion of lighting energy needed to make up the difference between the daylighting illuminance level at the given time step, and the design illuminance level.

Finally, the zone lighting electric reduction factor is passed to the thermal calculation, which uses this factor to reduce the heat gain from lights.

DElight Daylight Factor Calculation Differences from EnergyPlus Detailed Methods

- *Initial Interior Illuminance/Luminance Calculation:* DElight calculates the total initial contribution of light transmitted through all simple fenestration systems (i.e., windows and skylights) in the zone to the illuminance at each reference point, and to the luminance at each gridded nodal patch of interior surfaces. This differs from the models behind the “Daylighting:Controls” object (henceforth referred to as “EnergyPlus Detailed”) in two ways. The first is that EnergyPlus Detailed calculates initial illuminance values at reference points for each pair of reference point and aperture (window/skylight) in the zone, whereas DElight calculates the total contribution from all apertures to each reference point. The second difference from EnergyPlus Detailed is that the initial luminance of interior surface nodal patches is calculated to support the inter-reflection calculation described below. This calculation uses the same formula as EnergyPlus Detailed modified for arbitrarily oriented surfaces (i.e., non-horizontal), and to calculate luminance rather than illuminance. Note however, DElight does not account for interior surface obstructions (e.g., partitions) in this initial interior illuminance/luminance distribution. The EnergyPlus Detailed method does account for interior surface obstruction of initial illuminance distribution on reference points.
- *Reference Points:* DElight allows up to 100 reference points to be arbitrarily positioned within a daylighting zone. At this time all reference points are assumed to be oriented on a horizontal virtual surface “facing” toward the zenith and “seeing” the hemisphere above the horizontal plane.
- *Complex Fenestration System Calculation:* DElight calculates the contribution to the initial interior illuminance at each reference point, and to the luminance at each gridded nodal patch of interior surfaces, of the light transmitted by complex fenestration systems (CFS). The analysis of a CFS within DElight is based on the characterization of the system using bi-directional transmittance distribution functions (BTDF), which must be either pre-calculated (e.g., using ray-tracing techniques) or pre-measured, prior to analysis by DElight. A BTDF is a set of data for a given CFS, which gives the ratios of incident to transmitted light for a range of incoming and outgoing directions. As illustrated in Figure 60, a BTDF can be thought of as collapsing a CFS to a “black box” that is represented geometrically as a flat two-dimensional light-transmitting surface that will be treated as an aperture surface in the daylight zone description. For each incoming direction across the exterior hemisphere of the CFS, varying portions of that light are transmitted at multiple outgoing directions across the interior hemisphere of the CFS. The two-dimensional CFS “surface” and directional hemispheres are “abstract” in that they may not literally correspond to actual CFS component geometric details.

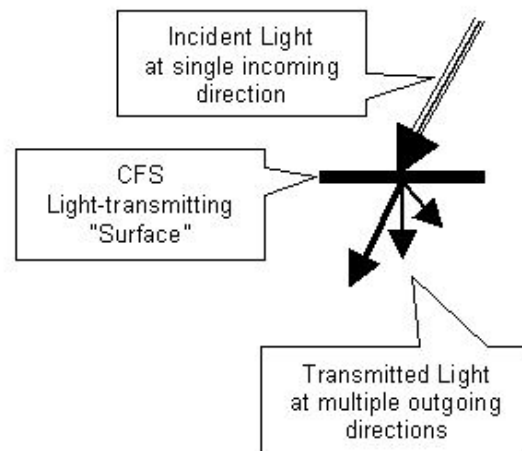


Figure 60. Bi-Directional Transmittance Data.

The pre-calculated or pre-measured BTDF for a CFS is independent of its final position and orientation within a building. Once a specific instance of a CFS aperture has been positioned within a building, the incident light from all exterior sources across the CFS exterior hemisphere can be integrated over all incident directions for each relevant transmitted direction to determine the light transmitted by the CFS surface in that direction. The light transmitted by the CFS aperture is then distributed to surfaces in the zone according to its non-uniform directionality. The algorithms for this BTDF treatment of CFS in DELight are still under development, and are subject to change in the future.

- *Inter-reflected Interior Illuminance/Luminance Calculation:* The effect of inter-reflection of the initial interior illuminance/luminance between interior reflecting surfaces is calculated using a radiosity method derived from Superlite [Modest, 1982]. This method subdivides each reflecting surface in the zone into nodal patches and uses view factors between all nodal patch pairs in an iterative calculation of the total contribution of reflected light within the zone. This method replaces the split-flux method used in EnergyPlus Detailed, resulting in a more accurate calculation of the varied distribution of inter-reflected light throughout the zone. The ability to input up to 100 reference points supports a more complete assessment of this distribution. Also, the radiosity method explicitly accounts for interior obstructions between pairs of nodal patches. The split-flux method used in the EnergyPlus Detailed approach only implicitly accounts for interior surfaces by including their reflectance and surface area in the zone average surface reflectance calculations.

DELight Time-Step Interior Daylighting Calculation Differences from EnergyPlus Detailed Methods

- *Interior Illuminance Calculation:* As discussed above, DELight only calculates daylight factors for the total contribution from all windows/skylights and CFS to each defined reference point. Thus DELight does not support dynamic control of fenestration shading during the subsequent time-step calculations, as does EnergyPlus Detailed.
- *Visual Quality:* DELight does not currently calculate a measure of visual quality such as glare due to daylighting. DELight does calculate luminance on nodal patches of all interior, reflecting surfaces. A variety of visual quality metrics could be calculated from these data in future implementations.
- *Electric Lighting Control Calculation:* Up to 100 reference points can be defined within a DELight daylighting zone. One or more of these reference points must be included in the control of the electric lighting system in the zone. Each reference point input includes the fraction of the zone controlled by that point. Values of 0.0 are valid, which allows the

definition of reference points for which interior illuminance levels are calculated, but which do not control the electric lighting. Any non-zero fraction is thus the equivalent of a relative weighting given to that reference point's influence on the overall electric lighting control. The sum of all fractions for defined reference points must less than or equal to 1.0 for this relative weighting to make physical sense. If the sum is less than 1.0, the remaining fraction is assumed to have no lighting control.

References

Modest, M. 1982. A General Model for the Calculation of Daylighting in Interior Spaces, Energy and Buildings 5, 66-79, and Lawrence Berkeley Laboratory report no. LBL-12599A.

Winkelmann, F.C. 1983. Daylighting Calculation in DOE-2. Lawrence Berkeley Laboratory report no. LBL-11353, January 1983.

Winkelmann, F.C. and S. Selkowitz. 1985. Daylighting Simulation in the DOE-2 Building Energy Analysis Program. Energy and Buildings 8, 271-286.

Daylighting Devices

Daylighting devices are used to improve daylighting in a zone. Besides their contribution to illuminance, daylighting devices also have a thermal impact on the zone heat balance. As a result the simulation of daylighting devices is tightly integrated into both the daylighting model and the zone heat balance.

There are two types of daylighting device in EnergyPlus: tubular daylighting devices and daylighting shelves.

Tubular Daylighting Devices

The input object DaylightingDevice:Tubular provides a special model for fenestration components known as Tubular Daylighting Devices (TDDs), also known as tubular skylights or light pipes. TDDs are constructed of three components: a dome, a pipe, and a diffuser.

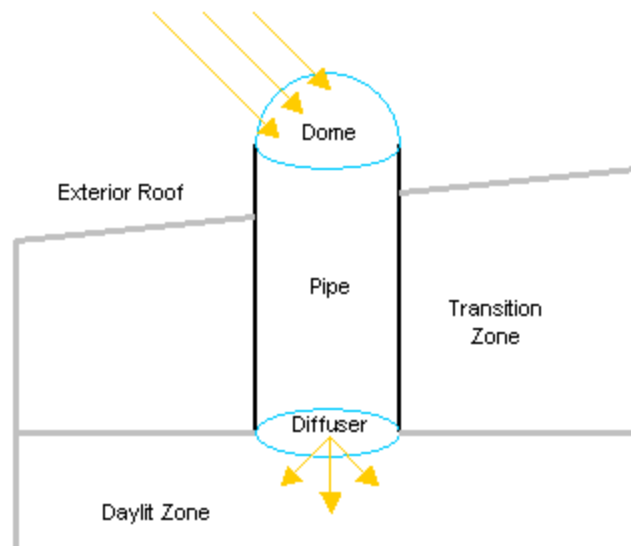


Figure 61. Tubular Daylighting Devices

The dome is typically a hemisphere made of clear plastic. It allows daylight into the pipe while keeping exterior weather out. The pipe is assumed to be a smooth cylinder with a highly reflective inside surface. The surface is usually either bare polished metal or a special

reflective sheet adhered to the inside. The pipe channels the daylight from the dome to the diffuser via multiple internal reflections. The diffuser is typically a flat frosted plastic cover. The diffuser evenly distributes the daylight to the zone.

In EnergyPlus the TDD model includes three different, but related, phenomena:

- Daylighting
- Solar gains
- Conductive/convective gains

Solar gains and conductive/convective gains are simulated by the zone heat balance. Daylighting is simulated independently.

For both daylighting and heat balance simulations, the dome and diffuser are treated as special window surfaces to take advantage of many of the standard daylighting and heat transfer routines. Together the dome and diffuser become "receiver" and "transmitter", i.e. radiation entering the dome ends up exiting the diffuser.

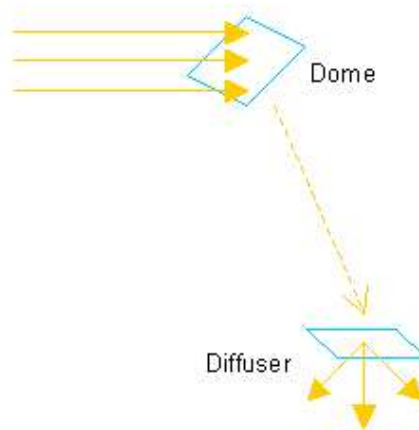


Figure 62. Dome and Diffuser Surfaces

The pipe is simulated by a separate code module. While several different measures for characterizing TDD performance are in use (Zhang 2002; Harrison 1998), the total transmittance of the TDD is most compatible with the EnergyPlus daylighting and heat balance code. Calculation of the transmittance of the pipe and the TDD for different types of radiation is fundamental to all phenomena covered by the model.

Pipe Beam Transmittance

The transmittance of beam radiation is derived from the integration of the transmittance of many discrete rays. The transmittance of a discrete ray through a pipe is dependent on the reflectivity of the inside pipe surface, the aspect ratio of the pipe, the incident angle of the ray, and the point of entry into the pipe.

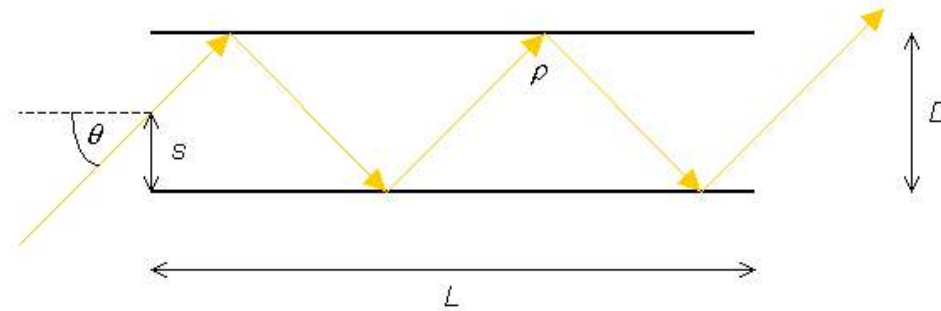


Figure 63. Discrete Ray in a Pipe

For an opaque surface, the reflectivity is:

$$\rho = 1 - \alpha$$

where α = surface absorptivity. Visible (i.e. daylighting) and solar absorptivities give visible and solar reflectivities, respectively. Measured reflectivities for commercial TDDs range from 0.90 to 0.99. Although the actual surface reflectivity is slightly dependent on the incident angle, the model assumes a constant reflectivity for all angles.

The full analytical expression for the transmittance of a beam of light in a highly reflective pipe has been developed by Swift and Smith and verified by experiment (1994). By integrating over all rays incident on the pipe entrance, they find the transmittance of a beam of collimated radiation to be:

$$\tau = \frac{4}{\pi} \int_{s=0}^1 \frac{s^2}{\sqrt{1-s^2}} \rho^{INT[a \tan \theta / s]} (1 - (1 - \rho)(a \tan \theta / s - INT[a \tan \theta / s])) ds$$

where

$a = L/D$, the aspect ratio of the TDD

ρ = surface reflectivity

θ = incident angle

s = entry point

This integral does not have an analytical solution and must be calculated numerically. It was found that a large number of points (100,000) were necessary to achieve an acceptable accuracy. Since the integration is time consuming and the transmittance of the pipe must be utilized many times at every time step, values are calculated over a range of incident angles and stored in a table. The tabulated values are interpolated to rapidly give the transmittance at any incident angle. A polynomial fit was also considered but it was found that interpolation gave superior results.

In the graph below, interpolated values from EnergyPlus are compared to the results of ray tracing simulations performed at the Florida Solar Energy Center for an incident angle of 30 degrees (McCluney 2003).

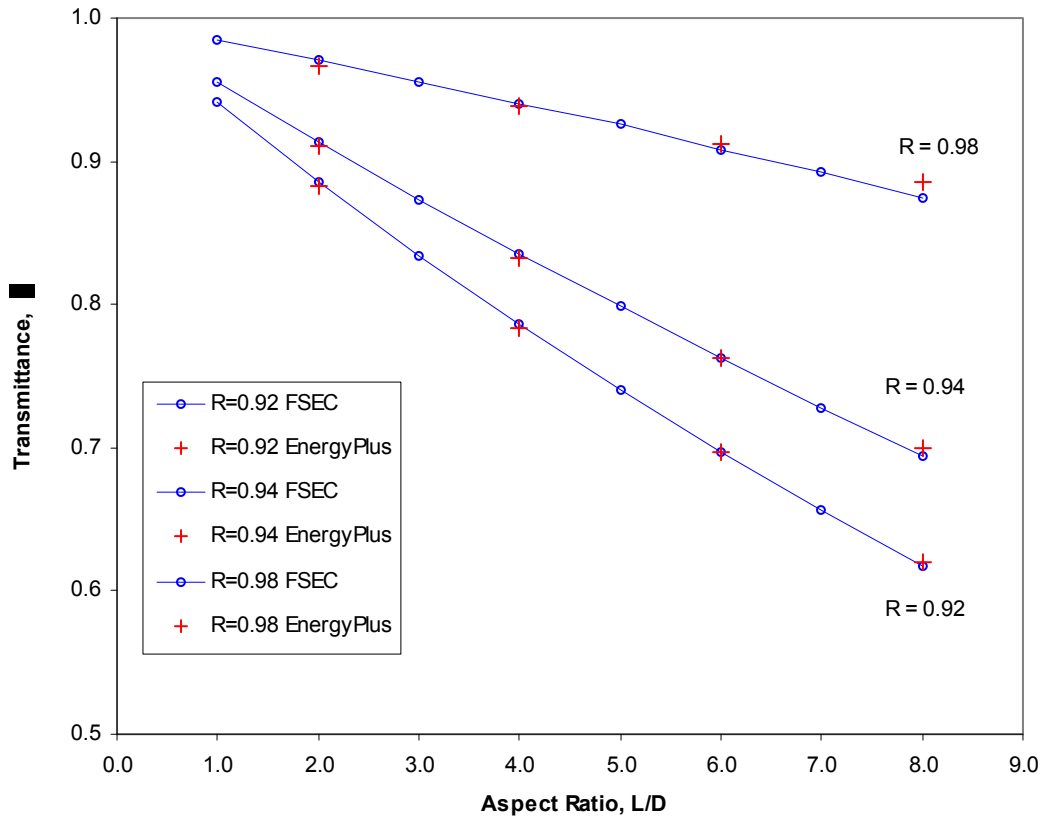


Figure 64. Pipe Transmittance Comparison.

During initialization of each unique TDD, the program integrates and tabulates values for the visible and solar transmittance of the pipe. The results are subsequently used in the daylighting simulation and heat balance simulation respectively.

The effect of bends in the pipe on beam transmittance is not included in this model. Recent research (Zhang 2002) has suggested that a 30 degree bend has a 20% loss in transmitted light. If the effect of bends must be simulated, it can be approximated by the user by appropriately decreasing the transmittance of the diffuser material.

TDD Beam Transmittance

The beam transmittance of the TDD takes into account the dome and diffuser transmittances in addition to the pipe transmittance.

$$\tau_{TDD}(\theta) = \tau_{dome}(\theta)\tau_{pipe}(\theta)\tau_{diffuser}$$

where

$\tau_{dome}(\theta)$ = beam transmittance of the dome glazing at the incident angle

$\tau_{pipe}(\theta)$ = beam transmittance of the pipe at the incident angle, as described above

$\tau_{diffuser}$ = diffuse transmittance of the diffuser glazing

The dome transmittance is calculated for a flat window. The model does not take into account refraction due to the curvature of the dome surface.

Diffuse transmittance is always assumed for the diffuser because multiple internal reflections in the pipe scatter the beam with a diffusing effect. Although the light exiting the pipe is not isotropic, it can be approximated as diffuse. The use of a frosted diffuser on the TDD, however, ensures that the light delivered to the zone is very close to isotropic diffuse.

The calculation of TDD diffuse transmittance is considerably more complex and is handled differently in the daylighting simulation and the heat balance simulation. The details are discussed in the following sections.

Daylighting

The daylighting simulation of the TDD treats the diffuser surface as a regular window illuminated from the outside by sun, sky, and ground. However, the TDD model replaces the window glazing transmittance with the appropriate TDD transmittance and converts all transmitted light to diffuse.

The illuminance due to the direct beam of the sun is found using the TDD beam transmittance $\tau_{TDD}(\theta)$ as described above. The incident angle θ is relative to the dome surface.

The illuminance due to sky radiation and ground reflected radiation is calculated with the normal daylighting model integration over the sky and ground within the viewable hemisphere of the dome. The transmittance of each sky or ground element is also found using the TDD beam transmittance at the incident angle of the sky or ground element relative to the dome.

Light from the diffuser is converted to diffuse inside the zone in the same way as an interior shade.

Solar Gains

Solar radiation incident on a window is calculated separately as sun, sky, and ground radiation. A different transmittance must be applied for each type of radiation.

For beam radiation the TDD beam transmittance $\tau_{TDD}(\theta)$ for the solar spectrum is used as described above. For sky and ground radiation a diffuse transmittance for the TDD must be developed.

The transmittance of diffuse radiation can be defined as the total transmitted flux divided by the total incident flux.

$$\tau_{diff} = \frac{\sum I_{trans}}{\sum I_{inc}}$$

Swift and Smith (1994) suggest a weighted integral of the beam transmittance over the hemisphere for an arbitrary angular distribution:

$$\tau_{diff} = \frac{\int_{\theta=0}^{\pi/2} \tau(\theta)P(\theta)\sin\theta d\theta}{\int_{\theta=0}^{\pi/2} P(\theta)\sin\theta d\theta}$$

where

$P(\theta)$ = angular distribution function

For isotropic diffuse radiation $P(\theta)$ is the cosine of the incident angle θ .

$$\tau_{diff,iso} = \frac{\int_{\theta=0}^{\pi/2} \tau(\theta)\cos\theta\sin\theta d\theta}{\int_{\theta=0}^{\pi/2} \cos\theta\sin\theta d\theta}$$

For a given pipe or TDD, $\tau_{diff,iso}$ is a constant. The program calculates $\tau_{diff,iso}$ once during initialization using a numerical integration.

The diffuse isotropic transmittance is useful, but not sufficient, for determining the transmittance of sky radiation. As described in the *Sky Radiance Model* section, sky radiation has an anisotropic distribution modeled as the superposition of three simple distributions: a diffuse isotropic background, a circumsolar brightening near the sun, and a horizon brightening. While the daylighting model is capable of calculating the luminance of any position in the sky, the solar code only calculates the ultimate irradiance on a surface. For this reason it is not possible to integrate over an angular distribution function for sky radiance. Instead the three sky distributions must be handled piecewise.

$$\tau_{diff,aniso} = \frac{\sum I_{trans,aniso}}{\sum I_{inc,aniso}} = \frac{I_{trans,iso} + I_{trans,circumsolar} + I_{trans,horiz}}{I_{inc,iso} + I_{inc,circumsolar} + I_{inc,horiz}}$$

Substituting in the appropriate transmittances:

$$\tau_{diff,aniso} = \frac{\tau_{diff,iso} I_{inc,iso} + \tau(\theta) I_{inc,circumsolar} + \tau_{diff,horiz} I_{inc,horiz}}{I_{inc,iso} + I_{inc,circumsolar} + I_{inc,horiz}}$$

where

$\tau_{diff,iso}$ = diffuse isotropic transmittance

$\tau(\theta)$ = beam transmittance at incident angle θ of sun

$\tau_{diff,horiz}$ = diffuse transmittance of the horizon, derived below

It is important to note that transmittances above are for the total TDD. The transmittance of the dome and diffuser must be included to account for their angular dependences as well.

The beam transmittance is used as an approximation for all circumsolar radiation.

The diffuse horizon transmittance is found by integrating the beam transmittance over the arc of the horizon.

$$\tau_{diff,horiz} = \frac{\sum I_{trans,horiz}}{\sum I_{inc,horiz}} = \frac{\int_{\theta=-\pi/2}^{\pi/2} \tau(\theta) \cos \theta d\theta}{\int_{\theta=-\pi/2}^{\pi/2} \cos \theta d\theta}$$

Since the radiance of the horizon is isotropic, and therefore constant across the entire horizon, the actual value of the radiance cancels out. The result is a constant that is calculated once during initialization.

Ground radiation is assumed to be isotropic diffuse. The transmittance of ground radiation is the diffuse isotropic transmittance.

$$\tau_{diff,gnd} = \tau_{diff,iso}$$

The solar flux transmitted by a TDD due to beam, sky, and ground radiation is calculated as normal for a window but uses the respective transmittances for the TDD.

$$q''_{TDD-trans,beam} = (I_{sun} \cos \theta) f_{sunlit} \tau_{TDD}(\theta)$$

$$q''_{TDD-trans,sky} = I_{h,sky} f_{skymult} \tau_{TDD,diff,aniso}$$

$$q''_{TDD-trans,wnd} = (I_{sun} \cos \theta + I_{h,sky}) F_{sg} \tau_{TDD,diff,iso}$$

where

I_{sun} = solar beam intensity of the sun

$I_{h,sky}$ = total horizontal diffuse solar radiation due to the sky

θ = incident angle of the beam on the dome

f_{sunlit} = sunlit beam fraction of the dome area

$f_{skymult}$ = anisotropic sky view multiplier (see AnisoSkyMult)

F_{sg} = view from ground to dome

$\tau_{TDD}(\theta)$ = TDD beam transmittance

$\tau_{TDD,diff,aniso}$ = TDD anisotropic sky transmittance

$\tau_{TDD,diff,iso}$ = TDD isotropic diffuse transmittance

Daylighting vs. Solar

The beam transmittance of a TDD is calculated in the same way for both daylighting and solar gains. If the visible and solar properties (i.e. absorptances in the input file) are the same, the reported TDD beam transmittances are equal.

However, because the daylighting and solar models treat diffuse radiation differently, the TDD diffuse transmittances reported for visible and solar radiation will not necessarily be equal, even though the properties may be the same.

Since the daylighting model calculates the diffuse illuminance using a sky and ground integration of many discrete elements, a visible diffuse transmittance is not required for the TDD daylighting simulation. For reporting purposes only, the visible diffuse transmittance is estimated concurrent with the sky and ground integration using:

$$\tau_{diff} = \frac{\int \tau_{TDD}(\theta) d\Phi_{inc}}{\int d\Phi_{inc}}$$

Conductive/Convective Gains

For conductive and convective heat gain, TDDs are treated as one entity with an effective thermal resistance (i.e. R-value) between the outside and inside surfaces. The outside face temperature of the dome and the inside face temperature of the diffuser are calculated as usual by the outside and inside heat balances respectively. The temperatures are then copied to the inside face of the dome and the outside face of the diffuser. Normal exterior and interior convection and IR radiation exchange occurs for both surfaces.

Although little research has been done on the thermal characteristics of TDDs, one experiment (Harrison 1998) reports an average effective thermal resistance of 0.279 m² K/W for a commercial TDD measuring 0.33 m in diameter by 1.83 m in length. This value, however, reflects a measurement from outdoor air temperature to indoor air temperature. The model assumes an effective thermal resistance from outside surface temperature to inside surface temperature.

Solar radiation is inevitably absorbed by the TDD before it reaches the zone. Every reflection in the pipe leaves behind some solar radiation according to the surface absorptance. Rays incident at a greater angle make more reflections and leave behind more absorbed solar in the pipe wall.

The total absorbed solar radiation in the TDD is the sum of the following gains:

- Inward bound solar absorbed by multiple pipe reflections
- Outward bound solar absorbed by multiple pipe reflections due to:
 - Reflection off of diffuser surface (inside of TDD)
 - Zone diffuse interior shortwave incident on the diffuser from lights, etc.
- Inward flowing absorbed solar in dome and diffuser glazing

$$Q_{abs,pipe} = Q_{abs,in} + Q_{abs,out} + Q_{abs,glazing}$$

The inward bound solar absorbed by the pipe is the difference between the solar transmitted by the dome and the solar incident on the diffuser.

$$Q_{abs,in} = q''_{trans,dome} A_{dome} - q''_{inc,diffuser} A_{diffuser}$$

The solar transmitted by the dome $q''_{trans,dome}$ is calculated as usual for a window. The solar incident on the diffuser $q''_{inc,diffuser}$ is more complicated because each component must be treated separately.

$$q''_{inc,diffuser} = q''_{beam} \frac{\tau_{TDD,beam}(\theta)}{\tau_{diffuser}} + q''_{sky} \frac{\tau_{TDD,aniso}(Hour)}{\tau_{diffuser}} + q''_{gnd} \frac{\tau_{TDD,iso}}{\tau_{diffuser}}$$

The outward bound solar absorbed by the pipe is given by:

$$Q_{abs,out} = \left(q''_{refl,diffuser} \frac{(1 - \tau_{TDD})}{\tau_{diffuser}} + q''_{zoneSW} (1 - \tau_{TDD}) \right) A_{diffuser}$$

where q''_{zoneSW} is the zone interior diffuse shortwave flux from window, lights, and ambient surface reflections, and

$$q''_{refl,diffuser} = q''_{inc,diffuser} - q''_{abs,diffuser} - q''_{trans,diffuser}$$

The inward flowing portion of solar absorbed in the dome and diffuser glazing is:

$$Q_{abs,glazing} = \frac{q''_{abs,dome} A_{dome}}{2} + \frac{q''_{abs,diffuser} A_{diffuser}}{2}$$

All absorbed solar radiation in the TDD is distributed among the transition zones that the pipe passes through between dome and diffuser. The transition zone heat gain is proportional to the length of the zone. Any exterior length of pipe also receives a proportional amount of heat, but this is lost to the outside.

References

Harrison, S. J., McCurdy, G. G., Cooke, R. 1998. "Preliminary Evaluation of the Daylighting and Thermal Performance of Cylindrical Skylights", Proceedings of International Daylight Conference, Ottawa, Canada, pp. 205-212.

McCluney, R. 2003. "Rating of Tubular Daylighting Devices for Visible Transmittance and Solar Heat Gain -- Final Report", FSEC-CR-1385-03, January 15, 2003, Florida Solar Energy Center, 1679 Clearlake Rd., Cocoa, FL 32922.

Swift, P. D., and Smith, G. B. 1995. "Cylindrical Mirror Light Pipes", Solar Energy Materials and Solar Cells 36, pp. 159-168.

Zhang, X., Muneer, T., and Kubie, J. 2002. "A Design Guide For Performance Assessment of Solar Light-Pipes", Lighting Research & Technology 34, 2, pp. 149-169.

Daylighting Shelves

The input object DaylightingDevice:Shelf provides a special model for light shelves used to augment daylighting. Light shelves are constructed from up to three components: a window, an inside shelf, and an outside shelf. The inside shelf acts to reflect all transmitted light from the upper window onto the ceiling of the zone as diffuse light. The outside shelf changes the amount of light incident on the window. All light reflected from the outside shelf also goes onto the zone ceiling. The inside shelf and outside shelf are both optional. However, if neither shelf is specified, the daylighting shelf object has no effect on the simulation.

The window is divided into two window surfaces: an upper window and a lower window. The upper window interacts with the daylighting shelf but the lower window does not, except to receive shading from the outside shelf.

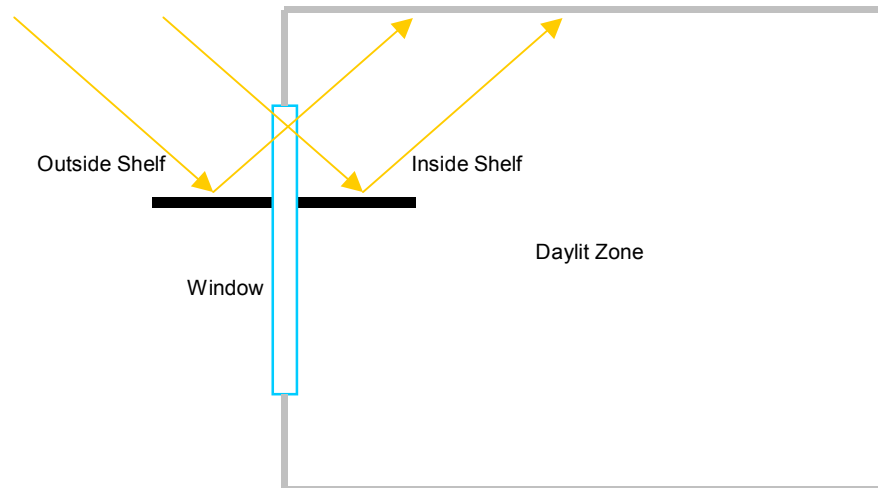


Figure 65. Daylighting Shelf Diagram

Daylighting shelves are simulated separately for daylighting and the zone heat balance. The general model is similar in both cases, but the details vary.

Inside Shelf Daylighting

The inside shelf is modeled in the daylighting simulation by converting all light transmitted by the upper window into diffuse upgoing flux. It is assumed that no beam or downgoing flux can pass the end of the shelf regardless of the shelf's position or orientation.

In the daylighting simulation this is accomplished by forcing all the transmitted flux to be upgoing:

$$\Phi_{CW} = \Phi$$

$$\Phi_{FW} = 0$$

where

Φ_{CW} = upgoing flux

Φ_{FW} = downgoing flux

Φ = total flux

Since it is assumed that all light falls on the inside shelf, it is implied that the upper window cannot contribute any direct illuminance (i.e. the upper window cannot be seen from anywhere in the zone). The remaining light is entirely interreflected sky-related and interreflected sun-related upgoing flux.

Inside Shelf Heat Balance

In the heat balance simulation the inside shelf is defined as an interzone heat transfer surface, e.g. partition. Since the inside shelf external boundary condition is required to refer to itself, the shelf is essentially equivalent to internal mass. Because the shelf surface has two sides that participate in the zone heat balance, the surface area is doubled by the program during initialization. Like internal mass, the shelf surface is allowed to interact convectively and radiatively with the zone air and other zone surfaces.

The zone interior solar distribution is modified by the inside shelf. Regardless of the solar distribution selected in the input file, all beam solar radiation transmitted by the upper window is incident on one side (half the doubled surface area) of the shelf surface. The beam radiation not absorbed is reflected throughout the zone as diffuse shortwave radiation. The treatment of sky and ground radiation is unchanged; both are added directly to the zone diffuse shortwave.

The total beam, sky, and ground radiation transmitted by the upper window does not change.

Outside Shelf Daylighting

In the daylighting model the luminous flux transmitted by the upper window is determined by integrating over the sky and ground and summing the luminance contribution of each sky or ground element. The luminance of any intervening exterior or interior surfaces is assumed to be zero. As a shading surface, the effect of the outside shelf during the integration is to block part of the view of the ground, thereby reducing the window transmitted flux due to diffuse ground luminance. After the integration is complete, the program calculates the amount of diffuse light that is reflected through the window from the outside shelf and adds it as a lump sum to the upgoing flux transmitted by the window.

The additional shelf upgoing flux is the sum of sun-related and sky-related flux:

$$\Phi_{shelf,CW} = \Phi_{shelf,sun} + \Phi_{shelf,sky}$$

where

$$\Phi_{shelf,sun} = (E_{sun} \cos \theta) f_{sunlit} \rho_{vis} F_{ws} \tau_{diff,vis}$$

$$\Phi_{shelf,sky} = E_{h,sky} f_{skymult} \rho_{vis} F_{ws} \tau_{diff,vis}$$

and

E_{sun} = exterior illuminance due to light from the sun

$E_{h,sky}$ = exterior horizontal illuminance due to light from the sky

θ = incident angle of the beam on the shelf

f_{sunlit} = sunlit beam fraction of the shelf surface area

$f_{skymult}$ = anisotropic sky view multiplier (see AnisoSkyMult)

ρ_{vis} = shelf surface reflectivity in the visible spectrum

F_{ws} = view factor from window to shelf

$\tau_{diff,vis}$ = diffuse window transmittance in the visible spectrum

The sunlit beam fraction f_{sunlit} and the anisotropic sky view multiplier $f_{skymult}$ are borrowed from the heat balance solar calculations.

The sunlit beam fraction f_{sunlit} takes into account the effect of shading due to other surfaces. Although shadows on the shelf surface change the luminance distribution of the shelf, there is no angular dependence because diffuse properties are assumed. Therefore, the flux is simply proportional to the sunlit fraction.

The anisotropic sky view multiplier f_{skymult} takes into account the anisotropic distribution of sky radiation, the shelf view factor to the sky, and shading. This value is utilized in the heat balance simulation for solar calculations but is not currently available in the daylighting simulation. A value of 1.0 is assumed until a better model is developed. For this reason the sky-related flux may be over-predicted for some building and shelf geometries. However, for clear sky conditions the sun-related flux is dominant and the resulting error is small.

The view factor to the outside shelf, F_{ws} , if not specified by the user in the input object, is an exact view factor calculated for adjacent perpendicular rectangles.

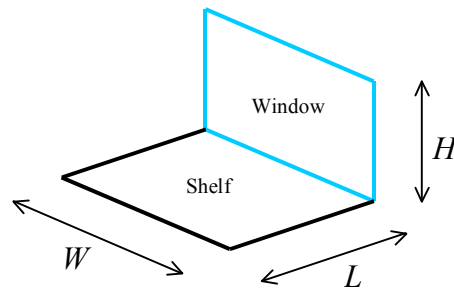


Figure 66. Window and Outside Shelf as Adjacent Perpendicular Rectangles.

For this geometry the view factor is given by (Mills 1995):

$$F_{\text{ws}} = \frac{1}{\pi M} \left\{ M \tan^{-1} \left(\frac{1}{M} \right) + N \tan^{-1} \left(\frac{1}{N} \right) - (M^2 + N^2)^{1/2} \tan^{-1} \left((M^2 + N^2)^{-1/2} \right) \right. \\ \left. + \frac{1}{4} \ln \left[\left(\frac{(1+M^2)(1+N^2)}{1+M^2+N^2} \right) \left(\frac{M^2(1+M^2+N^2)}{(1+M^2)(M^2+N^2)} \right)^{M^2} \left(\frac{N^2(1+M^2+N^2)}{(1+N^2)(M^2+N^2)} \right)^{N^2} \right] \right\}$$

where

$$M = H / W$$

$$N = L / W$$

Outside Shelf Heat Balance

The heat balance simulation does not do a sky and ground integration. View factors to sky and ground are used instead. Consequently, the heat balance calculation for the outside shelf is very similar to the daylighting calculation. The main difference is that the incident flux on the upper window is calculated first and reported. The transmitted and absorbed fractions are subsequently determined.

The solar flux incident on the upper window due to the shelf is given by:

$$q''_{\text{shelf-inc}} = q''_{\text{shelf-inc,sun}} + q''_{\text{shelf-inc,sky}}$$

where

$$q''_{shelf-inc,sun} = (I_{sun} \cos \theta) f_{sunlit} \rho_{sol} F_{ws}$$

$$q''_{shelf-inc,sky} = I_{h,sky} f_{skymult} \rho_{sol} F_{ws}$$

and

I_{sun} = solar beam intensity of the sun

$I_{h,sky}$ = total horizontal diffuse solar radiation due to the sky

θ = incident angle of the beam on the shelf

f_{sunlit} = sunlit beam fraction of the surface area

$f_{skymult}$ = anisotropic sky view multiplier (see AnisoSkyMult)

ρ_{sol} = shelf surface reflectivity in the solar spectrum

F_{ws} = view factor from window to shelf

The view factor F_{ws} is the same as described above for daylighting.

The total diffuse incident radiation due to the shelf is internally added to the ground diffuse incident radiation on the window. For reporting purposes the shelf radiation is included in the *Surface Ext Solar Ground Diffuse Incident* output variable.

With the incident radiation determined, the remaining window heat balance is calculated normally. The resulting transmitted diffuse radiation from the sky, ground, and shelf is:

$$q''_{trans} = (q''_{sky-inc} + q''_{gnd-inc} + q''_{shelf-inc}) \tau_{diff,sol}$$

where

$\tau_{diff,sol}$ = diffuse window transmittance in the solar spectrum

References

Mills, A. F. 1995. Heat and Mass Transfer, p. 499.

Window Calculation Module

This section describes two potential modeling approaches for Windows. The first (layer by layer) is implemented. The second, simple approach, is in development.

The primary Window calculation is a layer-by-layer approach where windows are considered to be composed of the following components, only the first of which, glazing, is required to be present:

- **Glazing**, which consists of one or more plane/parallel glass layers. If there are two or more glass layers, the layers are separated by gaps filled with air or another gas. The glazing optical and thermal calculations are based on algorithms from the WINDOW 4 and WINDOW 5 programs [Arasteh et al., 1989], [Finlayson et al., 1993]. Glazing layers are described using the input object WindowMaterial:Glazing.
- **Gap**, layers filled with air or another gas that separate glazing layers. Gaps are described using the input object WindowMaterial:Gas.
- **Frame**, which surrounds the glazing on four sides. Frames are described using the input object WindowProperty:FrameAndDivider.
- **Divider**, which consists of horizontal and/or vertical elements that divide the glazing into individual lites.
- **Shading device**, which is a separate layer, such as drapery, roller shade or blind, on the inside or outside of the glazing, whose purpose is to reduce solar gain, reduce heat loss

(movable insulation) or control daylight glare. Shading layers are described using "WindowProperty:ShadingControl" input objects.

In the following, the description of the layer-by-layer glazing algorithms is based on material from Finlayson et al., 1993. The frame and divider thermal model, and the shading device optical and thermal models, are new to EnergyPlus.

A second approach is being developed where windows are modeled in a simplified approach that requires minimal user input that is processed to develop an equivalent layer that then reuses much of the layer-by-model. This "Simple Window Construction" model is described below.

Optical Properties of Glazing

The solar radiation transmitted by a system of glass layers and the solar radiation absorbed in each layer depends on the solar transmittance, reflectance and absorptance properties of the individual layers. The absorbed solar radiation enters the glazing heat balance calculation that determines the inside surface temperature and, therefore, the heat gain to the zone from the glazing (see "Window Heat Balance Calculation"). The transmitted solar radiation is absorbed by interior zone surfaces and, therefore, contributes to the zone heat balance. In addition, the visible transmittance of the glazing is an important factor in the calculation of interior daylight illuminance from the glazing.

Table 24. Variables in Window Calculations

Mathematical variable	Description	Units	FORTTRAN variable
T	Transmittance	-	-
R	Reflectance	-	-
R^f, R^b	Front reflectance, back reflectance	-	-
$T_{i,j}$	Transmittance through glass layers i to j	-	-
T_{gl}^{dir}	Direct transmittance of glazing	-	-
$R_{i,j}^f, R_{i,j}^b$	Front reflectance, back reflectance from glass layers i to j	-	-
$R_{gl,f}^{dir}, R_{gl,b}^{dir}$	Direct front and back reflectance of glazing	-	-
A_i^f, A_i^b	Front absorptance, back absorptance of layer i	-	-
N	Number of glass layers	-	Nlayer
λ	Wavelength	microns	Wle
$E_s(\lambda)$	Solar spectral irradiance function	W/m ² -micron	E
V(λ)	Photopic response function of the eye	-	y30
ϕ	Angle of incidence (angle between surface normal and direction of incident beam radiation)	Rad	Phi

τ	Transmittivity or transmittance	-	tf0
ρ	Reflectivity or reflectance	-	rf0, rb0
α	Spectral absorption coefficient	m^{-1}	-
d	Glass thickness	M	Material%Thickness
n	Index of refraction	-	ngf, ngb
κ	Extinction coefficient	-	-
β	Intermediate variable	-	betaf, betab
P, p	A general property, such as transmittance	-	-
τ_{sh}	Shade transmittance	-	Material%Trans
ρ_{sh}	Shade reflectance	-	Material%ReflectShade
α_{sh}	Shade absorptance	-	Material%AbsorpSolar
$\tau_{bl}, \rho_{bl}, \alpha_{bl}$	Blind transmittance, reflectance, absorptance	-	-
Q, G, J	Source, irradiance and radiosity for blind optical properties calculation	W/m^2	-
F_{ij}	View factor between segments i and j	-	-
f_{switch}	Switching factor	-	SwitchFac
T	Transmittance	-	-
R	Reflectance	-	-
R^f, R^b	Front reflectance, back reflectance	-	-
$T_{i,j}$	Transmittance through glass layers i to j	-	-
$R_{i,j}^f, R_{i,j}^b$	Front reflectance, back reflectance from glass layers i to j	-	-
A_i^f, A_i^b	Front absorptance, back absorptance of layer i	-	-
N	Number of glass layers	-	Nlayer
λ	Wavelength	microns	Wle
$E_s(\lambda)$	Solar spectral irradiance function	W/m^2 -micron	E
$R(\lambda)$	Photopic response function of the eye	-	y30
φ'	Relative azimuth angle (angle between screen surface normal and vertical plane through sun, Ref. Figure 77)	Rad	SunAzimuthToScreenNormal

α'	Relative altitude angle (angle between screen surface horizontal normal plane and direction of incident beam radiation, Ref. Figure 77)	Rad	SunAltitudeToScreenNormal
ρ_{sc}	Beam-to-diffuse solar reflectance of screen material	-	Screens%ReflectCylinder
γ	Screen material aspect ratio	-	Screens%ScreenDiameterToSpacingRatio
A	Spectral absorption coefficient	m^{-1}	-
D	Glass thickness	M	Material%Thickness
N	Index of refraction	-	ngf, ngb
K	Extinction coefficient	-	-
B	Intermediate variable	-	betaf, betab
P, p	A general property, such as transmittance	-	-

Glass Layer Properties

In EnergyPlus, the optical properties of individual glass layers are given by the following quantities at normal incidence as a function of wavelength:

Transmittance, T

Front reflectance, R^f

Back reflectance, R^b

Here “front” refers to radiation incident on the side of the glass closest to the outside environment, and “back” refers to radiant incident on the side of the glass closest to the inside environment. For glazing in exterior walls, “front” is therefore the side closest to the outside air and “back” is the side closest to the zone air. For glazing in interior (i.e., interzone) walls, “back” is the side closest to the zone in which the wall is defined in and “front” is the side closest to the adjacent zone.

Glass Optical Properties Conversion

Conversion from Glass Optical Properties Specified as Index of Refraction and Transmittance at Normal Incidence

The optical properties of uncoated glass are sometimes specified by index of refraction, n , and transmittance at normal incidence, T .

The following equations show how to convert from this set of values to the transmittance and reflectance values required by Material:WindowGlass. These equations apply only to uncoated glass, and can be used to convert either spectral-average solar properties or spectral-average visible properties (in general, n and T are different for the solar and visible). Note that since the glass is uncoated, the front and back reflectances are the same and equal to the R that is solved for in the following equations.

Given n and T , find R :

$$r = \left(\frac{n-1}{n+1} \right)^2$$

$$\tau = \frac{\left[(1-r)^4 + 4r^2 T^2 \right]^{1/2} - (1-r)^2}{2r^2 T}$$

$$R = r + \frac{(1-r)^2 r \tau^2}{1-r^2 \tau^2}$$

Example:

$$T = 0.86156$$

$$n = 1.526$$

$$r = \left(\frac{1.526-1}{1.526+1} \right)^2$$

$$\tau = 0.93974$$

$$R = 0.07846$$

Simple Window Model

EnergyPlus includes an alternate model that allows users to enter in simplified window performance indices. This model is accessed through the WindowMaterial:SimpleGlazingSystem input object and converts the simple indices into an equivalent single layer window. (In addition a special model is used to determine the angular properties of the system – described below). Once the model generates the properties for the layer, the program reuses the bulk of the layer-by-layer model for subsequent calculations. The properties of the equivalent layer are determined using the step by step method outlined by Arasteh, Kohler, and Griffith (2009). The core equations are documented here. The reference contains additional information.

The simplified window model accepts U and SHGC indices and is useful for several reasons:

- Sometimes, the only thing that is known about the window are its U and SHGC;
- Codes, standards, and voluntary programs are developed in these terms;
- A single-layer calculation is faster than multi-layer calculations.

Note: This use of U and SHGC to describe the thermal properties of windows is only appropriate for specular glazings.

While it is important to include the ability to model windows with only U-value and SHGC, we note that any method to use U and SHGC alone in building simulation software will inherently be approximate. This is due primarily to the following factors:

- SHGC combines directly transmitted solar radiation and radiation absorbed by the glass which flows inward. These have different implications for space heating/cooling. Different windows with the same SHGC often have different ratios of transmitted to absorbed solar radiation.
- SHGC is determined at normal incidence; angular properties of glazings vary with number of layers, tints, coatings. So products which have the same SHGC, can have different angular properties.
- Window U-factors vary with temperatures.

Thus, for modeling specific windows, we recommend using more detailed data than just the U and SHGC, if at all possible.

The simplified window model determines the properties of an equivalent layer in the following steps.

Step 1. Determine glass-to-glass Resistance.

Window U-values include interior and exterior surface heat transfer coefficients. The resistance of the bare window product, or glass-to-glass resistance is augmented by these film coefficients so that,

$$\frac{1}{U} = R_{i,w} + R_{o,w} + R_{l,w}$$

Where,

$R_{i,w}$ is the resistance of the interior film coefficient under standard winter conditions in units of $\text{m}^2\cdot\text{K}/\text{W}$,

$R_{o,w}$ is the resistance of the exterior film coefficient under standard winter conditions in units of $\text{m}^2\cdot\text{K}/\text{W}$, and

$R_{l,w}$ is the resistance of the bare window under winter conditions (without the film coefficients) in units of $\text{m}^2\cdot\text{K}/\text{W}$.

The values for $R_{i,w}$ and $R_{o,w}$ depend on U and are calculated using the following correlations.

$$R_{i,w} = \frac{1}{(0.359073 \ln(U) + 6.949915)}; \quad \text{for } U < 5.85$$

$$R_{i,w} = \frac{1}{(1.788041U - 2.886625)}; \quad \text{for } U \geq 5.85$$

$$R_{o,w} = \frac{1}{(0.025342U + 29.163853)}$$

So that the glass-to-glass resistance is calculated using,

$$R_{l,w} = \frac{1}{U} - R_{i,w} - R_{o,w}.$$

Step 2. Determine Layer Thickness.

The thickness of the equivalent layer in units of meters is calculated using,

$$\text{Thickness} = 0.002; \quad \text{for } R_{l,w} > 7.0$$

$$\text{Thickness} = 0.05914 - \left(\frac{0.00714}{R_{l,w}} \right); \quad R_{l,w} \leq 7.0$$

Step 3. Determine Layer Thermal Conductivity

The effective thermal conductivity, λ_{eff} , of the equivalent layer is calculated using,

$$\lambda_{eff} = \frac{\text{Thickness}}{R_{l,w}}$$

Step 4. Determine Layer Solar Transmittance

The layer's solar transmittance at normal incidence, T_{sol} , is calculated using correlations that are function of SHGC and U-Factor.

$$T_{sol} = 0.920229 SHGC^2 + 0.20199 SHGC; \quad U > 4.5; SHGC \leq 0.75$$

$$T_{sol} = 1.487661 SHGC - 0.439392; \quad U > 4.5; SHGC > 0.75$$

$$T_{sol} = 0.41040 SHGC; \quad U < 3.4; SHGC \leq 0.15$$

$$T_{sol} = 0.085775 SHGC^2 + 0.963954 SHGC - 0.084958; \quad U < 3.4; SHGC > 0.15$$

And for U-values between 3.4 and 4.5, the value for T_{sol} is interpolated using results of the equations for both ranges.

Step 5. Determine Layer Solar Reflectance

The layer's solar reflectance is calculated by first determining the inward flowing fraction which requires values for the resistance of the inside and outside film coefficients under summer conditions, $R_{i,s}$ and $R_{o,s}$ respectively. The correlations are

$$R_{i,s} = \frac{1}{\left(29.436546 (SHGC - T_{Sol})^3 - 21.943415 (SHGC - T_{Sol})^2 + 9.945872 (SHGC - T_{Sol}) + 7.426151\right)}; U > 3.6$$

$$R_{i,s} = \frac{1}{\left(199.8208128 (SHGC - T_{Sol})^3 - 90.639733 (SHGC - T_{Sol})^2 + 19.737055 (SHGC - T_{Sol}) + 6.766575\right)}; U < 3.4$$

$$R_{o,s} = \frac{1}{\left(2.225824 (SHGC - T_{Sol}) + 20.57708\right)}; U > 3.6$$

$$R_{o,s} = \frac{1}{\left(5.763355 (SHGC - T_{Sol}) + 20.541528\right)}; U < 3.4$$

And for U-values between 3.4 and 4.5, the values are interpolated using results from both sets of equations.

The inward flowing fraction, $Frac_{inward}$, is then calculated using

$$Frac_{inward} = \frac{(R_{o,s} + 0.5 R_{l,w})}{(R_{o,s} + R_{l,w} + R_{i,s})}$$

The solar reflectances of the front face, $R_{s,f}$, and back face, $R_{s,b}$, are calculated using,

$$R_{s,f} = R_{s,b} = 1 - T_{Sol} - \frac{(SHGC - T_{Sol})}{Frac_{inward}}.$$

The thermal absorptance, or emittance, is taken as 0.84 for both the front and back and the long wave transmittance is 0.0.

Step 6. Determine Layer Visible Properties

The user has the option of entering a value for visible transmittance as one of the simple performance indices. If the user does not enter a value, then the visible properties are the same as the solar properties. If the user does enter a value then layer's visible transmittance at normal incidence, T_{Vis} , is set to that value. The visible light reflectance for the back surface is calculated using,

$$R_{Vis,b} = -0.7409 T_{Vis}^3 + 1.6531 T_{Vis}^2 - 1.2299 T_{Vis} + 0.4547$$

The visible light reflectance for the front surface is calculated using,

$$R_{Vis,f} = -0.0622 T_{Vis}^3 + 0.4277 T_{Vis}^2 - 0.4169 T_{Vis} + 0.2399$$

Glazing System Properties

The optical properties of a glazing system consisting of N glass layers separated by nonabsorbing gas layers (Figure 67. Schematic of transmission, reflection and absorption of solar radiation within a multi-layer glazing system.) are determined by solving the following recursion relations for $T_{i,j}$, the transmittance through layers i to j ; $R_{i,j}^f$ and $R_{i,j}^b$, the front and back reflectance, respectively, from layers i to j ; and A_j , the absorption in layer j . Here layer 1 is the outermost layer and layer N is the innermost layer. These relations account for multiple internal reflections within the glazing system. Each of the variables is a function of wavelength.

$$T_{i,j} = \frac{T_{i,j-1} T_{j,j}}{1 - R_{j,j}^f R_{j-1,i}^b} \quad (169)$$

$$R_{i,j}^f = R_{i,j-1}^f + \frac{T_{i,j-1}^2 R_{j,j}^f}{1 - R_{j,j}^f R_{j-1,i}^b} \quad (170)$$

$$R_{j,i}^b = R_{j,j}^b + \frac{T_{j,j}^2 R_{j-1,i}^b}{1 - R_{j-1,i}^b R_{j,j}^f} \quad (171)$$

$$A_j^f = \frac{T_{1,j-1}(1-T_{j,j}-R_{j,j}^f)}{1-R_{j,N}^f R_{j-1,1}^b} + \frac{T_{1,j}R_{j+1,N}^f(1-T_{j,j}-R_{j,j}^b)}{1-R_{j,N}^f R_{j-1,1}^b} \quad (172)$$

In Eq. (172) $T_{i,j} = 1$ and $R_{i,j} = 0$ if $i < 0$ or $j > N$.

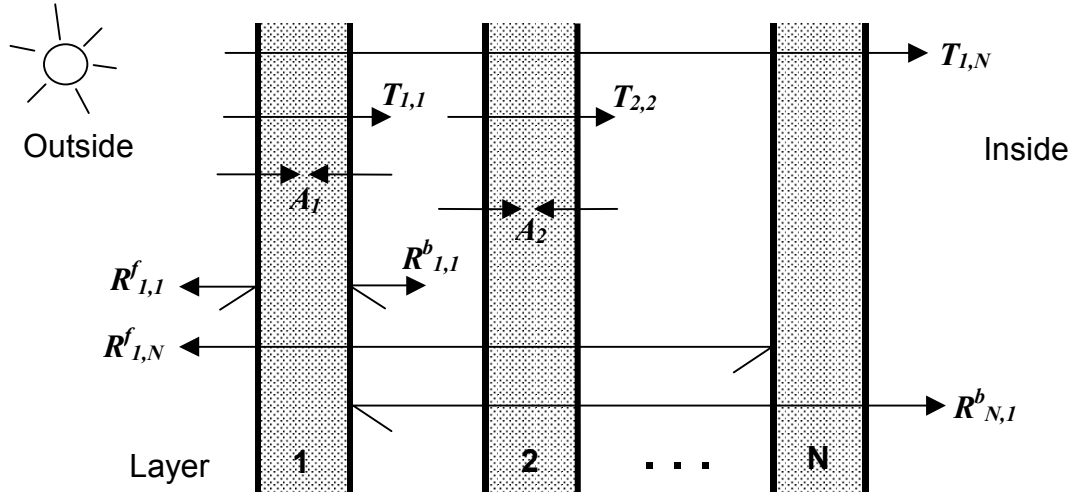


Figure 67. Schematic of transmission, reflection and absorption of solar radiation within a multi-layer glazing system.

As an example, for double glazing ($N=2$) these equations reduce to

$$T_{1,2} = \frac{T_{1,1}T_{2,2}}{1-R_{2,2}^f R_{1,1}^b}$$

$$R_{1,2}^f = R_{1,1}^f + \frac{T_{1,1}^2 R_{2,2}^f}{1-R_{2,2}^f R_{1,1}^b}$$

$$R_{2,1}^b = R_{2,2}^b + \frac{T_{2,2}^2 R_{1,1}^b}{1-R_{1,1}^f R_{2,2}^f}$$

$$A_1^f = (1-T_{1,1}-R_{1,1}^f) + \frac{T_{1,1}R_{2,2}^f(1-T_{1,1}-R_{1,1}^b)}{1-R_{2,2}^f R_{1,1}^b}$$

$$A_2^f = \frac{T_{1,1}(1-T_{2,2}-R_{2,2}^f)}{1-R_{2,2}^f R_{1,1}^b}$$

If the above transmittance and reflectance properties are input as a function of wavelength, EnergyPlus calculates “spectral average” values of the above glazing system properties by integrating over wavelength:

The spectral-average solar property is

$$P_s = \frac{\int P(\lambda)E_s(\lambda)d\lambda}{\int E_s(\lambda)d\lambda}$$

The spectral-average visible property is

$$P_v = \frac{\int P(\lambda)E_s(\lambda)V(\lambda)d\lambda}{\int E_s(\lambda)V(\lambda)d\lambda}$$

where $E_s(\lambda)$ is the solar spectral irradiance function and $V(\lambda)$ is the photopic response function of the eye. These functions are shown in Table 25 and Table 26. They are expressed as a set of values followed by the corresponding wavelengths for values.

If a glazing layer has optical properties that are roughly constant with wavelength, the wavelength-dependent values of $T_{i,i}$, $R_{i,i}^f$ and $R_{i,i}^b$ in Eqs. (169) to (172) can be replaced with constant values for that layer.

Table 25: Solar spectral irradiance function.

Air mass 1.5 terrestrial solar global spectral irradiance values (W/m ² -micron) on a 37° tilted surface. Corresponds to wavelengths in following data block. Based on ISO 9845-1 and ASTM E 892; derived from Optics5 data file ISO-9845GlobalNorm.std, 10-14-99.
0.0, 9.5, 42.3, 107.8, 181.0, 246.0, 395.3, 390.1, 435.3, 438.9, 483.7, 520.3, 666.2, 712.5, 720.7, 1013.1, 1158.2, 1184.0, 1071.9, 1302.0, 1526.0, 1599.6, 1581.0, 1628.3, 1539.2, 1548.7, 1586.5, 1484.9, 1572.4, 1550.7, 1561.5, 1501.5, 1395.5, 1485.3, 1434.1, 1419.9, 1392.3, 1130.0, 1316.7, 1010.3, 1043.2, 1211.2, 1193.9, 1175.5, 643.1, 1030.7, 1131.1, 1081.6, 849.2, 785.0, 916.4, 959.9, 978.9, 933.2, 748.5, 667.5, 690.3, 403.6, 258.3, 313.6, 526.8, 646.4, 746.8, 690.5, 637.5, 412.6, 108.9, 189.1, 132.2, 339.0, 460.0, 423.6, 480.5, 413.1, 250.2, 32.5, 1.6, 55.7, 105.1, 105.5, 182.1, 262.2, 274.2, 275.0, 244.6, 247.4, 228.7, 244.5, 234.8, 220.5, 171.5, 30.7, 2.0, 1.2, 21.2, 91.1, 26.8, 99.5, 60.4, 89.1, 82.2, 71.5, 70.2, 62.0, 21.2, 18.5, 3.2
Wavelengths (microns) corresponding to above data block
0.3000, 0.3050, 0.3100, 0.3150, 0.3200, 0.3250, 0.3300, 0.3350, 0.3400, 0.3450, 0.3500, 0.3600, 0.3700, 0.3800, 0.3900, 0.4000, 0.4100, 0.4200, 0.4300, 0.4400, 0.4500, 0.4600, 0.4700, 0.4800, 0.4900, 0.5000, 0.5100, 0.5200, 0.5300, 0.5400, 0.5500, 0.5700, 0.5900, 0.6100, 0.6300, 0.6500, 0.6700, 0.6900, 0.7100, 0.7180, 0.7244, 0.7400, 0.7525, 0.7575, 0.7625, 0.7675, 0.7800, 0.8000, 0.8160, 0.8237, 0.8315, 0.8400, 0.8600, 0.8800, 0.9050, 0.9150, 0.9250, 0.9300, 0.9370, 0.9480, 0.9650, 0.9800, 0.9935, 1.0400, 1.0700, 1.1000, 1.1200, 1.1300, 1.1370, 1.1610, 1.1800, 1.2000, 1.2350, 1.2900, 1.3200, 1.3500, 1.3950, 1.4425, 1.4625, 1.4770, 1.4970, 1.5200, 1.5390, 1.5580, 1.5780, 1.5920, 1.6100, 1.6300, 1.6460, 1.6780, 1.7400, 1.8000, 1.8600, 1.9200, 1.9600, 1.9850, 2.0050, 2.0350, 2.0650, 2.1000, 2.1480, 2.1980, 2.2700, 2.3600, 2.4500, 2.4940, 2.5370

Table 26: Photopic response function.

Photopic response function values corresponding to wavelengths in following data block. Based on CIE 1931 observer; ISO/CIE 10527, CIE Standard Colorimetric Observers; derived from Optics5 data file "CIE

1931 Color Match from E308.txt", which is the same as WINDOW4 file Cie31t.dat.
0.0000,0.0001,0.0001,0.0002,0.0004,0.0006,0.0012,0.0022,0.0040,0.0073, 0.0116,0.0168,0.0230,0.0298,0.0380,0.0480,0.0600,0.0739,0.0910,0.1126, 0.1390,0.1693,0.2080,0.2586,0.3230,0.4073,0.5030,0.6082,0.7100,0.7932, 0.8620,0.9149,0.9540,0.9803,0.9950,1.0000,0.9950,0.9786,0.9520,0.9154, 0.8700,0.8163,0.7570,0.6949,0.6310,0.5668,0.5030,0.4412,0.3810,0.3210, 0.2650,0.2170,0.1750,0.1382,0.1070,0.0816,0.0610,0.0446,0.0320,0.0232, 0.0170,0.0119,0.0082,0.0158,0.0041,0.0029,0.0021,0.0015,0.0010,0.0007, 0.0005,0.0004,0.0002,0.0002,0.0001,0.0001,0.0001,0.0000,0.0000,0.0000, 0.0000 /
Wavelengths (microns) corresponding to above data block
.380, .385, .390, .395, .400, .405, .410, .415, .420, .425, .430, .435, .440, .445, .450, .455, .460, .465, .470, .475, .480, .485, .490, .495, .500, .505, .510, .515, .520, .525, .530, .535, .540, .545, .550, .555, .560, .565, .570, .575, .580, .585, .590, .595, .600, .605, .610, .615, .620, .625, .630, .635, .640, .645, .650, .655, .660, .665, .670, .675, .680, .685, .690, .695, .700, .705, .710, .715, .720, .725, .730, .735, .740, .745, .750, .755, .760, .765, .770, .775, .780

Calculation of Angular Properties

Calculation of optical properties is divided into two categories: uncoated glass and coated glass.

Angular Properties for Uncoated Glass

The following discussion assumes that optical quantities such as transmissivity, reflectivity, absorptivity, and index of refraction are a function of wavelength, λ . If there are no spectral data the angular dependence is calculated based on the single values for transmittance and reflectance in the visible and solar range. In the visible range an average wavelength of 0.575 microns is used in the calculations. In the solar range an average wavelength of 0.898 microns is used.

The spectral data include the transmittance, T , and the reflectance, R . For uncoated glass the reflectance is the same for the front and back surfaces. For angle of incidence, ϕ , the transmittance and reflectance are related to the transmissivity, τ , and reflectivity, ρ , by the following relationships:

$$T(\phi) = \frac{\tau(\phi)^2 e^{-\alpha d / \cos \phi'}}{1 - \rho(\phi)^2 e^{-2\alpha d / \cos \phi'}} \quad (173)$$

$$R(\phi) = \rho(\phi) \left(1 + T(\phi) e^{-\alpha d / \cos \phi'} \right) \quad (174)$$

The spectral reflectivity is calculated from Fresnel's equation assuming unpolarized incident radiation:

$$\rho(\phi) = \frac{1}{2} \left(\left(\frac{n \cos \phi - \cos \phi'}{n \cos \phi + \cos \phi'} \right)^2 + \left(\frac{n \cos \phi' - \cos \phi}{n \cos \phi' + \cos \phi} \right)^2 \right) \quad (175)$$

The spectral transmittivity is given by

$$\tau(\phi) = 1 - \rho(\phi) \quad (176)$$

The spectral absorption coefficient is defined as

$$\alpha = 4\pi\kappa / \lambda \quad (177)$$

where κ is the dimensionless spectrally-dependent extinction coefficient and λ is the wavelength expressed in the same units as the sample thickness.

Solving Eq. (175) at normal incidence gives

$$n = \frac{1 + \sqrt{\rho(0)}}{1 - \sqrt{\rho(0)}} \quad (178)$$

Evaluating Eq. (174) at normal incidence gives the following expression for κ

$$\kappa = -\frac{\lambda}{4\pi d} \ln \frac{R(0) - \rho(0)}{\rho(0)T(0)} \quad (179)$$

Eliminating the exponential in Eqs. (173) and (174) gives the reflectivity at normal incidence:

$$\rho(0) = \frac{\beta - \sqrt{\beta^2 - 4(2 - R(0))R(0)}}{2(2 - R(0))} \quad (180)$$

where

$$\beta = T(0)^2 - R(0)^2 + 2R(0) + 1 \quad (181)$$

The value for the reflectivity, $\rho(0)$, from Eq. (180) is substituted into Eqs. (178) and (179). The result from Eq. (179) is used to calculate the absorption coefficient in Eq. (177). The index of refraction is used to calculate the reflectivity in Eq. (175) which is then used to calculate the transmittivity in Eq. (176). The reflectivity, transmissivity and absorption coefficient are then substituted into Eqs. (173) and (174) to obtain the angular values of the reflectance and transmittance.

Angular Properties for Coated Glass

A regression fit is used to calculate the angular properties of coated glass from properties at normal incidence. If the transmittance of the coated glass is > 0.645 , the angular dependence of uncoated clear glass is used. If the transmittance of the coated glass is ≤ 0.645 , the angular dependence of uncoated bronze glass is used. The values for the angular functions for the transmittance and reflectance of both clear glass ($\bar{\tau}_{clr}, \bar{\rho}_{clr}$) and bronze glass ($\bar{\tau}_{bnz}, \bar{\rho}_{bnz}$) are determined from a fourth-order polynomial regression:

$$\bar{\tau}(\phi) = \bar{\tau}_0 + \bar{\tau}_1 \cos(\phi) + \bar{\tau}_2 \cos^2(\phi) + \bar{\tau}_3 \cos^3(\phi) + \bar{\tau}_4 \cos^4(\phi)$$

and

$$\bar{\rho}(\phi) = \bar{\rho}_0 + \bar{\rho}_1 \cos(\phi) + \bar{\rho}_2 \cos^2(\phi) + \bar{\rho}_3 \cos^3(\phi) + \bar{\rho}_4 \cos^4(\phi) - \bar{\tau}(\phi)$$

The polynomial coefficients are given in Table 27.

Table 27: Polynomial coefficients used to determine angular properties of coated glass.

	0	1	2	3	4
$\bar{\tau}_{clr}$	-0.0015	3.355	-3.840	1.460	0.0288
$\bar{\rho}_{clr}$	0.999	-0.563	2.043	-2.532	1.054
$\bar{\tau}_{bnz}$	-0.002	2.813	-2.341	-0.05725	0.599
$\bar{\rho}_{bnz}$	0.997	-1.868	6.513	-7.862	3.225

These factors are used as follows to calculate the angular transmittance and reflectance:

For $T(0) > 0.645$:

$$T(\phi) = T(0)\bar{\tau}_{clr}(\phi)$$

$$R(\phi) = R(0)(1 - \bar{\rho}_{clr}(\phi)) + \bar{\rho}_{clr}(\phi)$$

For $T(0) \leq 0.645$:

$$T(\phi) = T(0)\bar{\tau}_{bnz}(\phi)$$

$$R(\phi) = R(0)(1 - \bar{\rho}_{bnz}(\phi)) + \bar{\rho}_{bnz}(\phi)$$

Angular Properties for Simple Glazing Systems

When the glazing system is modeled using the simplified method, an alternate method is used to determine the angular properties. The equation for solar transmittance as a function of incidence angle, $T(\phi)$, is,

$$T(\phi) = T(\phi = 0) \cos(\phi) \left(1 + (0.768 + 0.817 SHGC^4) \sin^3(\phi) \right)$$

where,

$T(\phi = 0)$ is the normal incidence solar transmittance, T_{Sol} .

The equation for solar reflectance as a function of incidence angle, $R(\phi)$, is,

$$R(\phi) = \frac{R(\phi = 0) \left(f_1(\phi) + f_2(\phi) \sqrt{SHGC} \right)}{R_{fit,o}}$$

where,

$$f_1(\phi) = \left(\left((2.403 \cos(\phi) - 6.192) \cos(\phi) + 5.625 \right) \cos(\phi) - 2.095 \right) \cos(\phi) + 1$$

$$f_2(\phi) = \left(\left((-1.188 \cos(\phi) + 2.022) \cos(\phi) + 0.137 \right) \cos(\phi) - 1.720 \right) \cos(\phi)$$

$$R_{fit,o} = 0.7413 - \left(0.7396 \sqrt{SHGC} \right)$$

Calculation of Hemispherical Values

The hemispherical value of a property is determined from the following integral:

$$P_{hemispherical} = 2 \int_0^{\frac{\pi}{2}} P(\phi) \cos(\phi) \sin(\phi) d\phi$$

The integral is evaluated by Simpson's rule for property values at angles of incidence from 0 to 90 degrees in 10-degree increments.

Optical Properties of Window Shading Devices

Shading devices affect the system transmittance and glass layer absorptance for short-wave radiation and for long-wave (thermal) radiation. The effect depends on the shade position (interior, exterior or between-glass), its transmittance, and the amount of inter-reflection between the shading device and the glazing. Also of interest is the amount of radiation absorbed by the shading device.

In EnergyPlus, shading devices are divided into four categories, "shades," "blinds," "screens," and "switchable glazing." "Shades" are assumed to be perfect diffusers. This means that direct radiation incident on the shade is reflected and transmitted as hemispherically uniform diffuse radiation: there is no direct component of transmitted radiation. It is also assumed that the transmittance, τ_{sh} , reflectance, ρ_{sh} , and absorptance, α_{sh} , are the same for the front and back of the shade and are independent of angle of incidence. Many types of drapery and pull-down roller devices are close to being perfect diffusers and can be categorized as "shades."

"Blinds" in EnergyPlus are slat-type devices such as venetian blinds. Unlike shades, the optical properties of blinds are strongly dependent on angle of incidence. Also, depending on slat angle and the profile angle of incident direct radiation, some of the direct radiation may pass between the slats, giving a direct component of transmitted radiation.

"Screens" are debris or insect protection devices made up of metallic or non-metallic materials. Screens may also be used as shading devices for large glazing areas where excessive solar gain is an issue. The EnergyPlus window screen model assumes the screen is composed of intersecting orthogonally-crossed cylinders, with the surface of the cylinders assumed to be diffusely reflecting. Screens may only be used on the exterior surface of a window construction. As with blinds, the optical properties affecting the direct component of transmitted radiation are dependent on the angle of incident direct radiation.

With "Switchable glazing," shading is achieved making the glazing more absorbing or more reflecting, usually by an electrical or chemical mechanism. An example is electrochromic glazing where the application of an electrical voltage or current causes the glazing to switch from light to dark.

Shades and blinds can be either fixed or moveable. If moveable, they can be deployed according to a schedule or according to a trigger variable, such as solar radiation incident on the window. Screens can be either fixed or moveable according to a schedule.

Shades

Shade/Glazing System Properties for Short-Wave Radiation

Short-wave radiation includes

- 1) Beam solar radiation from the sun and diffuse solar radiation from the sky and ground incident on the outside of the window,
- 2) Beam and/or diffuse radiation reflected from exterior obstructions or the building itself,
- 3) Solar radiation reflected from the inside zone surfaces and incident as diffuse radiation on the inside of the window,
- 4) Beam solar radiation from one exterior window incident on the inside of another window in the same zone, and
- 5) Short-wave radiation from electric lights incident as diffuse radiation on the inside of the window.

Exterior Shade

For an exterior shade we have the following expressions for the system transmittance, the effective system glass layer absorptance, and the system shade absorptance, taking inter-reflection between shade and glazing into account. Here, “system” refers to the combination of glazing and shade. The system properties are given in terms of the isolated shade properties (i.e., shade properties in the absence of the glazing) and the isolated glazing properties (i.e., glazing properties in the absence of the shade).

$$T_{sys}(\phi) = T_{1,N}^{dif} \frac{\tau_{sh}}{1 - R_f^{dif} \rho_{sh}}$$

$$T_{sys}^{dif} = T_{1,N}^{dif} \frac{\tau_{sh}}{1 - R_f^{dif} \rho_{sh}}$$

$$A_{j,f}^{sys}(\phi) = A_{j,f}^{dif} \frac{\tau_{sh}}{1 - R_f \rho_{sh}}, \quad j = 1 \text{ to } N$$

$$A_{j,f}^{dif,sys} = A_{j,f}^{dif} \frac{\tau_{sh}}{1 - R_f \rho_{sh}}, \quad j = 1 \text{ to } N$$

$$A_{j,b}^{dif,sys} = A_{j,b}^{dif} \frac{T_{1,N}^{dif} \rho_{sh}}{1 - R_f \rho_{sh}}, \quad j = 1 \text{ to } N$$

$$\alpha_{sh}^{sys} = \alpha_{sh} \left(1 + \frac{\tau_{sh} R_f}{1 - R_f \rho_{sh}} \right)$$

Interior Shade

The system properties when an interior shade is in place are the following.

$$T_{sys}(\phi) = T_{1,N}(\phi) \frac{\tau_{sh}}{1 - R_b^{dif} \rho_{sh}}$$

$$T_{sys}^{dif} = T_{1,N}^{dif} \frac{\tau_{sh}}{1 - R_b^{dif} \rho_{sh}}$$

$$A_{j,f}^{\text{sys}}(\phi) = A_{j,f}^{\text{dif}}(\phi) + T_{1,N}(\phi) \frac{\rho_{sh}}{1 - R_b^{\text{dif}} \rho_{sh}} A_{j,b}^{\text{dif}}, \quad j = 1 \text{ to } N$$

$$A_{j,f}^{\text{dif,sys}} = A_{j,f}^{\text{dif}} + T_{1,N}^{\text{dif}} \frac{\rho_{sh}}{1 - R_b^{\text{dif}} \rho_{sh}} A_{j,b}^{\text{dif}}, \quad j = 1 \text{ to } N$$

$$A_{j,b}^{\text{dif,sys}} = \frac{\tau_{sh}}{1 - R_b^{\text{dif}} \rho_{sh}} A_{j,b}^{\text{dif}}, \quad j = 1 \text{ to } N$$

$$\alpha_{sh}^{\text{sys}}(\phi) = T_{1,N}(\phi) \frac{\alpha_{sh}}{1 - R_b^{\text{dif}} \rho_{sh}}$$

$$\alpha_{sh}^{\text{dif,sys}} = T_{1,N}^{\text{dif}} \frac{\alpha_{sh}}{1 - R_b^{\text{dif}} \rho_{sh}}$$

Long-Wave Radiation Properties of Window Shades

Long-wave radiation includes

Thermal radiation from the sky, ground and exterior obstructions incident on the outside of the window,

Thermal radiation from other room surfaces incident on the inside of the window, and

Thermal radiation from internal sources, such as equipment and electric lights, incident on the inside of the window.

The program calculates how much long-wave radiation is absorbed by the shade and by the adjacent glass surface. The system emissivity (thermal absorptance) for an interior or exterior shade, taking into account reflection of long-wave radiation between the glass and shade, is given by

$$\varepsilon_{sh}^{\text{lw,sys}} = \varepsilon_{sh}^{\text{lw}} \left(1 + \frac{\tau_{sh}^{\text{lw}} \rho_{gl}^{\text{lw}}}{1 - \rho_{sh}^{\text{lw}} \rho_{gl}^{\text{lw}}} \right)$$

where ρ_{gl}^{lw} is the long-wave reflectance of the outermost glass surface for an exterior shade or the innermost glass surface for an interior shade, and it is assumed that the long-wave transmittance of the glass is zero.

The innermost (for interior shade) or outermost (for exterior shade) glass surface emissivity when the shade is present is

$$\varepsilon_{gl}^{\text{lw,sys}} = \varepsilon_{gl}^{\text{lw}} \frac{\tau_{sh}^{\text{lw}}}{1 - \rho_{sh}^{\text{lw}} \rho_{gl}^{\text{lw}}}$$

Switchable Glazing

For switchable glazing, such as electrochromics, the solar and visible optical properties of the glazing can switch from a light state to a dark state. The switching factor, f_{switch} , determines what state the glazing is in. An optical property, p , such as transmittance or glass layer absorptance, for this state is given by

$$p = (1 - f_{switch})p_{light} + f_{switch}p_{dark}$$

where

p_{light} is the property value for the unswitched, or light state, and p_{dark} is the property value for the fully switched, or dark state.

The value of the switching factor in a particular time step depends on what type of switching control has been specified: “schedule,” “trigger,” or “daylighting.” If “schedule,” f_{switch} = schedule value, which can be 0 or 1.

Thermochromic Windows

Thermochromic (TC) materials have active, reversible optical properties that vary with temperature. Thermochromic windows are adaptive window systems for incorporation into building envelopes. Thermochromic windows respond by absorbing sunlight and turning the sunlight energy into heat. As the thermochromic film warms it changes its light transmission level from less absorbing to more absorbing. The more sunlight it absorbs the lower the light level going through it. Figure 68 shows the variations of window properties with the temperature of the thermochromic glazing layer. By using the sun's own energy the window adapts based solely on the directness and amount of sunlight. Thermochromic materials will normally reduce optical transparency by absorption and/or reflection, and are specular (maintaining vision).

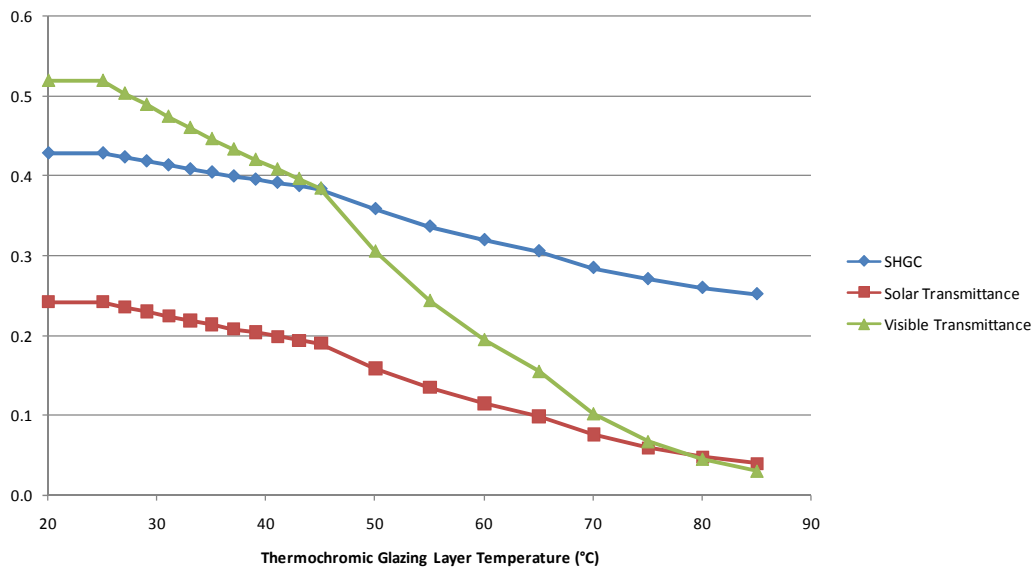


Figure 68. Variations of Window Properties with the Temperature of the Thermochromic Glazing Layer

On cloudy days the window is at full transmission and letting in diffuse daylighting. On sunny days the window maximizes diffuse daylighting and tints based on the angle of the sun relative to the window. For a south facing window (northern hemisphere) the daylight early and late in the day is maximized and the direct sun at mid day is minimized.

The active thermochromic material can be embodied within a laminate layer or a surface film. The overall optical state of the window at a given time is a function primarily of

- 1) thermochromic material properties
- 2) solar energy incident on the window
- 3) construction of the window system that incorporates the thermochromic layer
- 4) environmental conditions (interior, exterior, air temperature, wind, etc).

The tinted film, in combination with a heat reflecting, low-e layer allows the window to reject most of the absorbed radiation thus reducing undesirable heat load in a building. In the absence of direct sunlight the window cools and clears and again allows lower intensity diffuse radiation into a building. TC windows can be designed in several ways (Figure 69), with the most common being a triple pane windows with the TC glass layer in the middle a double pane windows with the TC layer on the inner surface of the outer pane or for sloped glazing a double pane with the laminate layer on the inner pane with a low-e layer toward the interior. The TC glass layer has variable optical properties depending on its temperature, with a lower temperature at which the optical change is initiated, and an upper temperature at which a minimum transmittance is reached. TC windows act as passive solar shading devices without the need for sensors, controls and power supplies but their optical performance is dependent on varying solar and other environmental conditions at the location of the window.

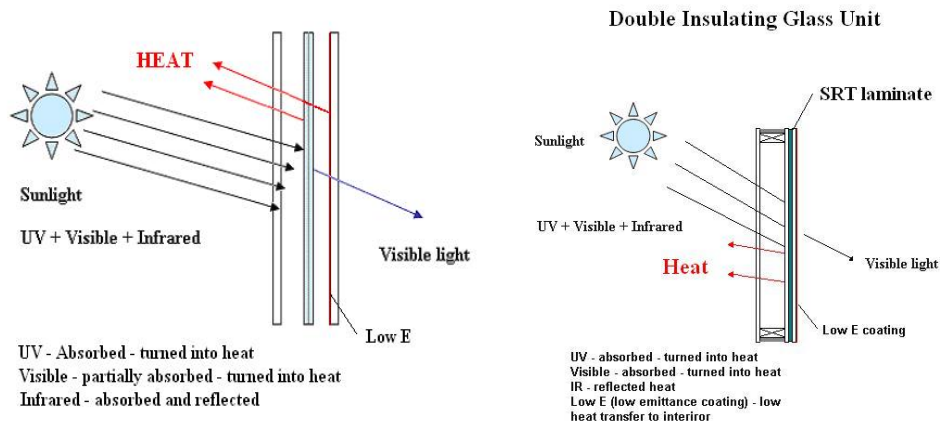


Figure 69. Configurations of Thermochromic Windows

EnergyPlus describes a thermochromic window with a Construction object which references a special layer defined with a WindowMaterial:GlazingGroup:Thermochromic object. The WindowMaterial:GlazingGroup:Thermochromic object further references a series of WindowMaterial:Glazing objects corresponding to each specification temperature of the TC layer. During EnergyPlus run time, a series of TC windows corresponding to each specification temperature is created once. At the beginning of a particular time step calculations, the temperature of the TC glass layer from the previous time step is used to look up the most closed specification temperature whose corresponding TC window construction will be used for the current time step calculations. The current time step calculated temperature of the TC glass layer can be different from the previous time step, but no iterations are done in the current time step for the new TC glass layer temperature. This is an approximation that considers the reaction time of the TC glass layer can be close to EnergyPlus simulation time step say 10 to 15 minutes.

Blinds

Window blinds in EnergyPlus are defined as a series of equidistant slats that are oriented horizontally or vertically. All of the slats are assumed to have the same optical properties. The overall optical properties of the blind are determined by the slat geometry (width, separation and angle) and the slat optical properties (front-side and back-side transmittance and reflectance). Blind properties for direct radiation are also sensitive to the “profile angle,” which is the angle of incidence in a plane that is perpendicular to the window plane and to the direction of the slats. The blind optical model in EnergyPlus is based on Simmler, Fischer and Winkelmann, 1996; however, that document has numerous typographical errors and should be used with caution.

The following assumptions are made in calculating the blind optical properties:

- The slats are flat.
- The spectral dependence of inter-reflections between slats and glazing is ignored; spectral-average slat optical properties are used.
- The slats are perfect diffusers. They have a perfectly matte finish so that reflection from a slat is isotropic (hemispherically uniform) and independent of angle of incidence, i.e., the reflection has no specular component. This also means that absorption by the slats is hemispherically uniform with no incidence angle dependence. If the transmittance of a slat is non-zero, the transmitted radiation is isotropic and the transmittance is independent of angle of incidence.
- Inter-reflection between the blind and wall elements near the periphery of the blind is ignored.
- If the slats have holes through which support strings pass, the holes and strings are ignored. Any other structures that support or move the slats are ignored.

Slat Optical Properties

The slat optical properties used by EnergyPlus are shown in the following table.

Table 28. Slat Optical Properties

$\tau_{dir,dif}$	Direct-to-diffuse transmittance (same for front and back of slat)
$\tau_{dif,dif}$	Diffuse-to-diffuse transmittance (same for front and back of slat)
$\rho_{dir,dif}^f, \rho_{dir,dif}^b$	Front and back direct-to-diffuse reflectance
$\rho_{dif,dif}^f, \rho_{dif,dif}^b$	Front and back diffuse-to-diffuse reflectance

It is assumed that there is no direct-to-direct transmission or reflection, so that $\tau_{dir,dir} = 0$, $\rho_{dir,dir}^f = 0$, and $\rho_{dir,dir}^b = 0$. It is further assumed that the slats are perfect diffusers, so that $\tau_{dir,dif}$, $\rho_{dir,dif}^f$ and $\rho_{dir,dif}^b$ are independent of angle of incidence. Until the EnergyPlus model is improved to take into account the angle-of-incidence dependence of slat transmission and reflection, it is assumed that $\tau_{dir,dif} = \tau_{dif,dif}$, $\rho_{dir,dif}^f = \rho_{dif,dif}^f$, and $\rho_{dir,dif}^b = \rho_{dif,dif}^b$.

Direct Transmittance of Blind

The direct-to-direct and direct-to-diffuse transmittance of a blind is calculated using the slat geometry shown in Figure 70 (a), which shows the side view of one of the cells of the blind. For the case shown, each slat is divided into two segments, so that the cell is bounded by a total of six segments, denoted by s_1 through s_6 (note in the following that s_i refers to both segment i and the length of segment i). The lengths of s_1 and s_2 are equal to the slat separation, h , which is the distance between adjacent slat faces. s_3 and s_4 are the segments illuminated by direct radiation. In the case shown in Figure 70 (a) the cell receives radiation by reflection of the direct radiation incident on s_4 and, if the slats have non-zero transmittance, by transmission through s_3 , which is illuminated from above.

The goal of the blind direct transmission calculation is to determine the direct and diffuse radiation leaving the cell through s_2 for unit direct radiation entering the cell through s_1 .

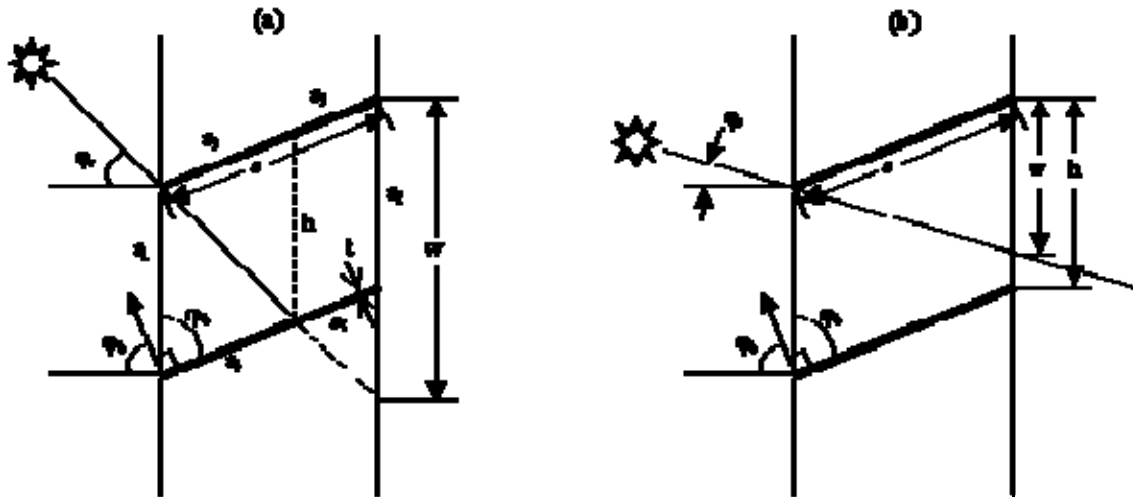


Figure 70. (a) Side view of a cell formed by adjacent slats showing how the cell is divided into segments, s_i , for the calculation of direct solar transmittance; (b) side view of a cell showing case where some of the direct solar passes between adjacent slats without touching either of them. In this figure ϕ_s is the profile angle and ϕ_b is the slat angle.

Direct-to-Direct Blind Transmittance

Figure 70 (b) shows the case where some of the direct radiation passes through the cell without hitting the slats. From the geometry in this figure we see that

$$\tau_{bl,f}^{dir,dir} = 1 - \frac{|w|}{h}, \quad |w| \leq h$$

where

$$w = s \frac{\cos(\phi_b - \phi_s)}{\cos \phi_s}$$

Note that we are assuming that the slat thickness is zero. A correction for non-zero slat thickness is described later.

Direct-to-Diffuse Blind Transmittance, Reflectance and Absorptance

The direct-to-diffuse and transmittance and reflectance of the blind are calculated using a radiosity method that involves the following three vector quantities:

J_i = the radiosity of segment s_i , i.e., the total radiant flux into the cell from s_i

G_i = the irradiance on the cell side of s_i

Q_i = the source flux from the cell side of s_i

Based on these definitions we have the following equations that relate J , G and Q for the different segments:

$$\begin{aligned}
J_1 &= Q_1 \\
J_2 &= Q_2 \\
J_3 &= Q_3 + \rho_{dif,dif}^b G_3 + \tau_{dif,dif} G_4 \\
J_4 &= Q_4 + \tau_{dif,dif} G_3 + \rho_{dif,dif}^f G_4 \\
J_5 &= Q_5 + \rho_{dif,dif}^b G_5 + \tau_{dif,dif} G_6 \\
J_6 &= Q_6 + \tau_{dif,dif} G_5 + \rho_{dif,dif}^f G_6
\end{aligned}$$

In addition we have the following equation relating G and J :

$$G_i = \sum_{j=1}^6 J_j F_{ji}, \quad i = 1, 6$$

where F_{ji} is the view factor between s_j and s_i , i.e., F_{ji} is the fraction of radiation leaving s_j that is intercepted by s_i .

Using $J_1 = Q_1 = 0$ and $J_2 = Q_2 = 0$ and combining the above equations gives the following equation set relating J and Q :

$$\begin{aligned}
J_3 - \rho_{dif,dif}^b \sum_{j=3}^6 J_j F_{j3} - \tau_{dif,dif} \sum_{j=3}^6 J_j F_{j4} &= Q_3 \\
J_4 - \tau_{dif,dif} \sum_{j=3}^6 J_j F_{j3} - \rho_{dif,dif}^f \sum_{j=3}^6 J_j F_{j4} &= Q_4 \\
J_5 - \rho_{dif,dif}^b \sum_{j=3}^6 J_j F_{j5} - \tau_{dif,dif} \sum_{j=3}^6 J_j F_{j6} &= Q_5 \\
J_6 - \tau_{dif,dif} \sum_{j=3}^6 J_j F_{j3} - \rho_{dif,dif}^f \sum_{j=3}^6 J_j F_{j6} &= Q_6
\end{aligned}$$

This can be written in the form

$$Q' = XJ' \tag{182}$$

where X is a 4x4 matrix and

$$J' = \begin{bmatrix} J_3 \\ J_4 \\ J_5 \\ J_6 \end{bmatrix} \quad Q' = \begin{bmatrix} Q_3 \\ Q_4 \\ Q_5 \\ Q_6 \end{bmatrix}$$

We then obtain J' from

$$J' = X^{-1}Q'$$

The view factors, F_{ij} , are obtained as follows. The cell we are dealing with is a convex polygon with n sides. In such a polygon the view factors must satisfy the following constraints:

$$\sum_{j=1}^n F_{ij} = 1, \quad i = 1, n$$

$$s_i F_{ij} = s_j F_{ji}, \quad i = 1, n; \quad j = 1, n$$

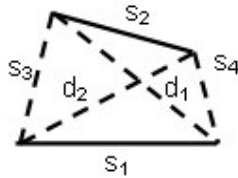
$$F_{ii} = 0, \quad i = 1, n$$

These constraints lead to simple equations for the view factors for $n = 3$ and 4. For $n = 3$, we have the following geometry and view factor expression:



$$F_{12} = \frac{s_1 + s_2 - s_3}{2s_1}$$

For $n = 4$ we have:



$$F_{12} = \frac{d_1 + d_2 - (s_3 + s_4)}{2s_1}$$

Applying these to the slat cell shown in Figure 71 we have the following:

$$F_{12} = \frac{d_1 + d_2 - 2s}{2h}$$

$$F_{13} = \frac{h + s_3 - d_3}{2h}, \text{ etc.}$$

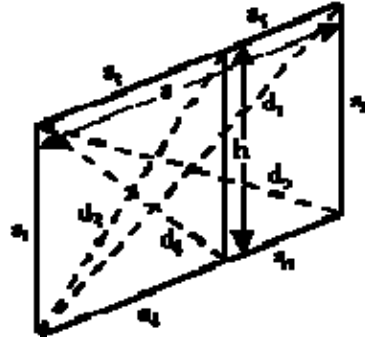


Figure 71. Slat cell showing geometry for calculation of view factors between the segments of the cell.

The sources for the direct-to-diffuse transmittance calculation are:

$$Q_1 = Q_2 = Q_5 = Q_6 = 0 \quad (\text{and therefore } J_1 = J_2 = 0)$$

$$\left. \begin{array}{l} Q_3 = \tau_{dir,dif} \\ Q_4 = \rho_{dir,dif}^f \end{array} \right\} \varphi_b \leq \varphi_s + \frac{\pi}{2} \quad (\text{beam hits front of slats})$$

$$\left. \begin{array}{l} Q_3 = \rho_{dir,dif}^b \\ Q_4 = \tau_{dir,dif} \end{array} \right\} \varphi_b > \varphi_s + \frac{\pi}{2} \quad (\text{beam hits back of slats})$$

For unit incident direct flux, the front direct-to-diffuse transmittance and reflectance of the blind are:

$$\begin{aligned} \tau_{bl,f}^{dir,dif} &= G_2 \\ \rho_{bl,f}^{dir,dif} &= G_1 \end{aligned}$$

where

$$G_2 = \sum_{j=3}^6 J_j F_{j2}$$

$$G_1 = \sum_{j=3}^6 J_j F_{j1}$$

and J_3 to J_6 are given by Eq. (182).

The front direct absorptance of the blind is then

$$\alpha_{bl,f}^{dir} = 1 - \tau_{bl,f}^{dir,dif} - \tau_{bl,f}^{dir,dir} - \rho_{bl,f}^{dir,dif}$$

The direct-to-diffuse calculations are performed separately for solar and visible slat properties to get the corresponding solar and visible blind properties.

Dependence on Profile Angle

The direct-to-direct and direct-to-diffuse blind properties are calculated for direct radiation profile angles (see Figure 70) ranging from -90° to $+90^\circ$ in 5° increments. (The "profile angle" is the angle of incidence in a plane that is perpendicular to the window *and*

perpendicular to the slat direction.) In the time step loop the blind properties for a particular profile angle are obtained by interpolation.

Dependence on Slat Angle

All blind properties are calculated for slat angles ranging from -90° to $+90^\circ$ in 10° increments. In the time-step loop the slat angle is determined by the slat-angle control mechanism and then the blind properties at that slat angle are determined by interpolation. Three slat-angle controls are available: (1) slat angle is adjusted to just block beam solar incident on the window; (2) slat angle is determined by a schedule; and (3) slat angle is fixed.

Diffuse-to-Diffuse Transmittance and Reflectance of Blind

To calculate the diffuse-to-diffuse properties, assuming uniformly distributed incident diffuse radiation, each slat bounding the cell is divided into two segments of equal length (Figure 72), i.e., $s_3 = s_4$ and $s_5 = s_6$. For front-side properties we have a unit source, $Q_1 = 1$. All the other Q_i are zero. Using this source value, we apply the methodology described above to obtain G_2 and G_1 . We then have

$$\begin{aligned}\tau_{bl,f}^{dif,dif} &= G_2 \\ \rho_{bl,f}^{dif,dif} &= G_1 \\ \alpha_{bl,f}^{dif} &= 1 - \tau_{bl,f}^{dif,dif} - \rho_{bl,f}^{dif,dif}\end{aligned}$$

The back-side properties are calculated in a similar way by setting $Q_2 = 1$ with the other Q_i equal to zero.

The diffuse-to-diffuse calculations are performed separately for solar, visible and IR slat properties to get the corresponding solar, visible and IR blind properties.

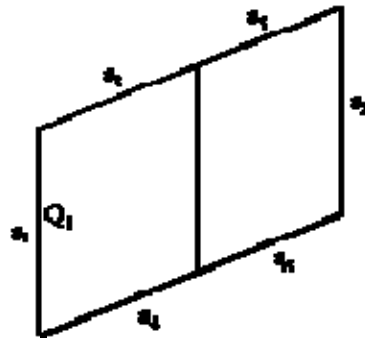


Figure 72. Slat cell showing arrangement of segments and location of source for calculation of diffuse-to-diffuse optical properties.

Blind properties for sky and ground diffuse radiation

For horizontal slats on a vertical window (the most common configuration) the blind diffuse-to-diffuse properties will be sensitive to whether the radiation is incident upward from the ground or downward from the sky (Figure 73). For this reason we also calculate the following solar properties for a blind consisting of horizontal slats in a vertical plane:

$$\tau_{bl,f}^{gnd-dif,dif} = \text{front transmittance for ground diffuse solar}$$

$$\tau_{bl,f}^{sky-dif,dif} = \text{front transmittance for sky diffuse solar}$$

$\rho_{bl,f}^{gnd-dif,dif}$ = front reflectance for ground diffuse solar

$\rho_{bl,f}^{sky-dif,dif}$ = front reflectance for sky diffuse solar

$\alpha_{bl,f}^{gnd-dif,dif}$ = front absorptance for ground diffuse solar

$\alpha_{bl,f}^{sky-dif,dif}$ = front absorptance for sky diffuse solar

These are obtained by integrating over sky and ground elements, as shown in Figure 73, treating each element as a source of direct radiation of irradiance $I(\phi_s)$ incident on the blind at profile angle ϕ_s . This gives:

$$\tau_{bl,f}^{sky-dif,dif} = \frac{\int_0^{\pi/2} \left[\tau_{bl,f}^{dir,dir}(\phi_s) + \tau_{bl,f}^{dir,dif}(\phi_s) \right] I_{sky}(\phi_s) \cos \phi_s d\phi_s}{\int_0^{\pi/2} I_{sky}(\phi_s) \cos \phi_s d\phi_s}$$

$$\rho_{bl,f}^{sky-dif,dif} = \frac{\int_0^{\pi/2} \rho_{bl,f}^{dir,dif} I_{sky}(\phi_s) \cos \phi_s d\phi_s}{\int_0^{\pi/2} I_{sky}(\phi_s) \cos \phi_s d\phi_s}$$

$$\alpha_{bl,f}^{sky-dif} = \frac{\int_0^{\pi/2} \alpha_{bl,f}^{dir} I_{sky}(\phi_s) \cos \phi_s d\phi_s}{\int_0^{\pi/2} I_{sky}(\phi_s) \cos \phi_s d\phi_s}$$

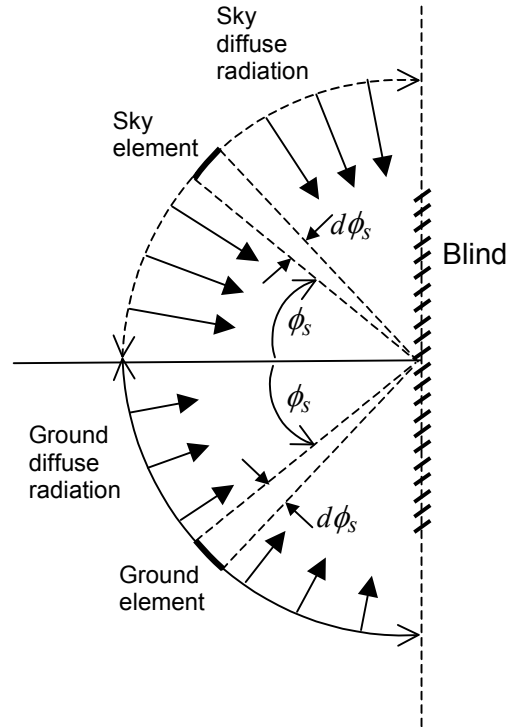


Figure 73. Side view of horizontal slats in a vertical blind showing geometry for calculating blind transmission, reflection and absorption properties for sky and ground diffuse radiation.

We assume that the sky radiance is uniform. This means that I_{sky} is independent of ϕ_s , giving:

$$\tau_{bl,f}^{sky-dif,dif} = \int_0^{\pi/2} \left[\tau_{bl,f}^{dir,dir} + \tau_{bl,f}^{dir,dif} \right] \cos \phi_s d\phi_s$$

$$\rho_{bl,f}^{sky-dif,dif} = \int_0^{\pi/2} \rho_{bl,f}^{dir,dif} \cos \phi_s d\phi_s$$

$$\alpha_{bl,f}^{sky-dif} = \int_0^{\pi/2} \alpha_{bl,f}^{dir} \cos \phi_s d\phi_s$$

The corresponding ground diffuse quantities are obtained by integrating ϕ_s from $-\pi/2$ to 0.

An improvement to this calculation would be to allow the sky radiance distribution to be non-uniform, i.e., to depend on sun position and sky conditions, as is done in the detailed daylighting calculation (see “Sky Luminance Distributions” under “Daylight Factor Calculation”).

Correction Factor for Slat Thickness

A correction has to be made to the blind transmittance, reflectance and absorptance properties to account for the amount of radiation incident on a blind that is reflected and absorbed by the slat edges (the slats are assumed to be opaque to radiation striking the slat edges). This is illustrated in Figure 74 for the case of direct radiation incident on the blind.

The slat cross-section is assumed to be rectangular. The quantity of interest is the fraction, f_{edge} , of direct radiation incident on the blind that strikes the slat edges. Based on the geometry shown in Figure 74 we see that

$$f_{edge} = \frac{t \cos \gamma}{\left(h + \frac{t}{\cos \xi}\right) \cos \varphi_s} = \frac{t \cos(\varphi_s - \xi)}{\left(h + \frac{t}{\cos \xi}\right) \cos \varphi_s} = \frac{t \sin(\varphi_b - \varphi_s)}{\left(h + \frac{t}{\sin \varphi_b}\right) \cos \varphi_s}$$

The edge correction factor for diffuse incident radiation is calculated by averaging this value of f_{edge} over profile angles, φ_s , from -90° to $+90^\circ$.

As an example of how the edge correction factor is applied, the following two equations show how blind front diffuse transmittance and reflectance calculated assuming zero slat thickness are modified by the edge correction factor. It is assumed that the edge transmittance is zero and that the edge reflectance is the same as the slat front reflectance, ρ_f .

$$\begin{aligned}\tau_{bl,f}^{dif,dif} &\rightarrow \tau_{bl,f}^{dif,dif} (1 - f_{edge}) \\ \rho_{bl,f}^{dif} &\rightarrow \rho_{bl,f}^{dif} (1 - f_{edge}) + f_{edge} \rho_f\end{aligned}$$

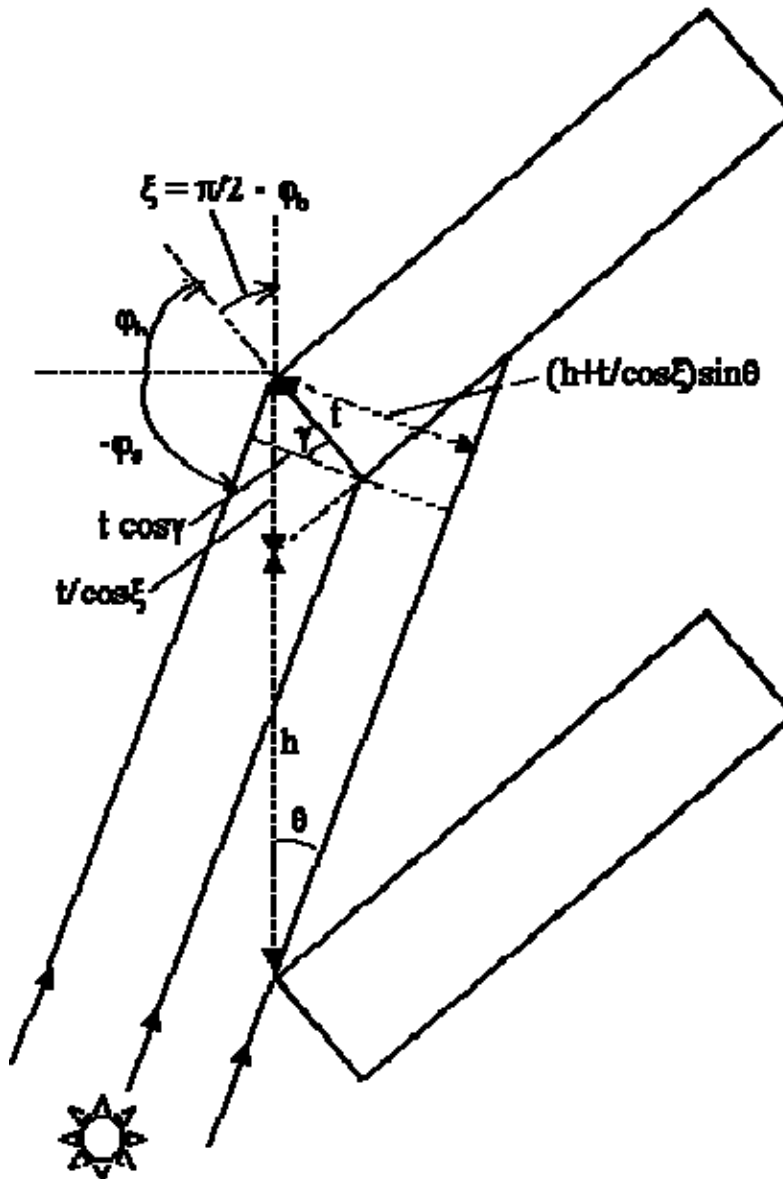


Figure 74. Side view of slats showing geometry for calculation of slat edge correction factor for incident direct radiation.

Comparison with ISO 15099 Calculation of Blind Optical Properties

Table 29 compares EnergyPlus and ISO 15099 [2001] calculations of blind optical properties for a variety of profile angles, slat angles and slat optical properties. The ISO 15099 calculation method is similar to that used in EnergyPlus, except that the slats are divided into five equal segments. The ISO 15099 and EnergyPlus results agree to within 12%, except for the solar transmittances for the 10-degree slat angle case. Here the transmittances are small (from 1% to about 5%) but differ by about a factor of up to two between ISO 15099 and EnergyPlus. This indicates that the slats should be divided into more than two segments at small slat angles.

Table 29. Comparison of blind optical properties calculated with the EnergyPlus and ISO 15099 methods. EnergyPlus values that differ by more than 12% from ISO 15099 values are shown in bold italics.

Slat properties										
Separation (m)	0.012		0.012		0.012		0.012		0.012	
Width (m)	0.016		0.016		0.016		0.016		0.016	
Angle (deg)	45		45		45		10		45	
IR transmittance	0.0		0.0		0.0		0.0		0.4	
IR emissivity, front side	0.9		0.9		0.9		0.9		0.55	
IR emissivity, back side	0.9		0.9		0.9		0.9		0.55	
Solar transmittance	0.0		0.0		0.0		0.0		0.4	
Solar reflectance, front side	0.70		0.55		0.70		0.70		0.50	
Solar reflectance, back side	0.70		0.55		0.40		0.40		0.50	
Solar Profile angle (deg)	0	60	0	60	0	60	0	60	0	60
Calculated blind properties (first row = ISO 15099 calculation, second row (in italics) = EnergyPlus calculation)										
Front solar transmittance, direct to direct	0.057	0.0	0.057	0.0	0.057	0.0	0.0	0.0	0.057	0.0
	<i>0.057</i>	<i>0.0</i>	<i>0.057</i>	<i>0.0</i>	<i>0.057</i>	<i>0.0</i>	<i>0.0</i>	<i>0.0</i>	<i>0.057</i>	<i>0.0</i>
Back solar transmittance, direct to direct	0.057	0.310	0.057	0.310	0.057	0.310	0.0	0.088	0.057	0.310
	<i>0.057</i>	<i>0.309</i>	<i>0.057</i>	<i>0.309</i>	<i>0.057</i>	<i>0.309</i>	<i>0.0</i>	<i>0.087</i>	<i>0.057</i>	<i>0.309</i>
Front solar transmittance, direct to diffuse	0.141	0.073	0.090	0.047	0.096	0.051	0.012	0.005	0.373	0.277
	<i>0.155</i>	<i>0.074</i>	<i>0.100</i>	<i>0.048</i>	<i>0.104</i>	<i>0.051</i>	<i>0.019</i>	<i>0.006</i>	<i>0.375</i>	<i>0.275</i>
Back solar transmittance, direct to diffuse	0.141	0.288	0.090	0.216	0.076	0.271	0.011	0.027	0.373	0.306
	<i>0.155</i>	<i>0.284</i>	<i>0.100</i>	<i>0.214</i>	<i>0.085</i>	<i>0.269</i>	<i>0.019</i>	<i>0.052</i>	<i>0.375</i>	<i>0.304</i>
Front solar reflectance, direct to diffuse	0.394	0.558	0.295	0.430	0.371	0.544	0.622	0.678	0.418	0.567
	<i>0.389</i>	<i>0.558</i>	<i>0.293</i>	<i>0.431</i>	<i>0.368</i>	<i>0.546</i>	<i>0.636</i>	<i>0.679</i>	<i>0.416</i>	<i>0.568</i>
Back solar reflectance, direct to diffuse	0.394	0.103	0.295	0.066	0.216	0.070	0.356	0.273	0.418	0.273
	<i>0.389</i>	<i>0.115</i>	<i>0.293</i>	<i>0.074</i>	<i>0.214</i>	<i>0.077</i>	<i>0.363</i>	<i>0.272</i>	<i>0.416</i>	<i>0.275</i>
Front solar transmittance, hemispherical diffuse to	0.332		0.294		0.291		0.038		0.495	
	<i>0.338</i>		<i>0.298</i>		<i>0.295</i>		<i>0.053</i>		<i>0.502</i>	
Back solar transmittance, hemispherical diffuse to diffuse	0.332		0.294		0.291		0.038		0.495	
	<i>0.338</i>		<i>0.298</i>		<i>0.295</i>		<i>0.053</i>		<i>0.502</i>	
Front hemispherical IR transmittance	0.227		0.227		0.227		0.0245		0.385	
	<i>0.227</i>		<i>0.227</i>		<i>0.227</i>		<i>0.025</i>		<i>0.387</i>	
Back hemispherical IR transmittance	0.227		0.227		0.227		0.0245		0.385	
	<i>0.227</i>		<i>0.227</i>		<i>0.227</i>		<i>0.025</i>		<i>0.387</i>	
Front hemispherical IR emissivity	0.729		0.729		0.729		0.890		0.536	
	<i>0.730</i>		<i>0.730</i>		<i>0.730</i>		<i>0.895</i>		<i>0.534</i>	
Back hemispherical IR emissivity	0.729		0.729		0.729		0.890		0.536	
	<i>0.730</i>		<i>0.730</i>		<i>0.730</i>		<i>0.895</i>		<i>0.534</i>	

Blind/Glazing System Properties for Short-Wave Radiation

When a blind is in place we have the following expressions for the system transmittance, the system glass layer absorptance, and the system blind absorptance, taking inter-reflection between blind and glazing into account. The system properties, indicated by “sys,” are given in terms of the isolated blind properties (i.e., blind properties in the absence of the glazing)—indicated by “bl”—and the isolated glazing properties (i.e., glazing properties in the absence of the blind)—indicated by “gl.”

Interior Blind

The system properties when an interior blind is in place are the following:

$$T_{f,sys}^{dir,all}(\phi, \phi_s) = T_{gl}^{dir}(\phi) \left(\tau_{bl,f}^{dir,dir}(\phi_s) + \tau_{bl,f}^{dir,dif}(\phi_s) + \frac{\tau_{bl,f}^{dif} \rho_{bl,f}^{dir,dif}(\phi_s) R_{gl,b}^{dif}}{1 - \rho_{bl,f}^{dif} R_{gl,b}^{dif}} \right)$$

$$A_{gl,j,f}^{dir,sys}(\phi, \phi_s) = A_{gl,j,f}^{dir}(\phi) + \frac{T_{gl}^{dir}(\phi) \alpha_{gl,j,b}^{dif} \rho_{bl,f}^{dir}(\phi_s)}{1 - \rho_{bl,f}^{dir}(\phi_s) R_{gl,b}^{dif}}, \quad j = 1, N$$

$$\alpha_{bl,f}^{dir,sys}(\phi, \phi_s) = T_{gl}^{dir}(\phi) \left(\alpha_{bl,f}^{dir}(\phi_s) + \frac{\rho_{bl,f}^{dir}(\phi_s) R_{gl,b}^{dif} \alpha_{bl,f}^{dif}}{1 - \rho_{bl,f}^{dif} R_{gl,b}^{dif}} \right)$$

$$T_{f,sys}^{dif,dif} = \frac{T_{gl}^{dif} \tau_{bl,f}^{dif,dif}}{1 - \rho_{bl,f}^{dif} R_{gl,b}^{dif}}$$

$$T_{f,sys}^{sky-dif,dif} = \frac{T_{gl}^{dif} \tau_{bl,f}^{sky-dif,dif}}{1 - \rho_{bl,f}^{sky-dif} R_{gl,b}^{dif}}$$

$$T_{f,sys}^{gnd-dif,dif} = \frac{T_{gl}^{dif} \tau_{bl,f}^{gnd-dif,dif}}{1 - \rho_{bl,f}^{gnd-dif} R_{gl,b}^{dif}}$$

$$A_{gl,j,f}^{dif,sys} = A_{gl,j,f}^{dif} + \frac{T_{gl}^{dif} \rho_{bl,f}^{dif} A_{gl,j,b}^{dif}}{1 - \rho_{bl,f}^{dif} R_{gl,b}^{dif}}, \quad j = 1, N$$

$$A_{gl,j,f}^{sky-dif,sys} = A_{gl,j,f}^{dif} + \frac{T_{gl}^{dif} \rho_{bl,f}^{sky-dif} A_{gl,j,b}^{dif}}{1 - \rho_{bl,f}^{sky-dif} R_{gl,b}^{dif}}, \quad j = 1, N$$

$$A_{gl,j,f}^{gnd-dif,sys} = A_{gl,j,f}^{dif} + \frac{T_{gl}^{dif} \rho_{bl,f}^{gnd-dif} A_{gl,j,b}^{dif}}{1 - \rho_{bl,f}^{gnd-dif} R_{gl,b}^{dif}}, \quad j = 1, N$$

$$\alpha_{bl,f}^{dif,sys} = \frac{T_{gl}^{dif} \alpha_{bl,f}^{dif}}{1 - \rho_{bl,f}^{dif} R_{gl,b}^{dif}}$$

$$\alpha_{bl,f}^{sky-dif,sys} = \frac{T_{gl}^{dif} \alpha_{bl,f}^{sky-dif}}{1 - \rho_{bl,f}^{sky-dif} R_{gl,b}^{dif}}$$

$$\alpha_{bl,f}^{gnd-dif,sys} = \frac{T_{gl}^{dif} \alpha_{bl,f}^{gnd-dif}}{1 - \rho_{bl,f}^{gnd-dif} R_{gl,b}^{dif}}$$

Exterior Blind

The system properties when an exterior blind is in place are the following:

$$T_{f,sys}^{dir,all}(\phi, \phi_s) = \tau_{bl,f}^{dir,dir}(\phi_s) \left(T_{gl}^{dir}(\phi) + \frac{T_{gl}^{dif} R_{gl,f}^{dir} \rho_{bl,b}^{dir,dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}} \right) + \frac{\tau_{bl}^{dir,dif}(\phi_s) T_{gl}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}$$

$$A_{gl,j,f}^{dir,sys}(\phi, \phi_s) = \tau_{bl,f}^{dir,dir}(\phi_s) A_{gl,j,f}^{dir}(\phi) + \frac{(\tau_{bl,f}^{dir,dir}(\phi_s) R_{gl}^{dir}(\phi) \rho_{bl,b}^{dir}(\phi_s) + \tau_{bl,f}^{dir,dif}(\phi_s)) A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}, \quad j=1, N$$

$$\alpha_{bl,f}^{dir,sys}(\phi, \phi_s) = \alpha_{bl,f}^{dir}(\phi_s) + \alpha_{bl,b}^{dir}(\phi_s) R_{gl,f}^{dir}(\phi) \tau_{bl,f}^{dir,dir}(\phi_s) + \frac{\alpha_{bl,b}^{dif} R_{gl,f}^{dif}}{1 - \rho_{bl,b}^{dif} R_{gl,f}^{dif}} (R_{gl,f}^{dir}(\phi) \tau_{bl,f}^{dir,dir}(\phi_s) \rho_{bl,b}^{dir}(\phi_s) + \tau_{bl,f}^{dir,dif}(\phi_s))$$

$$T_{f,sys}^{dif,dif} = \frac{\tau_{bl,f}^{dif,dif} T_{gl}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}$$

$$T_{f,sys}^{sky-dif,dif} = \frac{\tau_{bl,f}^{sky-dif,dif} T_{gl}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}$$

$$T_{f,sys}^{gnd-dif,dif} = \frac{\tau_{bl,f}^{gnd-dif,dif} T_{gl}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}$$

$$A_{gl,j,f}^{dif,sys} = \frac{\tau_{bl,f}^{dif,dif} A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}, \quad j=1, N$$

$$A_{gl,j,f}^{sky-dif,sys} = \frac{\tau_{bl,f}^{sky-dif,dif} A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}, \quad j=1, N$$

$$A_{gl,j,f}^{gnd-dif,sys} = \frac{\tau_{bl,f}^{gnd-dif,dif} A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}}, \quad j = 1, N$$

$$\alpha_{bl,f}^{dif,sys} = \alpha_{bl,f}^{dif} + \frac{\tau_{bl,f}^{dif,dif} R_{gl,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}} \alpha_{bl,b}^{dif}$$

$$\alpha_{bl,f}^{sky-dif,sys} = \alpha_{bl,f}^{sky-dif} + \frac{\tau_{bl,f}^{sky-dif,dif} R_{gl,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}} \alpha_{bl,b}^{dif}$$

$$\alpha_{bl,f}^{gnd-dif,sys} = \alpha_{bl,f}^{gnd-dif} + \frac{\tau_{bl,f}^{gnd-dif,dif} R_{gl,f}^{dif}}{1 - R_{gl,f}^{dif} \rho_{bl,b}^{dif}} \alpha_{bl,b}^{dif}$$

Blind/Glazing System Properties for Long-Wave Radiation

The program calculates how much long-wave radiation is absorbed by the blind and by the adjacent glass surface. The effective emissivity (long-wave absorptance) of an interior or exterior blind, taking into account reflection of long-wave radiation between the glass and blind, is given by

$$\varepsilon_{bl}^{lw,eff} = \varepsilon_{bl}^{lw} \left(1 + \frac{\tau_{bl}^{lw} \rho_{gl}^{lw}}{1 - \rho_{bl}^{lw} \rho_{gl}^{lw}} \right)$$

where ρ_{gl}^{lw} is the long-wave reflectance of the outermost glass surface for an exterior blind or the innermost glass surface for an interior blind, and it is assumed that the long-wave transmittance of the glass is zero.

The effective innermost (for interior blind) or outermost (for exterior blind) glass surface emissivity when the blind is present is

$$\varepsilon_{gl}^{lw,eff} = \varepsilon_{gl}^{lw} \frac{\tau_{bl}^{lw}}{1 - \rho_{bl}^{lw} \rho_{gl}^{lw}}$$

The effective inside surface emissivity is the sum of the effective blind and effective glass emissivities:

$$\varepsilon_{ins}^{lw,eff} = \varepsilon_{bl}^{lw,eff} + \varepsilon_{gl}^{lw,eff}$$

The effective temperature of the blind/glazing combination that is used to calculate the window's contribution to the zone's mean radiant temperature (MRT) is given by

$$T^{eff} = \frac{\varepsilon_{bl}^{lw,eff} T_{bl} + \varepsilon_{gl}^{lw,eff} T_{gl}}{\varepsilon_{bl}^{lw,eff} + \varepsilon_{gl}^{lw,eff}}$$

Solar Radiation Transmitted and Absorbed by a Window/Blind System

Let the direct solar incident on the window be

$$I_{dir,inc} = f_{sunlit} I_{dir,norm} \cos \phi \quad (W / m^2)$$

where f_{sunlit} is the fraction of the window that is sunlit (determined by the shadowing calculations), $I_{dir,norm}$ is the direct normal solar irradiance, and ϕ is the angle of incidence.

Let $I_{sky,inc}$ be the irradiance on the window due to diffuse solar radiation from the sky (W/m^2) and let $I_{gnd,inc}$ be the irradiance on the window due to diffuse solar radiation from the ground (W/m^2).

Then we have the following expressions for different classes of transmitted and absorbed solar radiation for the window/blind system (where ϕ_s is the direct solar profile angle), all in W/m^2 :

Direct solar entering zone from incident direct solar:

$$I_{dir,inc} T_{f,sys}^{dir,dir}(\phi, \phi_s)$$

Diffuse solar entering zone from incident direct solar:

$$I_{dir,inc} T_{f,sys}^{dir,dif}(\phi, \phi_s)$$

Direct solar absorbed by blind:

$$I_{dir,inc} \alpha_{bl,f}^{dir,sys}(\phi, \phi_s)$$

Direct solar absorbed by glass layers:

$$I_{dir,inc} A_{gl,j,f}^{dir,sys}(\phi, \phi_s), \quad j = 1, N$$

For windows whose blinds have vertical slats:

Diffuse solar entering zone from incident diffuse solar:

$$(I_{sky,inc} + I_{gnd,inc}) T_{f,sys}^{dif,dif}$$

Diffuse solar absorbed by blind:

$$(I_{sky,inc} + I_{gnd,inc}) \alpha_{bl,f}^{dif,sys}$$

Diffuse solar absorbed by glass layers:

$$(I_{sky,inc} + I_{gnd,inc}) A_{gl,j,f}^{dif,sys}, \quad j = 1, N$$

For windows of tilt angle γ whose blinds have horizontal slats:

(vertical windows have tilt = 90° , horizontal windows have tilt = 0°)

Diffuse solar entering zone from incident diffuse solar:

$$T_{f,sys}^{sky-dif,dif} \left[\left(1 - \frac{|\cos \gamma|}{2} \right) I_{sky,inc} + \frac{|\cos \gamma|}{2} I_{gnd,inc} \right] + T_{f,sys}^{gnd-dif,dif} \left[\frac{|\cos \gamma|}{2} I_{sky,inc} + \left(1 - \frac{|\cos \gamma|}{2} \right) I_{gnd,inc} \right]$$

Diffuse solar absorbed by blind:

$$\alpha_{bl,f}^{sky-dif,sys} \left[\left(1 - \frac{|\cos \gamma|}{2} \right) I_{sky,inc} + \frac{|\cos \gamma|}{2} I_{gnd,inc} \right] + \alpha_{bl,f}^{gnd-dif,sys} \left[\frac{|\cos \gamma|}{2} I_{sky,inc} + \left(1 - \frac{|\cos \gamma|}{2} \right) I_{gnd,inc} \right]$$

Diffuse solar absorbed by glass layers:

$$A_{gl,j,f}^{sky-dif,sys} \left[\left(1 - \frac{|\cos \gamma|}{2} \right) I_{sky,inc} + \frac{|\cos \gamma|}{2} I_{gnd,inc} \right] + A_{gl,j,f}^{gnd-dif,sys} \left[\frac{|\cos \gamma|}{2} I_{sky,inc} + \left(1 - \frac{|\cos \gamma|}{2} \right) I_{gnd,inc} \right], \quad j = 1, N$$

Screens

The model for window screens currently allows placement on the exterior surface of a window system (i.e., between glass and interior window screens can not be modeled). The exterior screen is modeled as a planar semi-transparent sheet having specular transmittance that is dependent on the angle of incidence of beam solar radiation. The screen transmittance algorithm includes two components. The first one, $T_{beam}(\alpha', \varphi')$, accounts for the blockage of the sun's rays by the screen material. This component accounts for the beam solar radiation passing through the screen openings without hitting the screen material. The second part, $T_{scatt}(\alpha', \varphi')$, accounts for the additional flux of transmitted beam solar radiation by diffuse reflectance (scattering) from the screen material. Since the reflected component is small compared with the incident beam and the direction of scattering is highly dependent on incident angle, the component of transmitted beam radiation due to screen material reflectance can be treated in one of three ways based on a user input to the model.

The user may elect not to model the inward reflected beam transmittance due to the uncertainty of the direction of scattering or its low magnitude for low-reflecting screen materials. The user may alternately choose to model the inwardly-reflected transmitted beam as an additive component to the direct beam transmittance in the same solid angle and direction. Finally, the additional flux due to the inward reflection of direct beam radiation may be modeled as hemispherically-diffuse transmittance.

This reflected beam transmittance component depends upon the diffuse (i.e., beam-to-diffuse) reflectance of the screen material, so this reflectance (ρ_{sc}) is a required input to the model. Guidance input values for this diffuse reflectance are provided, to account for screens that are dark, medium, or light colored in appearance, in the likely case that more accurate values for the material reflectance are difficult or time-consuming to obtain. If the diffuse reflectance of the screen material is known, use this value in place of the guidance provided.

The model is based on an orthogonal crossed cylinder geometry in which the screen material's cylindrical diameter and spacing are known. The model assumes that the screen material diameter and spacing are the same in both directions. Figure 75 shows a rendering of intersecting orthogonal crossed cylinders used as the basis for the EnergyPlus screen model.

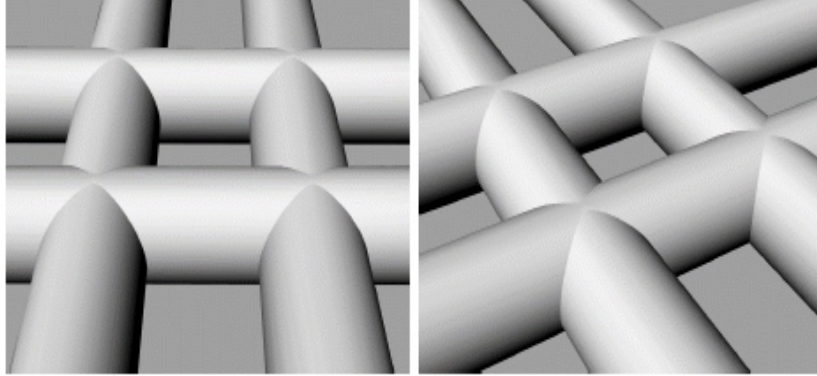


Figure 75. Screen model rendering of intersecting orthogonal crossed cylinders

If the required screen material dimensions are not available from the manufacturer, they may be determined using the following procedure:

- Lay the screen next to a finely-divided scale or ruler. A magnifying glass may be helpful in determining the screen material dimensions. Alternately, a photograph can be taken and the image enlarged.
- Determine the diameter D of an individual screen material “cylinder”. Average the diameter values if different in opposing directions.
- Determine the average center-to-center spacing S of the screen material or measure from one side of a “cylinder” to the same side of the next “cylinder” and record the spacing S . Average the spacing values if different in opposing directions.
- Enter these values as inputs to the exterior window screen model.

The screen material diameter and spacing are then used to determine the screen material aspect ratio for use in the screen model.

$$\gamma = D/S$$

where

γ = Screen material aspect ratio, dimensionless

D = Screen material diameter, m

S = Screen material spacing, m

Figure 76 below shows the input requirements for material diameter and spacing and the associated calculation for openness factor, the equivalent to T_{beam} at direct normal incidence.

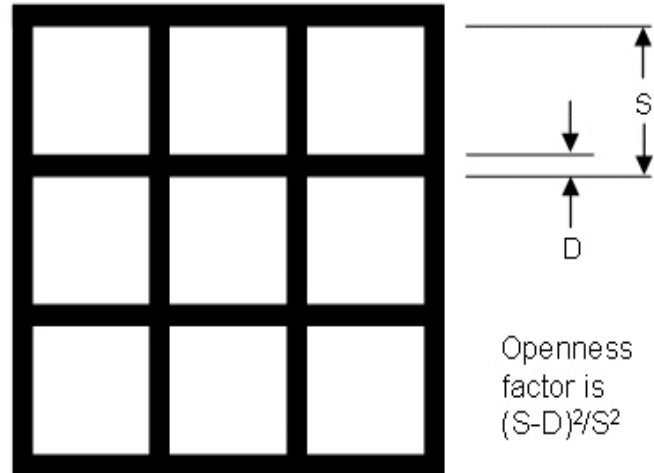


Figure 76. Physical screen material properties

Screen Beam Transmittance

The first component of the window screen transmittance model is a geometric representation of the open area of the screen material and is dependent on the angle of incident beam radiation. Figure 77 shows a schematic of a South-facing vertical window screen and the solar angles used in EnergyPlus. The window screen model is based on the relative angles of incidence of the sun's rays with respect to the window screen outward normal. In the figure, the relative solar azimuth and relative solar altitude are represented as ϕ' and α' , respectively.

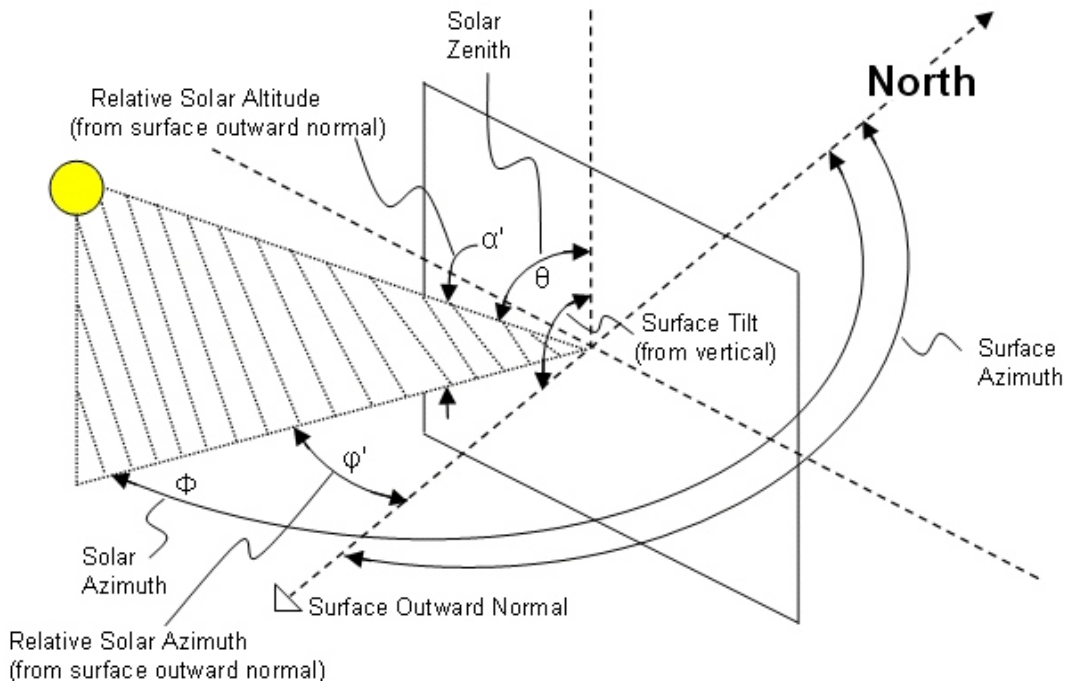


Figure 77. Schematic of a vertical window screen facing due South

Given the diffuse reflectance ρ_{sc} and the screen aspect ratio γ , the model takes the direction of solar incidence, the relative solar altitude angle α' and the relative solar azimuth angle ϕ' , illustrated in Figure 77, and calculates the direct beam transmittance $T_{beam}(\alpha', \phi')$ as follows.

Since the direct beam transmittance is only a function of the incident angle and the screen material aspect ratio, the following applies to both solar and visible radiation.

$$\alpha'' = \arctan(\tan \alpha' \sec \varphi')$$

$$\beta = \frac{\pi}{2} - \varphi'$$

$$T_y = 1 - \gamma \left[\cos \alpha'' + \sin \alpha'' \tan \alpha' (1 + \cot^2 \beta)^{\frac{1}{2}} \right]$$

$$\mu = \arccos \left(\left[\cos^2 \alpha' \cos^2 \varphi' + \sin^2 \alpha' \right]^{\frac{1}{2}} \right)$$

$$\varepsilon = \arccos \left(\frac{[\cos \alpha' \cos \varphi']}{\cos \mu} \right)$$

$$\eta = \frac{\pi}{2} - \varepsilon$$

$$\mu' = \arctan(\tan \mu \sec \varepsilon)$$

$$T_x = 1 - \gamma \left[\cos \mu' + \sin \mu' \tan \mu (1 + \cot^2 \eta)^{\frac{1}{2}} \right]$$

$$T_{beam}(\alpha', \varphi') = T_{beam}^{vis}(\alpha', \varphi') = T_x T_y$$

where

T_y = vertical component of direct beam transmittance

T_x = horizontal component of direct beam transmittance

T_{beam} = direct screen transmittance that accounts for beam solar radiation passing through the screen openings without hitting the screen material

T_{beam}^{vis} = direct visible screen transmittance that accounts for beam solar radiation passing through the screen openings without hitting the screen material

α' = relative solar altitude angle [radians]

φ' = relative solar azimuth angle [radians]

γ = Screen material aspect ratio, dimensionless

$\alpha'', \beta, \mu, \varepsilon, \eta, \mu'$ = intermediate variables

This first component of screen direct beam transmittance was developed using geometric principals and was verified using an optical ray tracing software program.

The second component of the window screen transmittance model is an empirical algorithm that accounts for the inward reflection of incident beam radiation off the screen material surface. The calculation procedure for the screen's transmittance via beam reflection, T_{scatt} (α' , φ') is as follows:

$$T_{scattmax} = 0.0229\gamma + 0.2971\rho_{sc} - 0.03624\gamma^2 + 0.04763\rho_{sc}^2 - 0.44416\gamma\rho_{sc}$$

$$T_{scattmax}^{vis} = 0.0229\gamma + 0.2971\rho_{sc}^{vis} - 0.03624\gamma^2 + 0.04763(\rho_{sc}^{vis})^2 - 0.44416\gamma\rho_{sc}^{vis}$$

$$\delta_{max} = 89.7 - 10\gamma/0.16$$

$$\delta = (\alpha_d'^2 + \varphi_d'^2)^{\frac{1}{2}}$$

$$Peak_{ratio} = 1.0 / (0.2\rho_{sc}(1-\gamma))$$

$$Peak_{ratio}^{vis} = 1.0 / (0.2\rho_{sc}^{vis}(1-\gamma))$$

$$T_{scatt}(\alpha', \varphi') = 0.2\rho_{sc}T_{scattmax}(1-\gamma) \left(1 + (Peak_{ratio} - 1)e^{-\frac{|\delta - \delta_{max}|^{2.0}}{600}} \right)$$

$$T_{scatt}^{vis}(\alpha', \varphi') = 0.2\rho_{sc}^{vis}T_{scattmax}^{vis}(1-\gamma) \left(1 + (Peak_{ratio}^{vis} - 1)e^{-\frac{|\delta - \delta_{max}|^{2.0}}{600}} \right)$$

$$IF(\delta > \delta_{max}) T_{scatt}(\alpha', \varphi') = 0.2\rho_{sc}T_{scattmax}(1-\gamma) \left(1 + (Peak_{ratio} - 1)e^{-\frac{|\delta - \delta_{max}|^{2.5}}{600}} \right) -$$

$$0.2\rho_{sc}T_{scattmax}(1-\gamma) \left(MAX \left(0.0, \frac{\delta - \delta_{max}}{90. - \delta_{max}} \right) \right)$$

$$IF(\delta > \delta_{max}) T_{scatt}^{vis}(\alpha', \varphi') = 0.2\rho_{sc}^{vis}T_{scattmax}^{vis}(1-\gamma) \left(1 + (Peak_{ratio}^{vis} - 1)e^{-\frac{|\delta - \delta_{max}|^{2.5}}{600}} \right) -$$

$$0.2\rho_{sc}^{vis}T_{scattmax}^{vis}(1-\gamma) \left(MAX \left(0.0, \frac{\delta - \delta_{max}}{90. - \delta_{max}} \right) \right)$$

where

$T_{scattmax}$ = maximum reflected (scattered) beam transmittance

$T_{scattmax}^{vis}$ = maximum visible reflected (scattered) beam transmittance

δ_{max}, δ = intermediate variables [degrees]

α'_d = relative solar altitude [degrees]

ϕ'_d = relative solar azimuth [degrees]

$Peak_{ratio}$ = Ratio of peak scattered beam transmittance to scattered beam transmittance at direct normal incidence.

$Peak_{ratio}^{vis}$ = Ratio of peak scattered visible transmittance to scattered visible transmittance at direct normal incidence.

ρ_{sc} = diffuse solar reflectance of the screen material

ρ_{sc}^{vis} = diffuse visible reflectance of the screen material

T_{scatt} = beam solar transmittance due to reflectance (scattering)

T_{scatt}^{vis} = beam visible transmittance due to reflectance (scattering)

The reflected (scattered) transmittance of incident beam radiation is an empirical model derived by curvefitting results from optical ray trace modeling. Ray traces were performed for a range of screen aspect ratios, diffuse screen reflectances, and relative solar azimuth and altitude angles. The surface of the screen cylinders was assumed to be diffusely reflecting, having the optical properties of a Lambertian surface. The transmitted flux due to reflection was determined by a hemispherical detector on the transmitted side of the screen.

These two components of beam solar transmittance are then used to specify the properties for beam-to-beam and beam-to-diffuse transmittance for the screen based on the user selection for Reflected Beam Transmittance Accounting Method in the WindowMaterial:Screen object. The calculations below apply to both the solar and visible beam solar transmittance.

If the user selects DoNotModel, the direct beam transmittance is set to T_{beam} and the reflected (scattered) portion of the beam solar transmittance is ignored:

$$T_{sc}^{dir,dir} = T_{beam}(\alpha', \phi')$$

$$T_{sc}^{dir,dif} = 0.0$$

$$T_{sc,vis}^{dir,dir} = T_{beam}(\alpha', \phi')$$

$$T_{sc,vis}^{dir,dif} = 0.0$$

where

$T_{sc}^{dir,dir}$ = direct-to-direct beam transmittance of the screen (output report variable Screen Beam-Beam Solar Transmittance)

$T_{sc}^{dir,dif}$ = direct-to-diffuse beam transmittance of the screen (output report variable Screen Beam-Diffuse Solar Transmittance)

$T_{sc,vis}^{dir,dir}$ = direct-to-direct visible transmittance of the screen

$T_{sc,vis}^{dir,dif}$ = direct-to-diffuse visible transmittance of the screen

If the user selects Model as Direct Beam, the reflected (scattered) portion of the beam solar transmittance is added to the direct beam transmittance T_{beam} in the same solid angle and direction of the unattenuated solar beam:

$$T_{sc}^{dir,dir} = T_{beam}(\alpha', \varphi') + T_{scatt}(\alpha', \varphi')$$

$$T_{sc}^{dir,dif} = 0.0$$

$$T_{sc,vis}^{dir,dir} = T_{beam}(\alpha', \varphi') + T_{scatt}^{vis}(\alpha', \varphi')$$

$$T_{sc,vis}^{dir,dif} = 0.0$$

If the user selects Model as Diffuse Beam, the direct beam transmittance is set to T_{beam} and the reflected (scattered) portion of the beam solar transmittance is modeled as diffuse hemispherical radiation:

$$T_{sc}^{dir,dir} = T_{beam}(\alpha', \varphi')$$

$$T_{sc}^{dir,dif} = T_{scatt}(\alpha', \varphi')$$

$$T_{sc,vis}^{dir,dir} = T_{beam}(\alpha', \varphi')$$

$$T_{sc,vis}^{dir,dif} = T_{scatt}^{vis}(\alpha', \varphi')$$

Screen Beam Reflectance

The screen reflectance (overall value for the screen assembly, accounting for the screen material itself and the open spaces between the screen material) is calculated by first subtracting the direct-to-direct screen transmittance from the unit incident beam. This approximates the fraction of incident beam solar radiation striking the screen that is not inwardly transmitted. The result is then multiplied by the screen material diffuse reflectance ρ_{sc} . The inwardly scattered transmittance is then subtracted from this quantity to obtain an

approximate value for the screen's reflectance R_{sc} to beam radiation incident as a function of the relative angles of incident radiation. This equation is used for both beam and visible reflectance:

$$R_{SC}^{dir,dif}(\alpha', \varphi') = \rho_{SC} (1 - T_{SC}^{dir,dif}) - T_{SC}^{dir,dif}$$

$$R_{SC,vis}^{dir,dif}(\alpha', \varphi') = \rho_{SC}^{vis} (1 - T_{SC,vis}^{dir,dif}) - T_{SC,vis}^{dir,dif}$$

Screen Beam Absorptance

The screen absorptance (overall value for the screen assembly, accounting for the screen material itself and the open spaces between the screen material) is calculated as the quantity of the unit incident flux (1) less the directly-transmitted component $T_{dir,dif}$ multiplied by the quantity 1 minus the screen material diffuse reflectance.

$$A_{SC}^{dir}(\alpha', \varphi') = (1 - T_{SC}^{dir,dif})(1 - \rho_{SC})$$

Screen Diffuse Properties

The transmittance of the screen to half-hemispherical diffuse (sky) radiation is calculated by performing a finite-element-summation, approximately equivalent to an integration over the solid angle of the beam transmittance, assuming uniform radiance. This single-number screen diffuse transmittance is then multiplied by the irradiance incident on the screen from a uniform half-hemisphere of sky- or ground-reflected radiation to determine the level of additional flux transmitted by the screen to the window from the diffuse sky or ground. The sun angles shown in the figure below represent the solar altitude angle (θ) and solar azimuth angle (Φ) in polar coordinates. These angles are used to calculate the average diffuse-to-diffuse properties for screens in the following derivations.

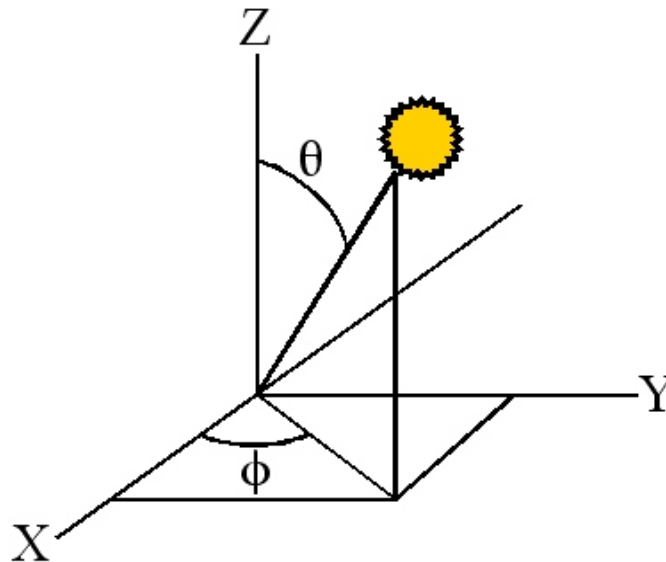


Figure 78. Sun Angles in Screen Calculations.

The screen transmittance to diffuse radiation $T_{dif,dif}(\gamma, \rho_{sc})$ is computed as the integrated average of the combined beam transmittance $T_{tot}(\gamma, \rho, \theta, \Phi)$ over the directions of incidence using spherical coordinates (θ, Φ) in which the z-axis is perpendicular to the plane of the screen. Using a finite element computation, this is:

$$T_{tot}(\gamma, \rho_{sc}, \theta_j, \phi_i) = T_{beam}(\alpha', \varphi') + T_{scatt}(\alpha', \varphi')$$

$$T_{sc}^{dif,dif}(\gamma, \rho_{sc}) = \frac{\sum_{j=1}^N \sum_{i=1}^M T_{tot}(\gamma, \rho_{sc}, \theta_j, \phi_i) \sin(\theta_j) \cos(\theta_j)}{\sum_{j=1}^N \sum_{i=1}^M \sin(\theta_j) \cos(\theta_j)}$$

where

θ = solar altitude angle in polar coordinates [radians]

ϕ = solar azimuth angle in polar coordinates [radians]

$T_{sc}^{dif,dif}(\gamma, \rho_{sc})$ = diffuse-diffuse transmittance (output report variable Screen Diffuse-Diffuse Solar Transmittance)

Similarly, the reflectance of the screen to diffuse radiation is given by

$$R_{sc}^{dif,dif}(\gamma, \rho_{sc}) = \frac{\sum_{j=1}^N \sum_{i=1}^M R_{sc}^{dir,dif}(\gamma, \rho_{sc}, \theta_j, \phi_i) \sin(\theta_j) \cos(\theta_j)}{\sum_{j=1}^N \sum_{i=1}^M \sin(\theta_j) \cos(\theta_j)}$$

There is an assumption in both of these formulas that the brightness of the sky (or ground) diffuse radiation is the same for all directions. For this reason, the solar azimuth angle Φ and solar altitude angle θ have a range of 0 to $\pi/2$ (instead of $-\pi/2$ to $+\pi/2$) because the screen is assumed to have identical optical properties for radiation incident at the same angles on either side of a vertical or horizontal plane perpendicular to the screen.

Since the screen direct transmittance model is derived with respect to a different coordinate axis labeling, a coordinate transform is needed in order to calculate the diffuse optical properties. In these calculations, for each spherical solar coordinates (θ , Φ) we need the corresponding screen relative solar coordinates (α' , φ') to evaluate the screen transmittance model for that direction.

For each θ and Φ in the summation, the corresponding values for the relative solar altitude α' and relative solar azimuth φ' needed to calculate screen transmittance are determined with the following coordinate transform equations:

$$\sin \alpha' = \sin \theta \cos \phi$$

$$\tan \varphi' = \tan \theta \sin \phi$$

The absorptance of the screen to diffuse incident radiation is calculated by subtracting the diffuse transmittance and diffuse reflectance from unity as follows:

$$A_{sc}^{dif}(\gamma, \rho_{sc}) = 1 - T_{sc}^{dif,dif}(\gamma, \rho_{sc}) - R_{sc}^{dif,dif}(\gamma, \rho_{sc})$$

Screen/Glass System Properties for Short-Wave Radiation

The combined system properties of the screen/glass combination are calculated using the properties of the screen in combination with the bare glass material properties.

Interreflections of radiation between the screen and glass surfaces are included. The following infinite series serves as an example for calculating the combined screen/glass system properties. The terms of the series are built up as illustrated in the following figure. The terms shown at the right of the figure represent each term in the infinite series for the combined screen/glass property (beam transmittance in this example).

For the example of beam transmittance, the incident solar beam strikes the screen at the incident angle associated with the current relative azimuth and altitude angle. The incident beam splits into reflected and transmitted components at the screen. The transmitted component is attenuated as it passes through the screen material by the screen's beam transmittance ($T_{sc}^{dir,dir}$, shown as T_{sc}^{dir} in the figure and equations below) at this incident angle. The reflected (scattered) transmittance of incident solar beam is also shown at this point and will be discussed later in this section. As the attenuated solar beam continues on towards the front glass surface, a portion of the screen-transmitted beam splits at the window surface into transmitted and reflected components. The reflected component reflects off the front surface of the glass material ($T_{sc}^{dir,dir} R_{gl,f}^{dir}$) and the transmitted component continues to travel through the glass material and is further attenuated by the glass beam transmittance. Thus the first term of the combined screen/glass solar beam transmittance is shown as $T_{sc}^{dir,dir} T_{gl}^{dir}$. Interreflections are accounted for by following the beam as it continues to reflect off the front surface of the glass material and the back surface of the screen material. Continuing on with the glass-reflected beam ($T_{sc}^{dir,dir} R_{gl,f}^{dir}$) described above, this beam strikes the back surface of the screen material at the same incident angle as the incident solar beam. This reflected beam is also assumed to be a collimated beam (solid lines) which strikes the back surface of the screen material and reflects as hemispherically-diffuse radiation (dotted lines). The reflective property of the screen material used here is the beam reflectance calculated at the incident solar angle ($R_{sc}^{dir,dif}$). A single ray of this diffuse light will be followed through the remaining steps and represents the energy associated with **all** diffuse rays interreflecting between the screen and glass layers. To determine the second term of the combined screen/glass beam transmittance, the diffusely-reflected ray ($T_{sc}^{dir,dir} R_{gl,f}^{dir} R_{sc}^{dir,dif}$) passes through and is attenuated by the glass layer. Since this ray originates from diffuse reflection, the attenuation of this ray is accounted for using the diffuse transmittance property of the glass. Thus, the second term is shown as $T_{sc}^{dir,dir} R_{gl,f}^{dir} R_{sc}^{dir,dif} T_{gl}^{dif}$. Defining the remaining terms continues in a similar fashion using diffuse properties of both the screen and glass material. Notice that the 3rd and 4th terms shown below are similar to the 2nd term, but additional terms are raised to increasing powers.

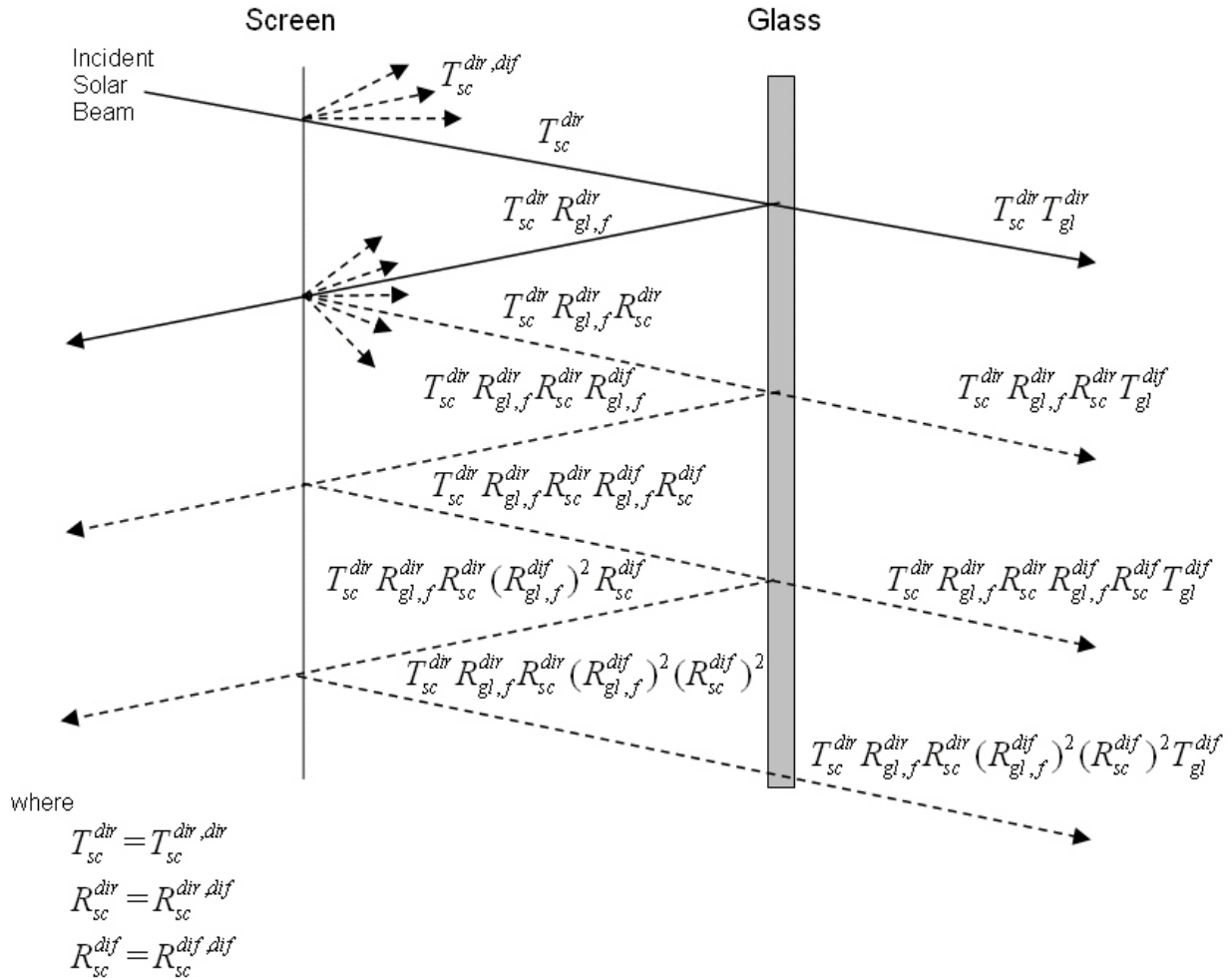


Figure 79. Screen/Glass System Transmittance Equation Schematic.

The screen/glass system transmittance equation shown in the figure above is repeated here in an alternate format to emphasize the recurring nature of the infinite series. This equation represents the final solar beam transmittance equation for the screen/glass combination. The recurring terms are shown as a summation of a quantity raised to the n power, with n ranging from 0 to infinity. Since the quantity $R_{gl,f}^{dif} R_{sc}^{dir,dif}$ is less than 1, the summation

$$\sum_{n=0}^{\infty} (R_{gl,f}^{dif} R_{sc}^{dir,dif})^n \text{ converges and can be expressed as } \frac{1}{1 - R_{gl,f}^{dif} R_{sc}^{dir,dif}}.$$

Since the reflected (scattered) transmittance of incident solar beam ($T_{sc}^{dir,dif}$) and the diffusely reflecting beam $T_{sc}^{dir,dif} R_{gl,f}^{dir} R_{sc}^{dir,dif}$ are both assumed to be hemispherically diffuse radiation, the reflected (scattered) transmittance of incident solar beam is added to the infinite series as shown below.

$$T_{sys}^{dir,all}(\alpha', \varphi', \phi) = T_{sc}^{dir,dif}(\alpha', \varphi') T_{gl}^{dir}(\phi) + (T_{sc}^{dir,dif}(\alpha', \varphi') R_{gl,f}^{dir} R_{sc}^{dir,dif}(\alpha', \varphi') + T_{sc}^{dir,dif}(\alpha', \varphi')) T_{gl}^{dif} \sum_{n=0}^{\infty} (R_{gl,f}^{dif} R_{sc}^{dir,dif})^n$$

— or —

$$T_{sys}^{dir,all}(\alpha', \varphi', \phi) = T_{sc}^{dir,dir}(\alpha', \varphi') T_{gl}^{dir}(\phi) + \frac{(T_{sc}^{dir,dir}(\alpha', \varphi') R_{gl,f}^{dir} R_{sc}^{dir,dif}(\alpha', \varphi') + T_{sc}^{dir,dif}(\alpha', \varphi')) T_{gl}^{dif}}{1 - R_{gl,f}^{dif} R_{sc}^{dif,dif}}$$

where

$$T_{sys}^{dir,all}(\alpha', \varphi', \phi) = \text{screen/glass system beam transmittance (output report variable Screen/Glass System Beam-Beam Solar Transmittance)}$$

Properties for beam absorptance of the individual glass layers and screen/glass combination are derived in a similar fashion to the transmittance calculation described above. Diffuse transmittance and absorptance of individual glass layers and the screen/glass combination are also shown here.

$$A_{gl,j,f}^{dir,sys}(\alpha', \varphi', \phi) = T_{sc}^{dir,dir}(\alpha', \varphi') A_{gl,j,f}^{dir}(\phi) + \frac{(T_{sc}^{dir,dir}(\alpha', \varphi') R_{gl}^{dir}(\phi) R_{sc}^{dir,dif}(\alpha', \varphi') + T_{sc}^{dir,dif}(\alpha', \varphi')) A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} R_{sc}^{dif,dif}}, j = 1, N$$

$$\alpha_{sc}^{dir,sys}(\alpha', \varphi', \phi) = A_{sc}^{dir}(\alpha', \varphi') (1 + R_{gl,f}^{dir}(\phi) T_{sc}^{dir,dir}(\alpha', \varphi')) + \frac{A_{sc}^{dif} R_{gl,f}^{dif}}{1 - R_{sc}^{dif,dif} R_{gl,f}^{dif}} (R_{gl,f}^{dir}(\phi) T_{sc}^{dir,dir}(\alpha', \varphi') R_{sc}^{dir,dif}(\alpha', \varphi'))$$

$$T_{sys}^{dif,dif} = \frac{T_{sc}^{dif,dif} T_{gl}^{dif}}{1 - R_{gl,f}^{dif} R_{sc}^{dif,dif}}$$

$$A_{gl,j,f}^{dif,sys} = \frac{T_{sc}^{dif,dif} A_{gl,j,f}^{dif}}{1 - R_{gl,f}^{dif} R_{sc}^{dif,dif}}, j = 1, N$$

$$\alpha_{sc}^{dif,sys} = A_{sc}^{dif} \left(1 + \frac{T_{sc}^{dif,dif} R_{gl,f}^{dif}}{1 - R_{gl,f}^{dif} R_{sc}^{dif,dif}} \right)$$

where

$$A_{gl,j,f}^{dir,sys}(\alpha', \varphi', \phi) = \text{glass layer beam absorptance including interreflections with screen material}$$

$$\alpha_{sc}^{dir,sys}(\alpha', \varphi', \phi) = \text{beam absorptance of screen material including interreflections with glass}$$

$$T_{sys}^{dif,dif} = \text{screen/glass system diffuse transmittance (output report variable Screen/Glass System Diffuse-Diffuse Solar Transmittance)}$$

$A_{gl,j,f}^{dif,sys}$ = glass layer diffuse absorptance including interreflections with screen material

$\alpha_{sc}^{dif,sys}$ = diffuse absorptance of screen material including interreflections with glass

Screen/Glazing System Properties for Long-Wave Radiation

The program calculates how much long-wave radiation is absorbed by the screen and by the adjacent glass surface. The effective long-wave emissivity (equal to the long-wave absorptance on a wavelength-by-wavelength basis or over the same spectral range) of an exterior screen, taking into account reflection of long-wave radiation between the glass and screen, is given by

$$\varepsilon_{sc}^{lw,eff} = \varepsilon_{sc}^{hw} \left(1 + \frac{T_{sc}^{dif,dif} \rho_{gl}^{hw}}{1 - R_{sc}^{dif,dif} \rho_{gl}^{hw}} \right)$$

where ρ_{gl}^{hw} is the long-wave reflectance of the outermost glass surface facing an exterior screen, and it is assumed that the long-wave transmittance of the glass is zero.

The effective outermost (for exterior screen) glass surface emissivity when the screen is present is

$$\varepsilon_{gl}^{lw,eff} = \varepsilon_{gl}^{hw} \frac{T_{sc}^{dif,dif}}{1 - R_{sc}^{dif,dif} \rho_{gl}^{hw}}$$

The effective inside surface emissivity is the sum of the effective screen and effective glass emissivities:

$$\varepsilon_{ins}^{lw,eff} = \varepsilon_{sc}^{lw,eff} + \varepsilon_{gl}^{lw,eff}$$

The effective temperature of the screen/glazing combination that is used to calculate the window's contribution to the zone's mean radiant temperature (MRT) is given by

$$T^{eff} = \frac{\varepsilon_{sc}^{lw,eff} T_{sc} + \varepsilon_{gl}^{lw,eff} T_{gl}}{\varepsilon_{sc}^{lw,eff} + \varepsilon_{gl}^{lw,eff}}$$

Solar Radiation Transmitted and Absorbed by a Window/Screen System

Let the direct solar incident on the window be

$$I_{dir,inc} = f_{sunlit} I_{dir,norm} \cos \phi \quad (W / m^2)$$

where f_{sunlit} is the fraction of the window that is sunlit (determined by the shadowing calculations), $I_{dir,norm}$ is the direct normal solar irradiance, and ϕ is the angle of incidence.

Let $I_{sky,inc}$ be the irradiance on the window due to diffuse solar radiation from the sky (W/m^2) and let $I_{gnd,inc}$ be the irradiance on the window due to diffuse solar radiation from the ground (W/m^2).

Then we have the following expressions for different classes of transmitted and absorbed solar radiation for the window/screen system, all in W/m^2 :

Direct and diffuse solar entering zone from incident direct solar:

$$I_{dir,inc} T_{sys}^{dir,all}(\alpha', \varphi')$$

Direct solar absorbed by screen:

$$I_{dir,inc} \alpha_{sc}^{dir,sys}(\alpha', \varphi')$$

Direct solar absorbed by glass layers:

$$I_{dir,inc} A_{gl,j,f}^{dir,sys}(\phi, \phi_s), \quad j = 1, N$$

Diffuse solar entering zone from incident diffuse solar:

$$(I_{sky,inc} + I_{gnd,inc}) T_{sys}^{dif,dif}$$

Diffuse solar absorbed by screen:

$$(I_{sky,inc} + I_{gnd,inc}) \alpha_{sc}^{dif,sys}$$

Diffuse solar absorbed by glass layers:

$$(I_{sky,inc} + I_{gnd,inc}) A_{gl,j,f}^{dif,sys}, \quad j = 1, N$$

Window Heat Balance Calculation

Table 30. Fortran Variables used in Window Heat Balance Calculations

Mathematical variable	Description	Units	FORTTRAN variable
N	Number of glass layers	-	nlayer
σ	Stefan-Boltzmann constant		sigma
ϵ_i	Emissivity of face i	-	emis
k_i	Conductance of glass layer i	W/m^2-K	scon
h_o, h_i	Outside, inside air film convective conductance	W/m^2-K	hcout, hcout
h_j	Conductance of gap j	W/m^2-K	hgap
T_o, T_i	Outdoor and indoor air temperatures	K	tout, tin
E_o, E_i	Exterior, interior long-wave radiation incident on window	W/m^2	outir, rmir
θ_i	Temperature of face i	K	thetas

S_i	Radiation (short-wave, and long-wave from zone internal sources) absorbed by face i	W/m^2	AbsRadGlassFace
I_{bm}^{ext}	Exterior beam normal solar irradiance	W/m^2	BeamSolarRad
I_{dif}^{ext}	Exterior diffuse solar irradiance on glazing	W/m^2	-
I_{sw}^{int}	Interior short-wave radiation (from lights and from reflected diffuse solar) incident on glazing from inside	W/m^2	QS
I_{lw}^{int}	Long-wave radiation from lights and equipment incident on glazing from inside	W/m^2	QL
φ	Angle of incidence	radians	-
A_j^f	Front beam solar absorptance of glass layer j	-	-
$A_j^{f,dif}, A_j^{b,dif}$	Front and back diffuse solar absorptance of glass layer j	-	AbsDiff, AbsDiffBack
A, B	Matrices used to solve glazing heat balance equations	W/m^2 , W/m^2-K	Aface, Bface
$h_{r,i}$	Radiative conductance for face i	W/m^2-K	hr(i)
$\Delta\theta_i$	Difference in temperature of face i between successive iterations	K	-

The Glazing Heat Balance Equations

The window glass face temperatures are determined by solving the heat balance equations on each face every time step. For a window with N glass layers there are $2N$ faces and therefore $2N$ equations to solve. Figure 80 shows the variables used for double glazing ($N=2$).

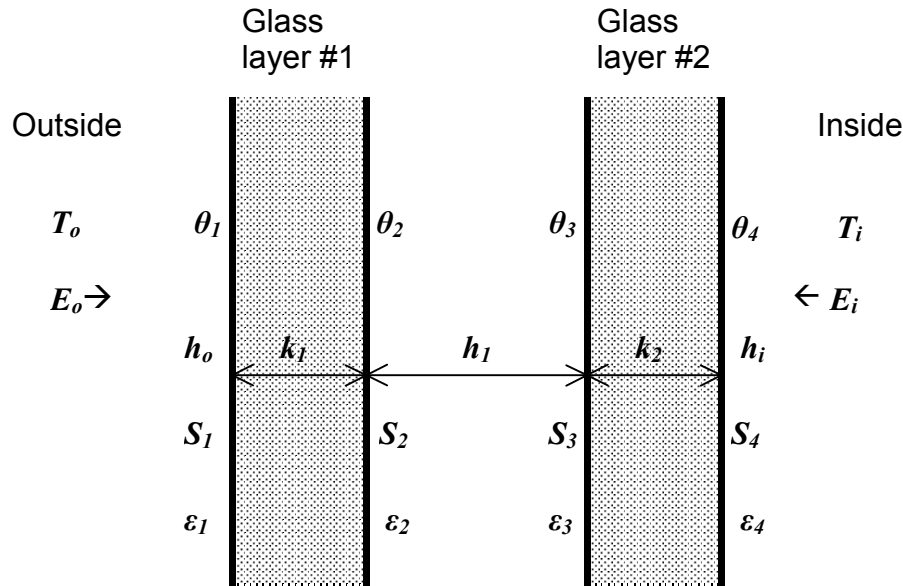


Figure 80. Glazing system with two glass layers showing variables used in heat balance equations.

The following assumptions are made in deriving the heat balance equations:

- 1) The glass layers are thin enough (a few millimeters) that heat storage in the glass can be neglected; therefore, there are no heat capacity terms in the equations.
- 2) The heat flow is perpendicular to the glass faces and is one dimensional. See "Edge of Glass Corrections," below, for adjustments to the gap conduction in multi-pane glazing to account for 2-D conduction effects across the pane separators at the boundaries of the glazing.
- 3) The glass layers are opaque to IR. This is true for most glass products. For thin plastic suspended films this is not a good assumption, so the heat balance equations would have to be modified to handle this case.
- 4) The glass faces are isothermal. This is generally a good assumption since glass conductivity is very high.
- 5) The short wave radiation absorbed in a glass layer can be apportioned equally to the two faces of the layer.

The four equations for double-glazing are as follows. The equations for single glazing ($N=1$) and for $N=3$ and $N=4$ are analogous and are not shown.

$$E_o \varepsilon_1 - \varepsilon_1 \sigma \theta_1^4 + k_1 (\theta_2 - \theta_1) + h_o (T_o - \theta_1) + S_1 = 0 \quad (183)$$

$$k_1 (\theta_1 - \theta_2) + h_1 (\theta_3 - \theta_2) + \sigma \frac{\varepsilon_2 \varepsilon_3}{1 - (1 - \varepsilon_2)(1 - \varepsilon_3)} (\theta_3^4 - \theta_2^4) + S_2 = 0 \quad (184)$$

$$h_1 (\theta_2 - \theta_3) + k_2 (\theta_4 - \theta_3) + \sigma \frac{\varepsilon_2 \varepsilon_3}{1 - (1 - \varepsilon_2)(1 - \varepsilon_3)} (\theta_2^4 - \theta_3^4) + S_3 = 0 \quad (185)$$

$$E_i \varepsilon_4 - \varepsilon_4 \sigma \theta_4^4 + k_2 (\theta_3 - \theta_4) + h_i (T_i - \theta_4) + S_4 = 0 \quad (186)$$

Absorbed Radiation

S_i in Equations (183) to (186) is the radiation (short-wave and long-wave from zone lights and equipment) absorbed on the i^{th} face. Short-wave radiation (solar and short-wave from lights) is assumed to be absorbed uniformly along a glass layer, so for the purposes of the heat balance calculation it is split equally between the two faces of a layer. Glass layers are assumed to be opaque to IR so that the thermal radiation from lights and equipment is assigned only to the inside (room-side) face of the inside glass layer. For N glass layers S_i is given by

$$S_{2j-1} = S_{2j} = \frac{1}{2} \left(I_{bm}^{ext} \cos \phi A_j^f(\phi) + I_{dif}^{ext} A_j^{f,dif} + I_{sw}^{int} A_j^{b,dif} \right), \quad j = 1 \text{ to } N$$

$$S_{2N} = S_{2N} + \varepsilon_{2N} I_{lw}^{int}$$

Here

I_{bm}^{ext} = exterior beam normal solar irradiance

I_{dif}^{ext} = exterior diffuse solar incident on glazing from outside

I_{sw}^{int} = interior short-wave radiation (from lights and from reflected diffuse solar) incident on glazing from inside

I_{lw}^{int} = long-wave radiation from lights and equipment incident on glazing from inside

ε_{2N} = emissivity (thermal absorptance) of the room-side face of the inside glass layer

Room-Side Convection

The correlation for room-side convection coefficient, h_i , is from ISO 15099 section 8.3.2.2. (Prior to EnergyPlus version 3.1, the value for h_i was modeled using the “Detailed” algorithm for opaque surface heat transfer, e.g. for a vertical surface $h_i = 1.31 |\Delta T|^{1/3}$; see section Detailed Natural Convection Algorithm). The ISO 15099 correlation is for still room air and is determined in terms of the Nusselt number, Nu , where

$$h_i = Nu \left(\frac{\lambda}{H} \right)$$

where,

λ is the thermal conductivity of air, and

H is the height of the window.

The Rayleigh number based on height, Ra_H , is calculated using,

$$Ra_H = \frac{\rho^2 H^3 g c_p |T_{surf,i} - T_{air}|}{T_{m,f} \mu \lambda}$$

where,

ρ is the density of air

g is the acceleration due to gravity,

c_p is the specific heat of air,

μ is the dynamic viscosity of air, and

$T_{m,f}$ is the mean film temperature in Kelvin given by,

$$T_{m,f} = T_{air} + \frac{1}{4}(T_{surf,i} - T_{air})$$

There are four cases for the Nusselt correlation that vary by the tilt angle in degrees, γ , and are based on heating conditions. For cooling conditions (where $T_{surf,i} > T_{air}$) the tilt angle is complemented so that $\gamma = 180 - \gamma$

Case A. $0^\circ \leq \gamma < 15^\circ$

$$Nu = 0.13Ra_H^{1/3}$$

Case B. $15^\circ \leq \gamma \leq 90^\circ$

$$Ra_{cv} = 2.5 \times 10^5 \left(\frac{e^{0.72\gamma}}{\sin \lambda} \right)^{1/5}$$

$$Nu = 0.56(Ra_H \sin \gamma)^{1/4}; \text{ for } Ra_H \leq Ra_{cv}$$

$$Nu = 0.13 \left(Ra_H^{1/3} - Ra_{cv}^{1/3} \right) + 0.56(Ra_{cv} \sin \gamma)^{1/4}; Ra_H > Ra_{cv}$$

Case C. $90^\circ < \gamma \leq 179^\circ$

$$Nu = 0.56(Ra_H \sin \gamma)^{1/4}; 10^5 \leq Ra_H \sin \gamma < 10^{11}$$

Case D. $179^\circ < \gamma \leq 180^\circ$

$$Nu = 0.58Ra_H^{1/5}; Ra_H \leq 10^{11}$$

The material properties are evaluated at the mean film temperature. Standard EnergyPlus psychrometric functions are used for ρ and c_p . Thermal conductivity is calculated using,

$$\lambda = 2.873 \times 10^{-3} + 7.76 \times 10^{-8} T_{m,f}$$

Kinematic viscosity is calculated using,

$$\mu = 3.723 \times 10^{-6} + 4.94 \times 10^{-8} T_{m,f}$$

This correlation depends on the surface temperature of the room-side glazing surface and is therefore included inside the window heat balance iteration loop.

Solving the Glazing Heat Balance Equations

The equations are solved as follows:

1) Linearize the equations by defining $h_{r,i} = \varepsilon_i \sigma \theta_i^3$. For example, Equation (183) becomes

$$E_o \varepsilon_1 - h_{r,1} \theta_1 + k_1 (\theta_2 - \theta_1) + h_o (T_o - \theta_1) + S_1 = 0$$

- 2) Write the equations in the matrix form $A\theta = B$
- 3) Use previous time step's values of θ_i as initial values for the current time step. For the first time step of a design day or run period the initial values are estimated by treating the layers as a simple RC network.
- 4) Save the θ_i for use in the next iteration: $\theta_{prev,i} = \theta_i$
- 5) Using θ_{2N} , reevaluate the room-side face surface convection coefficient h_i
- 6) Using the θ_i to evaluate the radiative conductances $h_{r,i}$
- 7) Find the solution $\theta = A^{-1}B$ by LU decomposition
- 8) Perform relaxation on the the new θ_i : $\theta_i \rightarrow (\theta_i + \theta_{prev,i})/2$
- 9) Go to step 4

Repeat steps 4 to 9 until the difference, $\Delta\theta_i$, between values of the θ_i in successive iterations is less than some tolerance value. Currently, the test is

$$\frac{1}{2N} \sum_{i=1}^{2N} |\Delta\theta_i| < 0.02K$$

If this test does not pass after 100 iterations, the tolerance is increased to 0.2K. If the test still fails the program stops and an error message is issued.

The value of the inside face temperature, θ_{2N} , determined in this way participates in the zone heat balance solution (see Outdoor/Exterior Convection) and thermal comfort calculation (see Occupant Thermal Comfort).

Edge-Of-Glass Effects

Table 31. Fortran Variables used in Edge of Glass calculations

Mathematical variable	Description	Units	FORTTRAN variable
\bar{h}	Area-weighted net conductance of glazing including edge-of-glass effects	W/m ² -K	-
A_{cg}	Area of center-of-glass region	m ²	CenterGIArea
A_{fe}	Area of frame edge region	m ²	FrameEdgeArea
A_{de}	Area of divider edge region	m ²	DividerEdgeArea
A_{tot}	Total glazing area	m ²	Surface%Area
h_{cg}	Conductance of center-of-glass region (without air films)	W/m ² -K	-
h_{fe}	Conductance of frame edge region (without air films)	W/m ² -K	-
h_{de}	Conductance of divider edge region (without air films)	W/m ² -K	-

h_{ck}	Convective conductance of gap k	W/m ² -K	-
h_{rk}	Radiative conductance of gap k	W/m ² -K	-
η	Area ratio	-	-
α	Conductance ratio	-	FrEdgeToCenterGIcondRatio, DivEdgeToCenterGIcondRatio

Because of thermal bridging across the spacer separating the glass layers in multi-pane glazing, the conductance of the glazing near the frame and divider, where the spacers are located, is higher than it is in the center of the glass. The area-weighted net conductance (without inside and outside air films) of the glazing in this case can be written

$$\bar{h} = (A_{cg}h_{cg} + A_{fe}h_{fe} + A_{de}h_{de}) / A_{tot} \quad (187)$$

where

h_{cg} = conductance of center-of-glass region (without air films)

h_{fe} = conductance of frame edge region (without air films)

h_{de} = conductance of divider edge region (without air films)

A_{cg} = area of center-of-glass region

A_{fe} = area of frame edge region

A_{de} = area of divider edge region

A_{tot} = total glazing area = $A_{cg} + A_{fe} + A_{de}$

The different regions are shown in Figure 81:

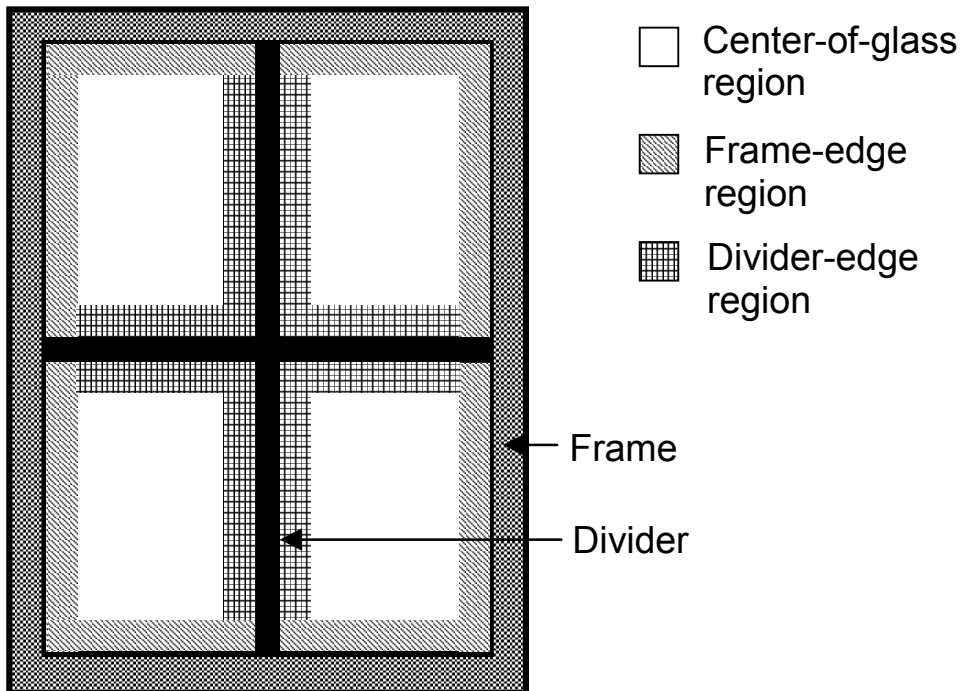


Figure 81: Different types of glass regions.

Equation (187) can be rewritten as

$$\bar{h} = h_{cg} (\eta_{cg} + \alpha_{fe} \eta_{fe} + \alpha_{de} \eta_{de}) \quad (188)$$

where

$$\eta_{cg} = A_{cg} / A_{tot}$$

$$\eta_{fe} = A_{fe} / A_{tot}$$

$$\eta_{de} = A_{de} / A_{tot}$$

$$\alpha_{fe} = h_{fe} / h_{cg}$$

$$\alpha_{de} = h_{de} / h_{cg}$$

The conductance ratios α_{fe} and α_{de} are user inputs obtained from Window 5. They depend on the glazing construction as well as the spacer type, gap width, and frame and divider type. In the EnergyPlus glazing heat balance calculation effective gap convective conductances are used to account for the edge-of-glass effects. These effective conductances are determined as follows for the case with two gaps (triple glazing). The approach for other numbers of gaps is analogous.

Neglecting the very small resistance of the glass layers, the center-of-glass conductance (without inside and outside air films) can be written as

$$h_{cg} = \left((h_{r,1} + h_{c,1})^{-1} + (h_{r,2} + h_{c,2})^{-1} \right)^{-1}$$

where

$h_{c,k}$ = convective conductance of the k^{th} gap

$h_{r,k}$ = radiative conductance of the k^{th} gap

$$= \frac{1}{2} \sigma \frac{\varepsilon_i \varepsilon_j}{1 - (1 - \varepsilon_i)(1 - \varepsilon_j)} (\theta_i + \theta_j)^3$$

$\varepsilon_i, \varepsilon_j$ = emissivity of the faces bounding the gap

θ_i, θ_j = temperature of faces bounding the gap (K)

Equation (188) then becomes

$$\bar{h} = (\eta_{cg} + \alpha_{fe} \eta_{fe} + \alpha_{de} \eta_{de}) \left((h_{r,1} + h_{c,1})^{-1} + (h_{r,2} + h_{c,2})^{-1} \right)^{-1} \quad (189)$$

We can also write \bar{h} in terms of effective convective conductances of the gaps as

$$\bar{h} = \left((h_{r,1} + \bar{h}_{c,1})^{-1} + (h_{r,2} + \bar{h}_{c,2})^{-1} \right)^{-1} \quad (190)$$

Comparing Eqs. (189) and (190) we obtain

$$h_{r,k} + \bar{h}_{c,k} = (\eta_{cg} + \alpha_{fe}\eta_{fe} + \alpha_{de}\eta_{de})(h_{r,k} + h_{c,k})$$

Using $\eta_{cg} = 1 - \eta_{fe} - \eta_{de}$ gives

$$\bar{h}_{c,k} = h_{r,k} (\eta_{fe}(\alpha_{fe} - 1) + \eta_{de}(\alpha_{de} - 1)) + h_{c,k} (1 + \eta_{fe}(\alpha_{fe} - 1) + \eta_{de}(\alpha_{de} - 1))$$

This is the expression used by EnergyPlus for the gap convective conductance when a frame or divider is present.

Apportioning of Absorbed Short-Wave Radiation in Shading Device Layers

If a shading device has a non-zero short-wave transmittance then absorption takes place throughout the shading device layer. The following algorithm is used to apportion the absorbed short-wave radiation to the two faces of the layer. Here f_1 is the fraction assigned to the face closest to the incident radiation and f_2 is the fraction assigned to the face furthest from the incident radiation.

$$f_1 = 1, f_2 = 0 \quad \text{if } \tau_{sh} = 0$$

Otherwise

$$f_1 = 0, f_2 = 0 \quad \text{if } \alpha_{sh} \leq 0.01$$

$$f_1 = 1, f_2 = 0 \quad \text{if } \alpha_{sh} > 0.999$$

$$\left. \begin{aligned} f_1 &= \frac{1 - e^{\frac{1}{2} \ln(1 - \alpha_{sh})}}{\alpha_{sh}} \\ f_2 &= 1 - f_1 \end{aligned} \right\} 0.01 < \alpha_{sh} \leq 0.999$$

Window Frame and Divider Calculation

For the zone heat balance calculation the inside surface temperature of the frame and that of the divider are needed. These temperatures are determined by solving the heat balance equations on the inside and outside surfaces of the frame and divider.

Table 32. Fortran Variables used in Window/Frame and Divider calculations

Mathematical variable	Description	Units	FORTRAN variable
$Q_{\text{ExtIR,abs}}$	IR from the exterior surround absorbed by outside frame surfaces	W	-
$Q_{\text{IR,emitted}}$	IR emitted by outside frame surfaces	W	-
Q_{conv}	Convection from outside air to outside frame surfaces	W	-
Q_{cond}	Conduction through frame from inside frame surfaces to outside frame surfaces	W	-
Q_{abs}	Solar radiation plus outside glass IR absorbed by outside of frame	W	-

$Q_{\text{abs,sol}}^{\text{dif}}$	Diffuse solar absorbed by outside frame surfaces, per unit frame face area	W/ m ²	-
$Q_{\text{abs,sol}}^{\text{bm}}$	Beam solar absorbed by outside frame surfaces, per unit frame face area	W/ m ²	-
$I_{\text{ext}}^{\text{dif}}$	Diffuse solar incident on window	W/ m ²	-
$I_{\text{ext}}^{\text{bm}}$	Direct normal solar irradiance	W/ m ²	-
$\alpha_{\text{sol}}^{\text{fr}}$	Solar absorptance of frame	-	FrameSolAbsorp
$R_{\text{gl}}^{\text{f,dif}}$	Front diffuse solar reflectance of glazing	-	
$R_{\text{gl}}^{\text{f,bm}}$	Front beam solar reflectance of glazing	-	
$\cos(\beta_{\text{face}})$	Cosine of angle of incidence of beam solar on frame outside face		CosIncAng
$\cos(\beta_{\text{h}})$	Cosine of angle of incidence of beam solar on frame projection parallel to window x-axis	-	CosIncAngHorProj
$\cos(\beta_{\text{v}})$	Cosine of angle of incidence of beam solar on frame projection parallel to window y-axis	-	CosIncAngVertProj
f_{sunlit}	Fraction of window that is sunlit	-	SunlitFrac
A_{f}	Area of frame's outside face (same as area of frame's inside face)	m ²	-
$A_{\text{p1}}, A_{\text{p2}}$	Area of frame's outside and inside projection faces	m ²	-
F_{f}	Form factor of frame's outside or inside face for IR	-	-
$F_{\text{p1}}, F_{\text{p2}}$	Form factor of frame outside projection for exterior IR; form factor of frame inside projection for interior IR	-	-
E_{o}	Exterior IR incident on window plane	W/m ²	outir
E_{i}	Interior IR incident on window plane	W/m ²	SurroundIRfromParentZone
ϵ_1, ϵ_2	Outside, inside frame surface emissivity	-	FrameEmis
θ_1, θ_2	Frame outside, inside surface temperature	K	FrameTempSurfOut, FrameTempSurfIn
$T_{\text{o}}, T_{\text{i}}$	Outdoor and indoor air temperatures	K	tout, tin

$h_{o,c}, h_{i,c}$	Frame outside and inside air film convective conductance	W/m ² -K	HOutConv, HInConv
k	Effective inside-surface to outside-surface conductance of frame per unit area of frame projected onto window plane	W/m ² -K	FrameConductance, FrameCon
S_1	Q_{abs}/A_f	W/m ² -K	FrameQRadOutAbs
S_2	Interior short-wave radiation plus interior IR from internal sources absorbed by inside of frame divided by A_f	W/m ² -K	FrameQRadInAbs
η_1, η_2	$A_{p1}/A_f, A_{p2}/A_f$	-	-
H	Height of glazed portion of window	m	Surface%Height
W	Width of glazed portion of window	m	Surface%Width
w_f, w_d	Frame width, divider width	m	FrameWidth, DividerWidth
p_{r1}, p_{r2}	Frame outside, inside projection	m	FrameProjectionOut, FrameProjectionIn
N_h, N_v	Number of horizontal, vertical dividers	-	HorDividers, VertDividers
$T_{o,r}, T_{i,r}$	Frame outside, inside radiative temperature	K	TOutRadFr, TInRadFr
$h_{o,r}, h_{i,r}$	Frame outside, inside surface radiative conductance	W/m ² -K	HOutRad, HInRad
A	Intermediate variable in frame heat balance solution	K	Afac
C	Intermediate variable in frame heat balance solution	-	Efac
B, D	Intermediate variables in frame heat balance solution	-	Bfac, Dfac

Frame Temperature Calculation

Figure 82 shows a cross section through a window showing frame and divider. The outside and inside frame and divider surfaces are assumed to be isothermal. The frame and divider profiles are approximated as rectangular since this simplifies calculating heat gains and losses (see "Error Due to Assuming a Rectangular Profile," below).

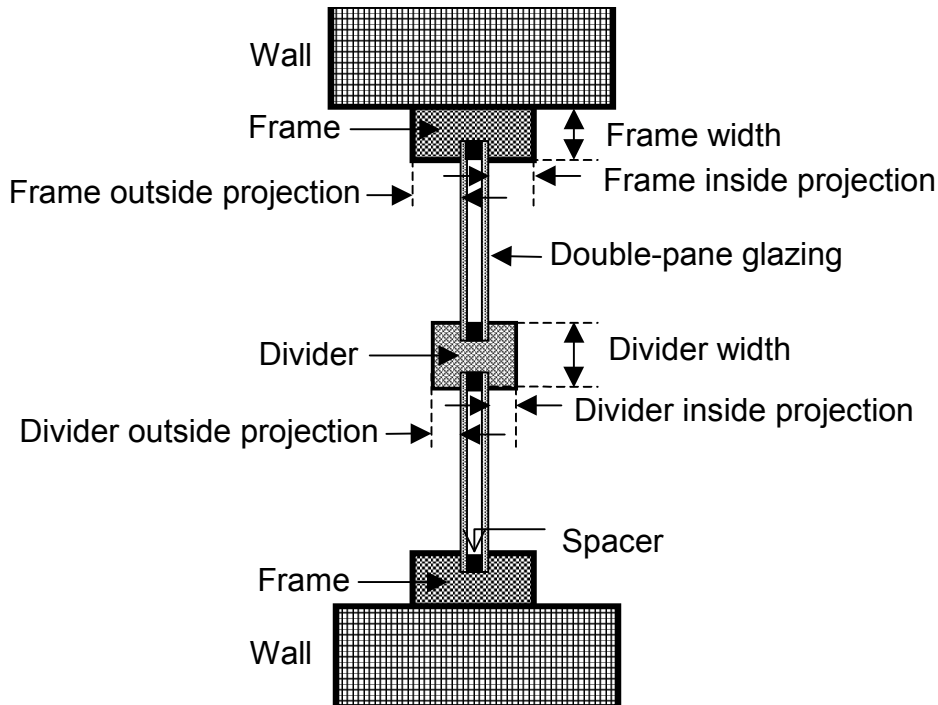


Figure 82. Cross section through a window showing frame and divider (exaggerated horizontally).

Frame Outside Surface Heat Balance

The outside surface heat balance equation is

$$Q_{ExtIR,abs} - Q_{IR,emitted} + Q_{conv} + Q_{cond} + Q_{abs} = 0$$

where

$Q_{ExtIR,abs}$ = IR from the exterior surround (sky and ground) absorbed by outside frame surfaces

$Q_{IR,emitted}$ = IR emitted by outside frame surfaces

Q_{conv} = convection from outside air to outside frame surfaces

Q_{cond} = conduction through frame from inside frame surfaces to outside frame surfaces

Q_{abs} = solar radiation (from sun, sky and ground) plus IR from outside window surface absorbed by outside frame surfaces (see "Calculation of Absorbed Solar Radiation," below).

The first term can be written as the sum of the exterior IR absorbed by the outside face of the frame and the exterior IR absorbed by the frame's outside projection surfaces.

$$Q_{ExtIR,abs} = \varepsilon_1 E_o A_f F_f + \varepsilon_1 E_o A_{p1} F_{p1}$$

where ε_1 is the outside surface emissivity.

The exterior IR incident on the plane of the window, E_o , is the sum of the IR from the sky, ground and obstructions. For the purposes of the frame heat balance calculation it is assumed to be isotropic. For isotropic incident IR, $F_f = 1.0$ and $F_{p1} = 0.5$, which gives

$$Q_{ExtIR,abs} = \varepsilon_1 E_o (A_f + \frac{1}{2} A_{p1})$$

The IR emitted by the outside frame surfaces is

$$Q_{ExtIR,emitted} = \varepsilon_1 \sigma (A_f + A_{p1}) \theta_1^4$$

The convective heat flow from the outside air to the outside frame surfaces is

$$Q_{conv} = h_{o,c} (A_f + A_{p1}) (T_o - \theta_1)$$

The conduction through the frame from inside to outside is

$$Q_{cond} = k A_f (\theta_2 - \theta_1)$$

Note that A_f is used here since the conductance, k , is, by definition, per unit area of frame projected onto the plane of the window.

Adding these expressions for the Q terms and dividing by A_f gives

$$E_o \varepsilon_1 \left(1 + \frac{1}{2} \eta_1 \right) - \varepsilon_1 (1 + \eta_1) \theta_1^4 + h_{o,c} (1 + \eta_1) (T_o - \theta_1) + k (\theta_2 - \theta_1) + S_1 = 0 \quad (191)$$

where $S_1 = Q_{abs}/A_f$ and

$$\eta_1 = \frac{A_{p1}}{A_f} = \left(\frac{p_{f,1}}{w_f} \right) \frac{H + W - (N_h + N_v) w_d}{H + W + 2w_f}$$

We linearize Eq. (191) as follows.

Write the first two terms as

$$\varepsilon_1 (1 + \eta_1) \left[E_o \left(1 + \frac{1}{2} \eta_1 \right) / (1 + \eta_1) - \theta_1^4 \right]$$

and define a radiative temperature

$$T_{o,r} = \left[E_o \left(1 + \frac{1}{2} \eta_1 \right) / (1 + \eta_1) \right]^{1/4}$$

This gives

$$\varepsilon_1 (1 + \eta_1) [T_{o,r}^4 - \theta_1^4]$$

which, within a few percent, equals

$$\varepsilon_1 (1 + \eta_1) \frac{(T_{o,r} + \theta_1)^3}{2} (T_{o,r} - \theta_1)$$

Defining an outside surface radiative conductance as follows

$$h_{o,r} = \varepsilon_1(1 + \eta_1) \frac{(T_{o,r} + \theta_1)^3}{2}$$

then gives

$$h_{o,r}(T_{o,r} - \theta_1)$$

The final outside surface heat balance equation in linearized form is then

$$h_{o,r}(T_{o,r} - \theta_1) + h_{o,c}(1 + \eta_1)(T_o - \theta_1) + k(\theta_2 - \theta_1) + S_1 = 0 \quad (192)$$

Frame Inside Surface Heat Balance

A similar approach can be used to obtain the following linearized *inside* surface heat balance equation:

$$h_{i,r}(T_{i,r} - \theta_2) + h_{i,c}(1 + \eta_2)(T_i - \theta_2) + k(\theta_1 - \theta_2) + S_2 = 0 \quad (193)$$

where

$$T_{i,r} = [E_i(1 + \frac{1}{2}\eta_2)/(1 + \eta_2)]^{1/4}$$

$$\eta_2 = \frac{A_{p2}}{A_f} = \left(\frac{p_{f,2}}{w_f} \right) \frac{H + W - (N_h + N_v)w_d}{H + W + 2w_f}$$

and E_i is the interior IR irradiance incident on the plane of the window.

Solving Eqs. (192) and (193) simultaneously gives

$$\theta_2 = \frac{D + CA}{1 - CB}$$

with

$$A = \frac{h_{o,r}T_{o,r} + h_{o,c}T_o + S_1}{h_{o,r} + k + h_{o,c}}$$

$$B = \frac{k}{h_{o,r} + k + h_{o,c}}$$

$$C = \frac{k}{h_{i,r} + k + h_{i,c}}$$

$$D = \frac{h_{i,r}T_{i,r} + h_{i,c}T_i + S_2}{h_{i,r} + k + h_{i,c}}$$

Calculation of Solar Radiation Absorbed by Frame

The frame outside face and outside projections and inside projections absorb beam solar radiation (if sunlight is striking the window) and diffuse solar radiation from the sky and ground. For the outside surfaces of the frame, the absorbed diffuse solar per unit frame face area is

$$Q_{abs,sol}^{dif} = I_{ext}^{dif} \alpha_{fr,sol} (A_f + F_{p1} A_{p1}) / A_f = I_{ext}^{dif} \alpha_{fr,sol} \left(1 + 0.5 \frac{A_{p1}}{A_f} \right)$$

If there is no exterior window shade, I_{ext}^{dif} includes the effect of diffuse solar reflecting off of the glazing onto the outside frame projection, i.e.,

$$I_{ext}^{dif} \rightarrow I_{ext}^{dif} (1 + R_{gl}^{f,dif})$$

The beam solar absorbed by the outside face of the frame, per unit frame face area is

$$Q_{abs,sol}^{bm,face} = I_{ext}^{bm} \alpha_{fr,sol} \cos \beta_{face} f_{sunlit}$$

The beam solar absorbed by the frame outside projection parallel to the window x-axis is

$$Q_{abs,sol}^{bm,h} = I_{ext}^{bm} \alpha_{fr,sol} \cos \beta_h p_{f1} (W - N_v w_d) f_{sunlit} / A_f$$

Here it is assumed that the sunlit fraction, f_{sunlit} , for the window can be applied to the window frame. Note that at any given time beam solar can strike only one of the two projection surfaces that are parallel to the window x-axis. If there is no exterior window shade, I_{ext}^{bm} includes the effect of beam solar reflecting off of the glazing onto the outside frame projection, i.e.,

$$I_{ext}^{bm} \rightarrow I_{ext}^{bm} (1 + R_{gl}^{f,bm})$$

The beam solar absorbed by the frame outside projection parallel to the window y-axis is

$$Q_{abs,sol}^{bm,v} = I_{ext}^{bm} \alpha_{fr,sol} \cos \beta_v p_{f1} (H - N_h w_d) f_{sunlit} / A_f$$

Using a similar approach, the beam and diffuse solar absorbed by the *inside* frame projections is calculated, taking the transmittance of the glazing into account.

Error Due to Assuming a Rectangular Profile

Assuming that the inside and outside frame profile is rectangular introduces an error in the surface heat transfer calculation if the profile is non-rectangular. The percent error in the calculation of convection and emitted IR is approximately $100 \left| \frac{L_{profile,rect} - L_{profile,actual}}{L_{profile,rect}} \right|$, where $L_{profile,rect}$ is the profile length for a rectangular profile ($w_f + p_{f1}$ for outside of frame or $w_f + p_{f2}$ for inside of frame) and $L_{profile,actual}$ is the

actual profile length. For example, for a circular profile vs a square profile the error is about 22%. The error in the calculation of absorbed beam radiation is close to zero since the beam radiation intercepted by the profile is insensitive to the shape of the profile. The error in the absorbed diffuse radiation and absorbed IR depends on details of the shape of the profile. For example, for a circular profile vs. a square profile the error is about 15%.

Divider Temperature Calculation

The divider inside and outside surface temperatures are determined by a heat balance calculation that is analogous to the frame heat balance calculation described above.

Beam Solar Reflection from Window Reveal Surfaces

This section describes how beam solar radiation that is reflected from window reveal surfaces is calculated. Reflection from outside reveal surfaces—which are associated with the setback of the glazing from the outside surface of the window's parent wall—increases the solar gain through the glazing. Reflection from inside reveal surfaces—which are associated with the setback of the glazing from the inside surface of the window's parent wall—decreases the solar gain to the zone because some of this radiation is reflected back out of the window.

The amount of beam solar reflected from reveal surfaces depends, among other things, on the extent to which reveal surfaces are shadowed by other reveal surfaces. An example of this shadowing is shown in Figure 83. In this case the sun is positioned such that the top reveal surfaces shadow the left and bottom reveal surfaces. And the right reveal surfaces shadow the bottom reveal surfaces. The result is that the left/outside, bottom/outside, left/inside and bottom/inside reveal surfaces each have sunlit areas. Note that the top and right reveal surfaces are facing away from the sun in this example so their sunlit areas are zero.

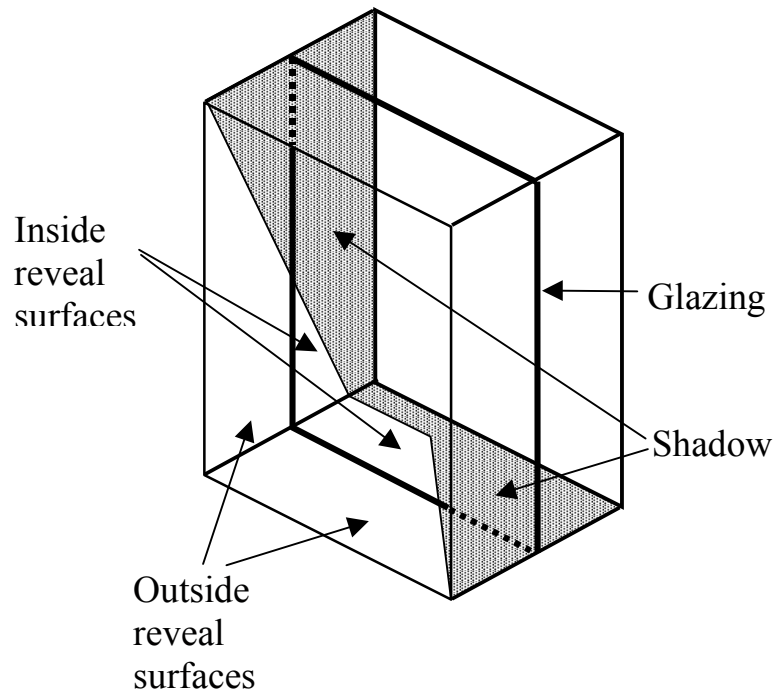


Figure 83. Example of shadowing of reveal surfaces by other reveal surfaces.

The size of the shadowed areas, and the size of the corresponding illuminated areas, depends on the following factors:

- The sun position relative to the window
- The height and width of the window

- The depth of the outside and inside reveal surfaces

We will assume that the reveal surfaces are perpendicular to the window plane and that the window is rectangular. Then the above factors determine a unique shadow pattern. From the geometry of the pattern the shadowed areas and corresponding illuminated areas can be determined. This calculation is done in subroutine CalcBeamSolarReflectedFromWinRevealSurface in the SolarShading module. The window reveal input data is specified in the WindowProperty:FrameAndDivider object expect for the depth of the outside reveal, which is determined from the vertex locations of the window and its parent wall.

If an exterior shading device (shade, screen or blind) is in place it is assumed that it blocks beam solar before it reaches outside or inside reveal surfaces. Correspondingly, it is assumed that an interior or between-glass shading device blocks beam solar before it reaches inside reveal surfaces.

Representative shadow patterns are shown in Figure 84 for a window with no shading device, and without and with a frame. The case with a frame has to be considered separately because the frame can cast an additional shadow on the inside reveal surfaces.

The patterns shown apply to both vertical and horizontal reveal surfaces. It is important to keep in mind that, for a window of arbitrary tilt, if the left reveal surfaces are illuminated the right surfaces will not be, and vice versa. And if the bottom reveal surfaces are illuminated the top surfaces will not be, and vice versa. (Of course, for a vertical window, the top reveal surfaces will never be illuminated by beam solar if the reveal surfaces are perpendicular to the glazing, as is being assumed.

For each shadow pattern in Figure 84, equations are given for the shadowed areas $A_{1,sh}$ and $A_{2,sh}$ of the outside and inside reveal surfaces, respectively. The variables in these equations are the following (see also Figure 85):

d_1 = depth of outside reveal, measured from the outside plane of the glazing to the edge of the reveal, plus one half of the glazing thickness.

d_2 = depth of inside reveal (or, for illumination on bottom reveal surfaces, inside sill depth), measured from the inside plane of the glazing to the edge of the reveal or the sill, plus one half of the glazing thickness.

L = window height for vertical reveal surfaces or window width for horizontal reveal surfaces

α = vertical solar profile angle for shadowing on vertical reveal surfaces or horizontal solar profile angle for shadowing on horizontal reveal surfaces.

$p_1(p_2)$ = distance from outside (inside) surface of frame to glazing midplane.

d_2' = depth of shadow cast by top reveal on bottom reveal, or by left reveal on right reveal, or by right reveal on left reveal.

d_2'' = depth of shadow cast by frame.

For simplicity it is assumed that, for the case without a frame, the shadowed and illuminated areas extend into the glazing region. For this reason, d_1 and d_2 are measured from the midplane of the glazing. For the case with a frame, the beam solar absorbed by the surfaces formed by the frame outside and inside projections perpendicular to the glazing is calculated as described in "Window Frame and Divider Calculation: Calculation of Solar Radiation Absorbed by Frame."

Figure 84. Expression for area of shaded regions for different shadow patterns: (a) window without frame, (b) window with frame

	<p style="text-align: right;">(a)</p> $A_{1,sh} = \frac{1}{2} d_1^2 \tan \alpha$ $A_{2,sh} = \frac{1}{2} (d_1 + d_2)^2 \tan \alpha - A_{1,sh}$
	$A_{1,sh} = \frac{1}{2} d_1^2 \tan \alpha$ $A_{2,sh} = d_2' L + \frac{1}{2} (d_1 + d_2 - d_2')^2 \tan \alpha - A_{1,sh}$
	$A_{1,sh} = (d_2' - d_2) L + \frac{1}{2} (d_1 + d_2 - d_2')^2 \tan \alpha$ $A_{2,sh} = d_2 L$
	$A_{1,sh} = d_1 L - \frac{1}{2} \frac{L^2}{\tan \alpha}$ $A_{2,sh} = d_2 L$

	<p style="text-align: right;">(b)</p> $A_{1,sh} = \frac{1}{2}(d_1 - p_1)^2 \tan \alpha$ $A_{2,sh} = d_2^* L + \frac{1}{2}(d_1 + d_2^*)^2 \tan \alpha - \frac{1}{2}(d_1 + p_1 + d_2^*)^2 \tan \alpha$
	$A_{1,sh} = \frac{1}{2}(d_1 - p_1)^2 \tan \alpha$ $A_{2,sh} = d_2' L + d_2^* L + \frac{1}{2}(d_1 + d_2 - d_2')^2 \tan \alpha - \frac{1}{2}(d_1 + p_1 + d_2^*)^2 \tan \alpha$
	$A_{1,sh} = (d_2' - (d_2 + p_1))L + \frac{1}{2}(d_1 + d_2 - d_2')^2 \tan \alpha$ $A_{2,sh} = (d_2 - p_2)L$
	$A_{1,sh} = (d_1 - p_1)L - \frac{1}{2} \frac{L^2}{\tan \alpha}$ $A_{2,sh} = (d_2 - p_2)L$

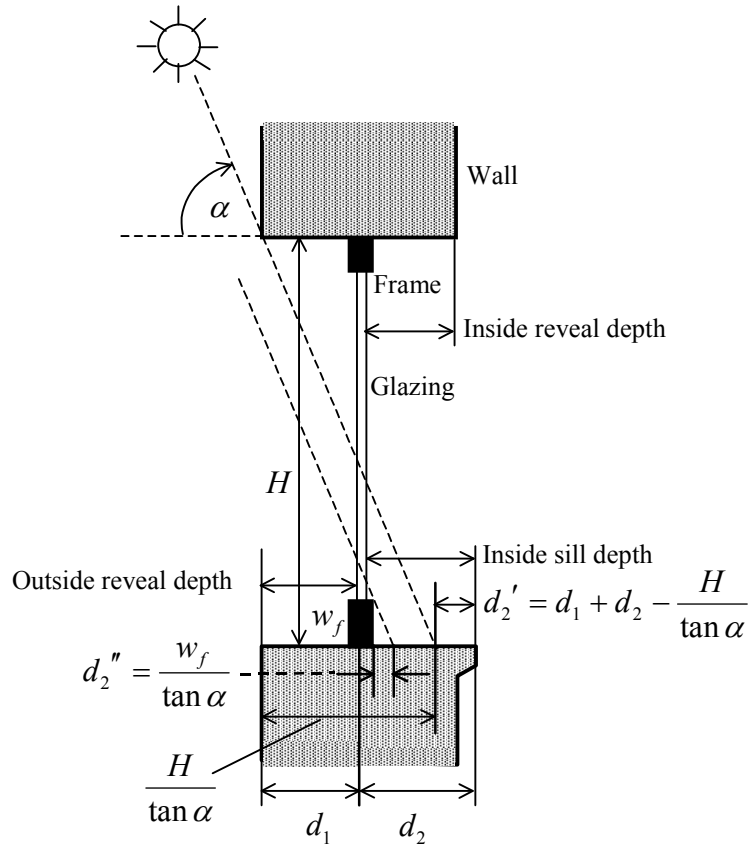


Figure 85. Vertical section through a vertical window with outside and inside reveal showing calculation of the shadows cast by the top reveal onto the inside sill and by the frame onto the inside sill.

The following logic gives expressions for the shadowed areas for all possible shadow patterns. Here:

$$d1 = d_1$$

$$d2 = d_2$$

$$P1 = p_1$$

$$P2 = p_2$$

$$f1 = d_1 - p_1$$

$$f2 = d_2 - p_2$$

$$d2prime = d_2'$$

$$d2prime2 = d_2''$$

$$d12 = d_1 + d_2 - d_2'$$

$$\text{TanAlpha} = \tan \alpha$$

$$A1sh = A_{1,sh}$$

$$A2sh = A_{2,sh}$$

$$L = L$$

L1 = average distance to frame of illuminated area of outside reveal (used to calculate view factor to frame).

L2 = average distance to frame of illuminated area of inside reveal (used to calculate view factor to frame).

```

IF(window does not have a frame) THEN
  IF(d2prime <= d2) THEN
    IF(d12*TanAlpha <= L) THEN
      A1sh = 0.5*TanAlpha*d1**2
      A2sh = d2prime*L + 0.5*TanAlpha*d12**2 - A1sh
    ELSE ! d12*TanAlpha > L
      IF(d1*TanAlpha <= L) THEN
        A1sh = 0.5*TanAlpha*d1**2
        A2sh = d2*L - 0.5*TanAlpha*(L/TanAlpha - d1)**2
      ELSE ! d1*TanAlpha > L
        A1sh = d1*L - (0.5/TanAlpha)*L**2
        A2sh = d2*L
      END IF
    END IF
  ELSE ! d2prime > d2
    A2sh = d2*L
    IF(d2prime < d1+d2) THEN
      IF(d12*TanAlpha <= L) THEN
        A1sh = L*(d2prime-d2) + 0.5*TanAlpha*d12**2
      ELSE ! d12*TanAlpha > L
        A1sh = d1*L - 0.5*L**2/TanAlpha
      END IF
    ELSE ! d2prime >= d1+d2
      A1sh = d1*L
    END IF
  END IF
ELSE ! Window has a frame
  f1 = d1-P1
  f2 = d2-P2
  d2prime2 = FrameWidth/TanGamma
  IF(vertical reveal) THEN ! Vertical reveal
    IF(InsReveal+0.5*GlazingThickness <= P2) d2 = P2 + 0.001
  ELSE ! Horizontal
    IF(bottom reveal surfaces may be illuminated) THEN
      ! Bottom reveal surfaces may be illuminated
      IF(InsSillDepth+0.5*GlazingThickness<=P2) d2= P2 + 0.001
    ELSE
      ! Top reveal surfaces may be illuminated
      IF(InsReveal+0.5*GlazingThickness <= P2) d2 = P2 + 0.001
    END IF
  END IF
  IF(d2prime <= f2) THEN
    ! Shadow from opposing reveal does not go beyond inside
    ! surface of frame
    IF(d12*TanAlpha <= L) THEN
      A1sh = 0.5*TanAlpha*f1**2
      L1 = f1*(f1*TanAlpha/(6*L)+0.5)
      IF(d2-(d2prime+d2prime2+P2) >= 0.) THEN
        A2sh = (d2prime+d2prime2)*L + &
          0.5*TanAlpha*((d1+d2-d2prime)**2-d1+p2+d2prime2)**2)
        L2 = d2prime2 + 0.5*(d2-(d2prime+d2prime2+P2))
      ELSE ! d2-(d2prime+d2prime2+P2) < 0.
        ! Inside reveal is fully shadowed by frame and/or
        !opposing reveal
        A2sh = f2*L
        L2 = f2
      END IF
    END IF
  END IF

```

```

      END IF
      ELSE ! d12*TanAlpha >= L
        IF((d1+P2)*TanAlpha <= L) THEN
          A1sh = 0.5*TanAlpha*f1**2
          L1 = f1*((f1*TanAlpha)/(6*L) + 0.5)
          IF((d1+P2+d2prime2)*TanAlpha >= L) THEN
            A2sh = f2*L
            L2 = f2
          ELSE ! (d1+P2+d2prime2)*TanAlpha < L
            A2sh = f2*L - 0.5*(L-(d1+P2)*TanAlpha)**2/TanAlpha &
              + d2prime2*(L-(d1+P2+d2prime2/2)*TanAlpha)
            L2 = d2prime2 + (L/TanAlpha - (d1+P2+d2prime2))/3
          END IF
        ELSE ! (d1+P2)*TanAlpha > L
          L2 = f2
          A2sh = f2*L
          IF(f1*TanAlpha <= L) THEN
            A1sh = 0.5*TanAlpha*f1**2
            L1 = f1*((f1*TanAlpha)/(6*L) + 0.5)
          ELSE ! f1*TanAlpha > L
            A1sh = f1*L - 0.5*L**2/TanAlpha
            L1 = f1-(L/TanAlpha)/3
          END IF
        END IF
      END IF
    ELSE
      ! d2prime > f2 -- Shadow from opposing reveal goes beyond
      ! inside of frame
      A2sh = f2*L
      L2 = f2
      IF(d2prime >= d1+d2) THEN
        A1sh = 0.0
        L1 = f1
      ELSE ! d2prime < d1+d2
        IF(d2prime <= d2+P1) THEN
          IF(f1*TanAlpha <= L) THEN
            A1sh = 0.5*TanAlpha*f1**2
            L1 = f1*((f1*TanAlpha)/(6*L) + 0.5)
          ELSE ! f1*TanAlpha > L
            A1sh = f1*L - 0.5*L**2/TanAlpha
            L1 = f1 - (L/TanAlpha)/3
          END IF
        ELSE ! d2prime > d2+P1
          IF(d12*TanAlpha <= L) THEN
            A1sh = L*(d2prime-(d2+P1)) + 0.5*TanAlpha*d12**2
            L1 = (L*(f1-d12/2)-d12*TanAlpha* &
              (f1/2-d12/3))/(L-d12*TanAlpha/2)
          ELSE ! d12*TanAlpha > L
            A1sh = f1*L - 0.5*L**2/TanAlpha
            L1 = f1 - (L/TanAlpha)/3
          END IF
        END IF
      END IF
    END IF
    FracToGlassOuts = 0.5*(1.0 - ATAN(FrameWidth/L1)/PiOvr2)
    FracToGlassIns = 0.5*(1.0 - ATAN(FrameWidth/L2)/PiOvr2)
  END IF ! End of check if window has frame

```

The beam solar reflected from a sunlit region of area A is given by

$$R = I_b A \cos \beta (1 - a)$$

where

R = reflected solar radiation [W]

I_B = beam normal irradiance [W/m^2]

A = sunlit area [m^2]

β = beam solar angle of incidence on reveal surface

a = solar absorptance of reveal surface

All reflected radiation is assumed to be isotropic diffuse. For outside reveal surfaces it is assumed that $R/2$ goes toward the window and $R/2$ goes to the exterior environment. Of the portion that goes toward the window a fraction F_1 goes toward the frame, if present, and $1 - F_1$ goes toward the glazing.

The view factor F_1 to the frame calculated by assuming that the illuminated area can be considered to be a line source. Then the area-weighted average distance, L_1 , of the source to the frame is calculated from the shape of the illuminated area (see above pseudo-code). Then F_1 is related as follows to the average angle subtended by the frame of width w_f :

$$F_1 = \frac{\tan^{-1}(w_f / L_1)}{\pi / 2}$$

For the portion going towards the frame, $(R/2)F_1a_f$ is absorbed by the frame (where a_f is the solar absorptance of the frame) and contributes to the frame heat conduction calculation. The rest, $(R/2)F_1(1 - a_f)$, is assumed to be reflected to the exterior environment.

If the glazing has diffuse transmittance τ_{diff} , diffuse front reflectance ρ_{diff}^f , and layer front absorptance $\alpha_{l,diff}^f$, then, of the portion, $(R/2)(1 - F_1)$, that goes toward the glazing, $(R/2)(1 - F_1)\tau_{diff}$ is transmitted to the zone, $(R/2)(1 - F_1)\alpha_{l,diff}^f$ is absorbed in glass layer l and contributes to the glazing heat balance calculation, and $(R/2)(1 - F_1)\rho_{diff}^f$ is reflected to the exterior environment.

The beam solar absorbed by an outside reveal surface is added to the other solar radiation absorbed by the outside of the window's parent wall.

For inside reveal surfaces it is assumed that $R/2$ goes towards the window and $R/2$ goes into the zone. Of the portion that goes toward the window a fraction $(R/2)F_2$ goes toward the frame, if present, and $(R/2)(1 - F_2)$ goes toward the glazing (F_2 is calculated using a method analogous to that used for F_1). For the portion going towards the frame, $(R/2)F_2a_f$ is absorbed by the frame and contributes to the frame heat conduction calculation. The rest, $(R/2)F_2(1 - a_f)$, is assumed to be reflected back into the zone.

If the glazing has diffuse back reflectance ρ_{diff}^b , and layer back absorptance $\alpha_{l,diff}^b$, then, of the portion $(R/2)(1 - F_2)$ that goes toward the glazing, $(R/2)(1 - F_2)\tau_{diff}$ is transmitted

back out the glazing, $(R/2)(1-F_2)\alpha_{i,diff}^b$ is absorbed in glass layer l and contributes to the glazing heat balance calculation, and $(R/2)(1-F_2)\rho_{diff}^b$ is reflected into the zone.

The beam solar absorbed by an inside reveal surface is added to the other solar radiation absorbed by the inside of the window's parent wall.

Shading Device Thermal Model

Shading devices in EnergyPlus can be on the exterior or interior sides of the window or between glass layers. The window shading device thermal model accounts for the thermal interactions between the shading layer (shade, screen or blind) and the adjacent glass, and between the shading layer and the room (for interior shading) or the shading layer and the outside surround (for exterior shading).

An important feature of the shading device thermal model is calculating the natural convection airflow between the shading device and glass. This flow affects the temperature of the shading device and glazing and, for interior shading, is a determinant of the convective heat gain from the shading layer and glazing to the zone air. The airflow model is based on one described in the ISO Standard 15099, "Thermal Performance of Windows, Doors and Shading Devices—Detailed Calculations" [ISO15099, 2001]. (Between-glass forced airflow is also modeled; see "Airflow Windows.")

The following effects are considered by the shading device thermal model:

- For interior and exterior shading device: Long-wave radiation (IR) from the surround absorbed by shading device, or transmitted by the shading device and absorbed by the adjacent glass. For interior shading the surround consists of the other zone surfaces. For exterior shading the surround is the sky and ground plus exterior shadowing surfaces and exterior building surfaces "seen" by the window.
- Inter-reflection of IR between the shading device and adjacent glass.
- Direct and diffuse solar radiation absorbed by the shading device.
- Inter-reflection of solar radiation between shading layer and glass layers.
- Convection from shading layer and glass to the air in the gap (or, for between-glass shading, gaps) between the shading layer and adjacent glass, and convection from interior shading layer to zone air or from exterior shading layer to outside air.
- Natural convection airflow in the gap (or, for between-glass shading, gaps) between shading layer and adjacent glass induced by buoyancy effects, and the effect of this flow on the shading-to-gap and glass-to-gap convection coefficients.
- For interior shading, convective gain (or loss) to zone air from gap airflow.

In the following it is assumed that the shading device, when in place, covers the glazed part of the window (and dividers, if present) and is parallel to the glazing. For interior and exterior shading devices it is assumed that the shading layer is separated from the glazing by an air gap. A between-glass shading layer is assumed to be centered between two glass layers and separated from the adjacent glass layers by gaps that is filled with the same gas. If the window has a frame, it is assumed that the shading device does *not* cover the frame.

Heat Balance Equations for Shading Device and Adjacent Glass

If a window shading device is deployed the heat balance equations for the glass surfaces facing the shading layer are modified, and two new equations, one for each face of the shading layer, are added. Figure 86 illustrates the case of double glazing with an interior shading device.

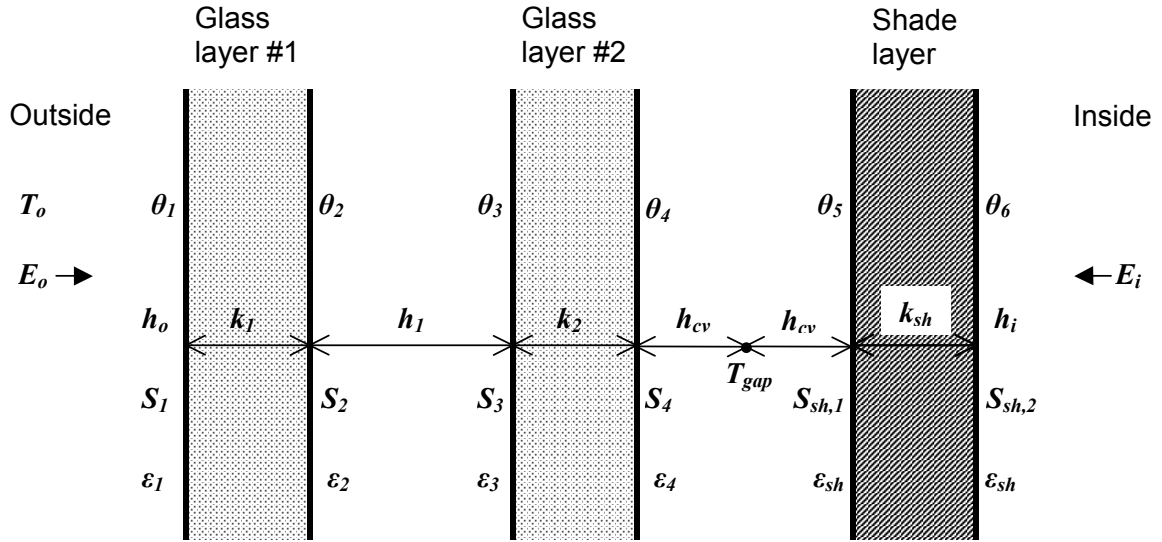


Figure 86. Glazing system with two glass layers and an interior shading layer showing variables used in heat balance equations.

The heat balance equation for the glass surface facing the gap between glass and shading layer (called in the following, "gap") is

$$\frac{E_i \epsilon_4 \tau_{sh}}{1 - \rho_4 \rho_{sh}} + \frac{\sigma \epsilon_4}{1 - \rho_4 \rho_{sh}} \left[\theta_5^4 \epsilon_{sh} - \theta_4^4 (1 - \rho_{sh}) \right] + k_2 (\theta_3 - \theta_4) + h_{cv} (T_{gap} - \theta_4) + S_4 = 0$$

where

τ_{sh} = IR diffuse transmittance of shading device

ϵ_{sh} = diffuse emissivity of shading device

ρ_{sh} = IR diffuse reflectance of shading device ($= 1 - (\tau_{sh} + \epsilon_{sh})$)

θ_5 = temperature of the surface of the shading layer that faces the gap (K).

The term $1 - \rho_4 \rho_{sh}$ accounts for the inter-reflection of IR radiation between glass and shading layer.

The convective heat transfer from glass layer #2 to the air in the gap is

$$q_{c,gl} = h_{cv} (\theta_4 - T_{gap})$$

where

T_{gap} = effective mean temperature of the gap air (K).

h_{cv} = convective heat transfer coefficient from glass or shading layer to gap air (W/m^2K).

The corresponding heat transfer from shading layer to gap air is

$$q_{c,sh} = h_{cv} (\theta_5 - T_{gap})$$

The convective heat transfer coefficient is given by

$$h_{cv} = 2h_c + 4v \quad (194)$$

where

h_c = surface-to-surface heat transfer coefficient for non-vented (closed) cavities (W/m²K)

v = mean air velocity in the gap (m/s).

The quantities h_{cv} and T_{gap} depend on the airflow velocity in the gap, which in turn depends on several factors, including height of shading layer, glass/shading layer separation (gap depth), zone air temperature for interior shading or outside air temperature for exterior shading, and shading layer and glass face temperatures. The calculation of h_{cv} and T_{gap} is described in the following sections.

The heat balance equation for the shading layer surface facing the gap is

$$\frac{E_i \tau_{sh} \rho_4 \varepsilon_{sh}}{1 - \rho_4 \rho_{sh}} + \frac{\sigma \varepsilon_{sh}}{1 - \rho_4 \rho_{sh}} \left[\varepsilon_4 \theta_4^4 - \theta_5^4 (1 - \rho_4 (\varepsilon_{sh} + \rho_{sh})) \right] + k_{sh} (\theta_6 - \theta_5) + h_{cv} (T_{gap} - \theta_5) + S_{sh,1} = 0$$

where

k_{sh} = shading layer conductance (W/m²K).

θ_6 = temperature of shading layer surface facing the zone air (K).

$S_{sh,1}$ = solar radiation plus short-wave radiation from lights plus IR radiation from lights and zone equipment absorbed by the gap-side face of the shading layer (W/m²K).

The heat balance equation for the shading layer surface facing the zone air is

$$E_i \varepsilon_{sh} - \varepsilon_{sh} \sigma \theta_6^4 + k_{sh} (\theta_5 - \theta_6) + h_i (T_i - \theta_6) + S_{sh,2} = 0$$

where

$S_{sh,2}$ = solar radiation plus short-wave radiation from lights plus IR radiation from lights and zone equipment absorbed by the zone-side face of the shading layer (W/m²K).

Solving for Gap Airflow and Temperature

For interior and exterior shading devices a pressure-balance equation is used to determine gap air velocity, gap air mean equivalent temperature and gap outlet air temperature given values of zone air temperature (or outside temperature for exterior shading), shading layer face temperatures and gap geometry. The pressure balance equates the buoyancy pressure acting on the gap air to the pressure losses associated with gap airflow between gap inlet and outlet [ISO15099, 2001]. The variables used in the following analysis of the interior shading case are shown in Figure 87.

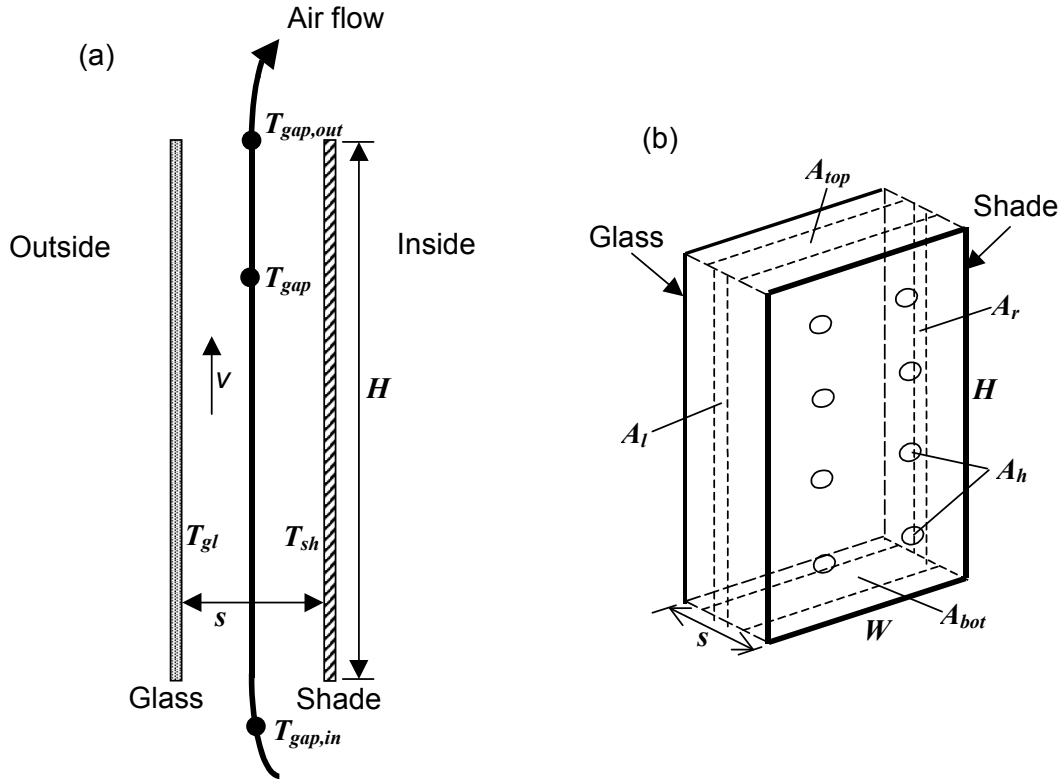


Figure 87. Vertical section (a) and perspective view (b) of glass layer and interior shading layer showing variables used in the gap airflow analysis. The opening areas A_{bot} , A_{top} , A_l , A_r and A_h are shown schematically.

Pressure Balance Equation

The pressure balance equation for airflow through the gap is

$$\Delta p_T = \Delta p_B + \Delta p_{HP} + \Delta p_Z \quad (195)$$

Here, Δp_T is the driving pressure difference between room air and gap air. It is given by

$$\Delta p_T = \rho_0 T_0 g H \sin \phi \frac{|T_{gap} - T_{gap,in}|}{T_{gap} T_{gap,in}}$$

where

ρ_0 = density of air at temperature T_0 (kg/m^3)

T_0 = reference temperature (283K)

g = acceleration due to gravity (m/s^2)

H = height of shading layer (m)

ϕ = tilt angle of window (vertical = 90°)

T_{gap} = effective mean temperature of the gap air (K)

$T_{gap,in}$ = gap inlet temperature (= zone air temperature for interior shading) (K)

The Δp_B term is due to the acceleration of air to velocity v (Bernoulli's law). It is given by

$$\Delta p_B = \frac{\rho}{2} v^2 \quad (\text{Pa})$$

where ρ is the gap air density evaluated at T_{gap} (kg/m^3).

The Δp_{HP} term represents the pressure drop due to friction with the shading layer and glass surfaces as the air moves through the gap. Assuming steady laminar flow, it is given by the Hagen-Poiseuille law for flow between parallel plates [Munson et al. 1998]:

$$\Delta p_{HP} = 12\mu \frac{H}{s^2} v \quad (\text{Pa})$$

where μ is the viscosity of air at temperature T_{gap} (Pa-s).

The Δp_Z term is the sum of the pressure drops at the inlet and outlet openings:

$$\Delta p_Z = \frac{\rho v^2}{2} (Z_{in} + Z_{out}) \quad (\text{Pa})$$

Here, the inlet pressure drop factor, Z_{in} , and the outlet pressure drop factor, Z_{out} , are given by

$$Z_{in} = \left(\frac{A_{gap}}{0.66 A_{eq,in}} - 1 \right)^2$$

$$Z_{out} = \left(\frac{A_{gap}}{0.60 A_{eq,out}} - 1 \right)^2$$

where

$A_{eq,in}$ = equivalent inlet opening area (m^2)

$A_{eq,out}$ = equivalent outlet opening area (m^2)

A_{gap} = cross-sectional area of the gap = sW (m^2)

If $T_{gap} > T_{gap,in}$

$$A_{eq,in} = A_{bot} + \frac{A_{top}}{2(A_{bot} + A_{top})} (A_l + A_r + A_h)$$

$$A_{eq,out} = A_{top} + \frac{A_{bot}}{2(A_{bot} + A_{top})} (A_l + A_r + A_h)$$

If $T_{gap} \leq T_{gap,in}$

$$A_{eq,in} = A_{top} + \frac{A_{bot}}{2(A_{bot} + A_{top})} (A_l + A_r + A_h)$$

$$A_{eq,out} = A_{bot} + \frac{A_{top}}{2(A_{bot} + A_{top})} (A_l + A_r + A_h)$$

Here, the area of the openings through which airflow occurs (see Figure 87 and Figure 88) are defined as follows:

A_{bot} = area of the bottom opening (m^2)

A_{top} = area of the top opening (m^2)

A_l = area of the left-side opening (m^2)

A_r = area of the right-side opening (m^2)

A_h = air permeability of the shading device expressed as the total area of openings ("holes") in the shade surface (these openings are assumed to be uniformly distributed over the shade) (m^2)

Figure 88 shows examples of A_{bot} , A_{top} , A_l and A_r for different shading device configurations. These areas range from zero to a maximum value equal to the associated shade/screen/blind-to-glass cross-sectional area; i.e., A_{bot} and $A_{top} \leq sW$, A_l and $A_r \leq sH$.

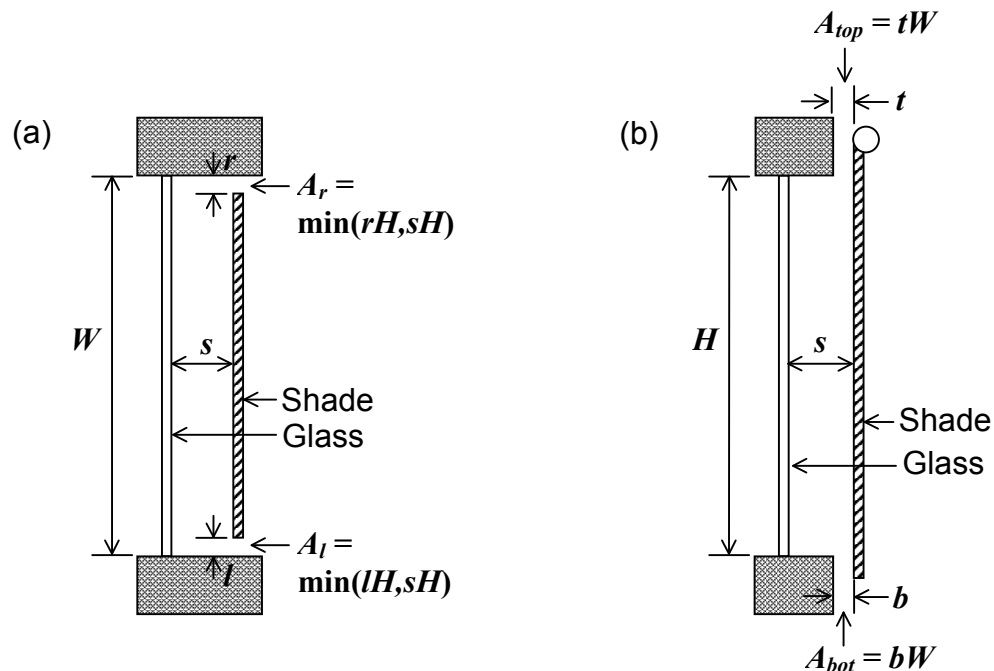


Figure 88. Examples of openings for an interior shading layer covering glass of height H and width W . Not to scale. (a) Horizontal section through shading layer with openings on the left and right sides (top view). (b) Vertical section through shading layer with openings at the top and bottom (side view).

Expression for the Gap Air Velocity

Expressing Equation (194) in terms of v yields the following quadratic equation:

$$\frac{\rho v^2}{2} (1 + Z_{in} + Z_{out}) + \frac{12\mu H}{s^2} v - \rho_0 T_0 g H \sin \phi \frac{|T_{gap,in} - T_{gap}|}{T_{gap,in} T_{gap}} = 0$$

Solving this gives

$$v = \frac{\left[\left(\frac{12\mu H}{s^2} \right)^2 + \frac{2\rho^2 (1 + Z_{in} + Z_{out}) \rho_0 T_0 g H \sin \phi |T_{gap,in} - T_{gap}|}{T_{gap,in} T_{gap}} \right]^{1/2} - \frac{12\mu H}{s^2}}{\rho (1 + Z_{in} + Z_{out})} \quad (196)$$

The choice of the root of the quadratic equation is dictated by the requirement that $v = 0$ if $T_{gap,in} = T_{gap}$.

Gap Outlet Temperature and Equivalent Mean Air Temperature

The temperature of air in the gap as a function of distance, h , from the gap inlet (Figure 89) is

$$T_{gap}(h) = T_{ave} - (T_{ave} - T_{gap,in})e^{-h/H_0}$$

where

$$T_{ave} = \frac{T_{gl} + T_{sh}}{2} \quad (197)$$

is the average temperature of the glass and shading layer surfaces facing the gap (K).

H_0 = characteristic height (m), given by

$$H_0 = \frac{\rho C_p s}{2h_{cv}} v$$

where C_p is the heat capacity of air.

The gap outlet temperature is given by

$$T_{gap,out} = T_{ave} - (T_{ave} - T_{gap,in})e^{-H/H_0} \quad (198)$$

The thermal equivalent mean temperature of the gap air is

$$T_{gap} = \frac{1}{H} \int_0^H T_{gap}(h) dh = T_{ave} - \frac{H_0}{H} (T_{gap,out} - T_{gap,in}) \quad (199)$$

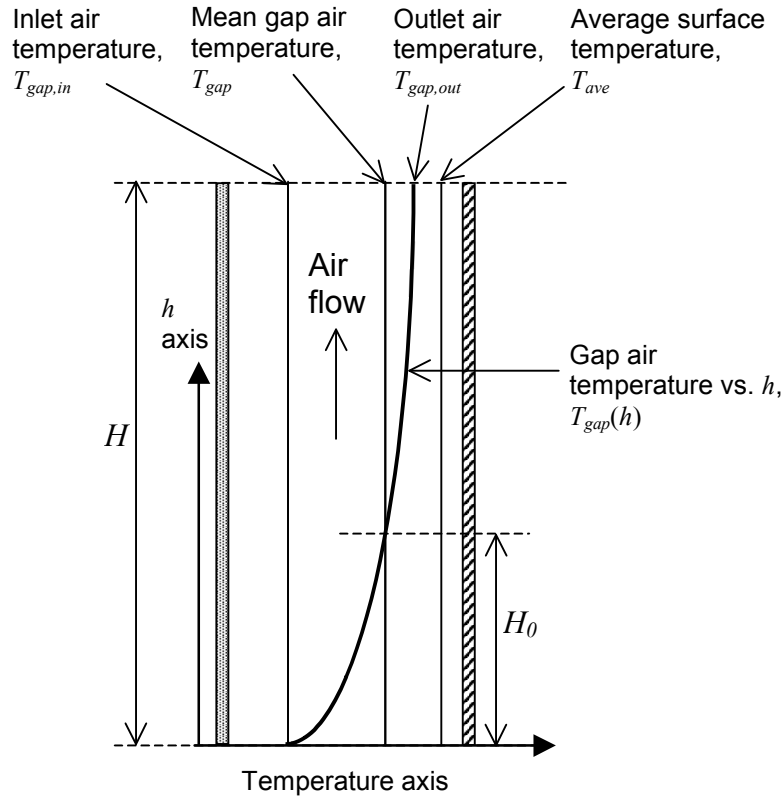


Figure 89. Variation of gap air temperature with distance from the inlet for upward flow.

Solution Sequence for Gap Air Velocity and Outlet Temperature

The routine WinShadeGapFlow is called within the glazing heat balance iterative loop in SolveForWindowTemperatures to determine v and $T_{gap,out}$. The solution sequence in WinShadeGapFlow is as follows:

At start of iteration, guess T_{gap} as $((T_{gl} + T_{sh})/2 + T_{gap,in})/2$. Thereafter use value from previous iteration.

Get still-air conductance, h_c , by calling WindowGasConductance and NusseltNumber.

Get v from Equation (196)

Get h_{cv} from Equation (194)

Get T_{ave} from Equation (197)

Get $T_{gap,out}$ from Equation (198)

Get new value of T_{gap} from Equation (199)

The values of h_{cv} and T_{gap} so determined are then used in the window heat balance equations to find new values of the face temperatures of the glass and shading layers. These temperatures are used in turn to get new values of h_{cv} and T_{gap} until the whole iterative process converges.

Convective Heat Gain to Zone from Gap Airflow

The heat added (or removed) from the air as it passes through the gap produces a convective gain (or loss) to the zone air given by

$$q_v = LW \left[h_{cv} (T_{gl} - T_{gap}) + h_{cv} (T_{sh} - T_{gap}) \right] = 2h_{cv} LW (T_{ave} - T_{gap}) \quad (W)$$

This can also be expressed as

$$q_v = \dot{m} C_p (T_{gap,out} - T_{gap,in}) \quad (W)$$

where the air mass flow rate in the gap is given by

$$\dot{m} = \rho A_{gap} v \quad (\text{kg/s})$$

Heat Balance Equations for Between-Glass Shading Device

In EnergyPlus shading devices are allowed between the two glass panes of double glazing and between the two inner glass panes of triple glazing. Figure 90 shows the case of a between-glass shading device in double glazing.

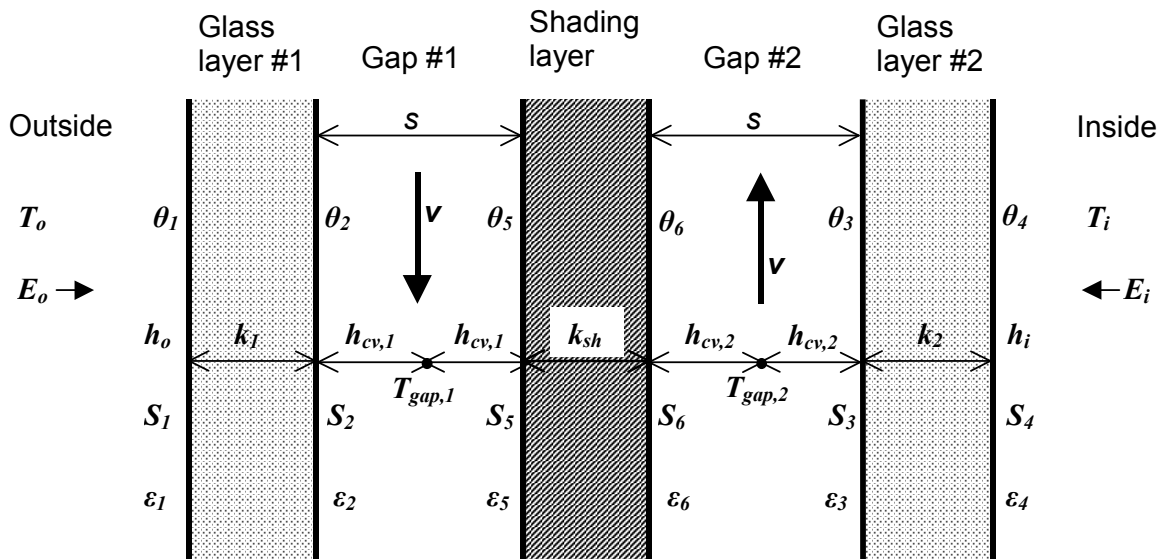


Figure 90. Glazing system with two glass layers and a between-glass shading device showing variables used in the heat balance equations.

The heat balance equations for the two glass surfaces facing the shading device are the following.

For face #2:

$$h_{cv,1} (T_{gap,1} - \theta_2) + k_1 (\theta_1 - \theta_2) + \frac{\sigma \epsilon_2}{1 - \rho_2 R_1} \left[\frac{\tau_{sh}}{1 - \rho_6 \rho_3} (\epsilon_3 \theta_3^4 + \epsilon_6 \theta_6^4 \rho_3) + \epsilon_5 \theta_5^4 + \epsilon_2 \theta_2^4 R_1 \right] - \sigma \epsilon_2 \theta_2^4 + S_2 = 0$$

where

$$R_1 = \rho_5 + \frac{\tau_{sh}^2 \rho_3}{1 - \rho_6 \rho_3}$$

$T_{gap,1}$ = effective mean air temperature in gap 1 (K)

$h_{cv,1}$ = convective heat transfer coefficient from glass or shading layer to gas in gap 1 (W/m²K)

For face #3:

$$h_{cv,2}(T_{gap,2} - \theta_3) + k_2(\theta_4 - \theta_3) + \frac{\sigma \varepsilon_3}{1 - \rho_3 R_2} \left[\frac{\tau_{sh}}{1 - \rho_5 \rho_2} (\varepsilon_2 \theta_2^4 + \varepsilon_5 \theta_5^4 \rho_2) + \varepsilon_6 \theta_6^4 + \varepsilon_7 \theta_3^4 R_2 \right] - \sigma \varepsilon_3 \theta_3^4 + S_3 = 0$$

where

$$R_2 = \rho_6 + \frac{\tau_{sh}^2 \rho_2}{1 - \rho_3 \rho_2}$$

$T_{gap,2}$ = effective mean air temperature in gap 2 (K)

$h_{cv,2}$ = convective heat transfer coefficient from glass or shading layer to gas in gap 2 (W/m²K)

The heat balance equations for the shading layer faces are:

For face #5:

$$h_{cv,1}(T_{gap,1} - \theta_5) + k_{sh}(\theta_6 - \theta_5) + \frac{\sigma \varepsilon_5}{1 - \rho_2 R_1} \left[\frac{\tau_{sh} \rho_2}{1 - \rho_5 \rho_3} (\varepsilon_3 \theta_3^4 + \varepsilon_6 \theta_6^4 \rho_3) + \varepsilon_2 \theta_2^4 + \varepsilon_5 \theta_5^4 \rho_2 \right] - \sigma \varepsilon_5 \theta_5^4 + S_5 = 0$$

For face #6:

$$h_{cv,2}(T_{gap,2} - \theta_3) + k_2(\theta_4 - \theta_3) + \frac{\sigma \varepsilon_3}{1 - \rho_3 R_2} \left[\frac{\tau_{sh}}{1 - \rho_5 \rho_2} (\varepsilon_2 \theta_2^4 + \varepsilon_5 \theta_5^4 \rho_2) + \varepsilon_6 \theta_6^4 + \varepsilon_7 \theta_3^4 R_2 \right] - \sigma \varepsilon_3 \theta_3^4 + S_3 = 0$$

The convective heat transfer coefficients are given by

$$\begin{aligned} h_{cv,1} &= 2h_{c,1} + 4v \\ h_{cv,2} &= 2h_{c,2} + 4v \end{aligned} \quad (200)$$

where

$h_{c,1}, h_{c,2}$ = surface-to-surface heat transfer coefficients for gap #1 and #2, respectively, when these gaps are non-vented (closed).

v = air velocity in the gaps (m/s). It is assumed that the gap widths are equal, so that the velocity of flow in the gaps is equal and opposite, i.e., when the airflow is upward in gap #1 it is downward in gap #2 and vice-versa.

In analogy to the interior or exterior shading device case, the air velocity is determined by solving the following pressure balance equation:

$$\Delta p_{T,1,2} = \Delta p_{B,1} + \Delta p_{HP,1} + \Delta p_{Z,1} + \Delta p_{B,2} + \Delta p_{HP,2} + \Delta p_{Z,2} \quad (201)$$

where the driving pressure difference between gap #1 and #2 is

$$\Delta p_{T,1,2} = \rho_0 T_0 g H \sin \phi \frac{|T_{gap,1} - T_{gap,2}|}{T_{gap,1} T_{gap,2}} \quad (\text{Pa})$$

The pressure drops on the right-hand side of this equation are:

$$\begin{aligned} \Delta p_{B,i} &= \frac{\rho_{gap,i}}{2} v^2 \\ \Delta p_{HP,i} &= 12 \mu_{gap,i} \frac{H}{s^2} \\ \Delta p_{Z,i} &= \frac{\rho_{gap,i} v^2}{2} (Z_{in,i} + Z_{out,i}) \end{aligned}$$

where $i = \text{gap number (1 or 2)}$.

It can be shown that $Z_{in,1} + Z_{out,1} = Z_{in,2} + Z_{out,2}$. Then, inserting these pressure drop expressions into (201), we obtain the following expression for the airflow velocity:

$$v = \frac{\left[\left(\frac{12(\mu_{gap,1} + \mu_{gap,2})H}{s^2} \right)^2 + 2\Delta p_{T,1,2}(\rho_{gap,1} + \rho_{gap,2})(1 + Z_{in} + Z_{out}) \right]^{1/2} - \frac{12(\mu_{gap,1} + \mu_{gap,2})H}{s^2}}{(\rho_{gap,1} + \rho_{gap,2})(1 + Z_{in} + Z_{out})} \quad (202)$$

The choice of the sign of the square root term is dictated by the requirement that $v = 0$ if $\Delta p_{T,1,2} = 0$, i.e., $T_{gap,1} = T_{gap,2}$.

Given v we can now calculate $T_{gap,1}$ and $T_{gap,2}$, which gives $\Delta p_{T,1,2}$. The procedure is as follows. We have

$$T_{gap,1,out} = T_{ave,1} - (T_{ave,1} - T_{gap,1,in}) \xi_1$$

where $T_{ave,1} = (\theta_2 + \theta_5)/2$ and $\xi_1 = e^{-\frac{H}{H_{0,1}}}$ with $H_{0,1} = \rho_{gap,1} C_p s v / (2h_{cv,1})$. Since $T_{gap,1,in} = T_{gap,2,out}$ this gives:

$$T_{gap,1,out} = T_{ave,1} - (T_{ave,1} - T_{gap,2,out}) \xi_1$$

Similarly,

$$T_{gap,2,out} = T_{ave,2} - (T_{ave,2} - T_{gap,1,out})\xi_2$$

Solving these simultaneous equations gives:

$$T_{gap,1,out} = \frac{T_{ave,1}(1 - \xi_1) + \xi_1 T_{ave,2}(1 - \xi_2)}{1 - \xi_1 \xi_2}$$

$$T_{gap,2,out} = \frac{T_{ave,2}(1 - \xi_2) + \xi_2 T_{ave,1}(1 - \xi_1)}{1 - \xi_1 \xi_2}$$

Using these in

$$T_{gap,1} = T_{ave,1} - \frac{H_{0,1}}{H} (T_{gap,1,out} - T_{gap,2,out})$$

gives

$$T_{gap,1} = T_{ave,1} - \frac{H_{0,1}}{H} \xi (T_{ave,1} - T_{ave,2}) \quad (203)$$

with

$$\xi = \frac{(1 - \xi_1)(1 - \xi_2)}{1 - \xi_1 \xi_2}$$

Similarly, from

$$T_{gap,2} = T_{ave,2} - \frac{H_{0,2}}{H} (T_{gap,2,out} - T_{gap,1,out})$$

we get

$$T_{gap,2} = T_{ave,2} - \frac{H_{0,2}}{H} \xi (T_{ave,2} - T_{ave,1}) \quad (204)$$

The overall solution sequence is as follows.

At start of iteration guess $T_{gap,1} = T_{ave,1}$ and $T_{gap,2} = T_{ave,2}$. Then

- 1) Get $\rho_{gap,1}$, $\rho_{gap,2}$, $\mu_{gap,1}$, $\mu_{gap,2}$ using $T_{gap,1}$, $T_{gap,2}$.
- 2) Get still-air conductances $h_{c,1}$, $h_{c,2}$ by calling WindowGasConductance and NusseltNumber.
- 3) Get ν from Equation (202)
- 4) Get $h_{cv,1}$, $h_{cv,2}$ from Equation (200)
- 5) Get $T_{ave,1}$, $T_{ave,2}$

6) Get $H_{o,1}$, $H_{o,2}$, ξ_1 and ξ_2 .

7) Get $T_{gap,1}$, $T_{gap,2}$ from Equations (203) and (204)

The values $h_{cv,1}$, $h_{cv,2}$, $T_{gap,1}$ and $T_{gap,2}$ are then used in the face heat balance equations to find new values of the face temperatures θ_2 , θ_3 , θ_5 and θ_6 . These are used in turn to get new values of $h_{cv,1}$, $h_{cv,2}$, $T_{gap,1}$ and $T_{gap,2}$ until the whole iterative process converges.

Airflow Windows

In airflow windows forced air flows in the gap between adjacent layers of glass. Such windows are also known as “heat-extract windows” and “climate windows.”

Five configurations of airflow windows are modeled (Figure 91) that depend on the source and destination of forced air. The allowed combinations of Airflow Source and Airflow Destination are:

- InsideAir → OutsideAir
- InsideAir → InsideAir
- InsideAir → ReturnAir
- OutsideAir → InsideAir
- OutsideAir → OutsideAir

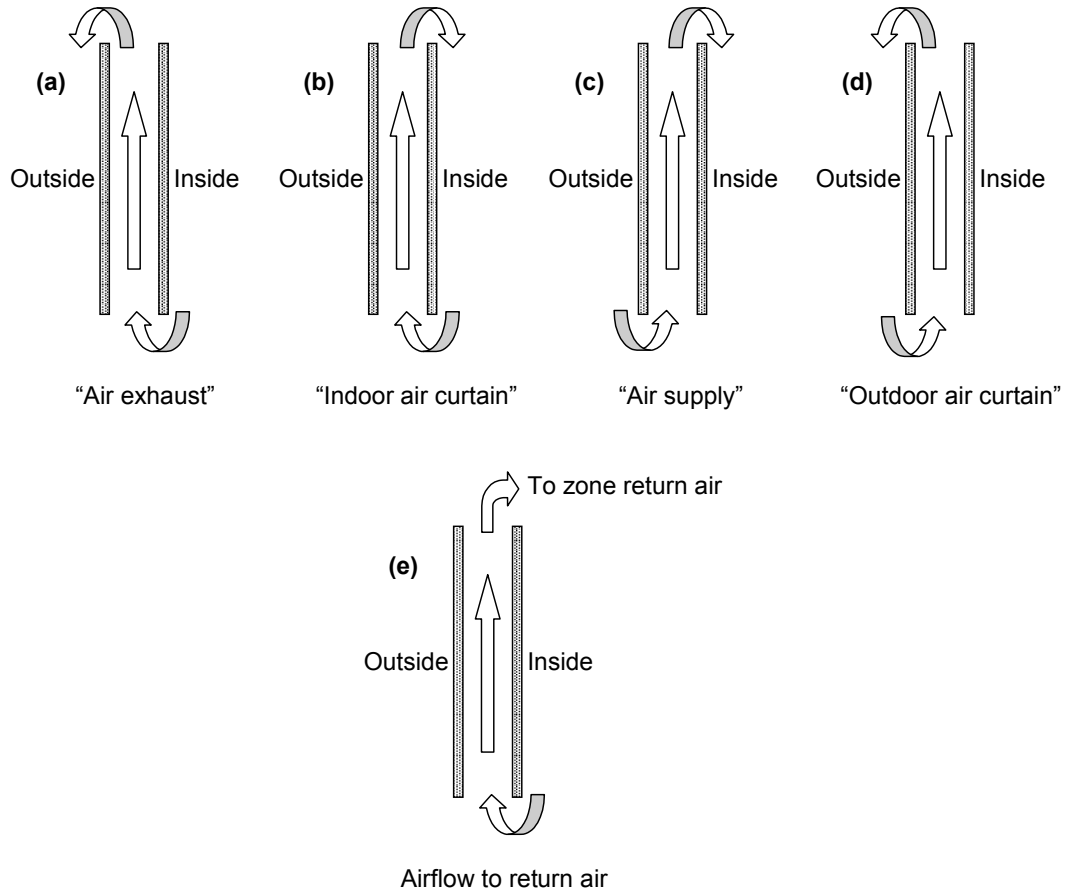


Figure 91. Gap airflow configurations for airflow windows. From "Active facades," Version no. 1, Belgian Building Research Institute, June 2002.

A common application of airflow windows is to reduce the zone cooling load by exhausting indoor air through the window, thereby picking up and rejecting heat from the glazing (Figure 91).

Figure 92 shows the variables used in the heat balance equations for forced airflow in a double-glazed window.

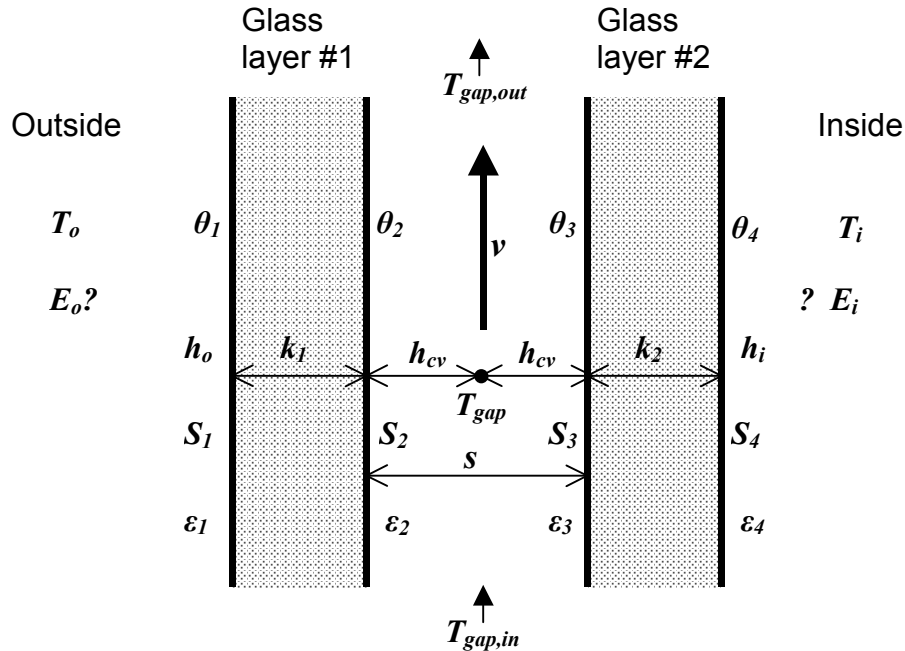


Figure 92. Glazing system with forced airflow between two glass layers showing variables used in the heat balance equations.

The heat balance equation for the left-hand glass surface facing the gap in Figure 92 is:

$$k_1(\theta_1 - \theta_2) + h_{cv}(T_{gap} - \theta_2) + \sigma \frac{\epsilon_2 \epsilon_3}{1 - (1 - \epsilon_2)(1 - \epsilon_3)} (\theta_3^4 - \theta_2^4) + S_2 = 0$$

The corresponding equation for the right-hand glass surface facing the gap is:

$$k_2(\theta_4 - \theta_3) + h_{cv}(T_{gap} - \theta_3) + \sigma \frac{\epsilon_2 \epsilon_3}{1 - (1 - \epsilon_2)(1 - \epsilon_3)} (\theta_2^4 - \theta_3^4) + S_3 = 0$$

Here,

T_{gap} = effective mean temperature of the gap air (K)

h_{cv} = convective heat transfer coefficient from glass to gap air (W/m²K).

The convective heat transfer coefficient is given by

$$h_{cv} = 2h_c + 4v$$

where

h_c = glass-to-glass heat transfer coefficient for non-vented (closed) cavity (W/m²K)

v = mean air velocity in the gap (m/s).

The air velocity is determined by the gap cross-sectional area in the flow direction and the air flow rate, which is an input value that is constant or can vary according to a user-specified schedule:

$$v = \frac{F}{A_{gap}} \quad (\text{m/s})$$

where

F = airflow rate (m^3/s)

A_{gap} = gap cross-sectional area in direction of flow (m^2)

It is assumed that the airflow is uniform across the width of the window.

The mean temperature of the gap air is given by the following expression, whose derivation follows that for (199) for the case of an interior shading device:

$$T_{gap} = T_{ave} - \frac{H}{H_0}(T_{gap,out} - T_{gap,in})$$

where

$$T_{ave} = \frac{\theta_2 + \theta_3}{2}$$

H = glazing height (m)

$$H_0 = \frac{\rho C_p s}{2h_{cv}} v$$

$T_{gap,in}$ = gap air inlet temperature (T_i if the airflow source is indoor air, T_o if the airflow source is outside air) (K)

The outlet air temperature is given by

$$T_{gap,out} = T_{ave} - (T_{ave} - T_{gap,in})e^{-H/H_0}$$

The equations for glass face #1 and #4 are the same as those for no airflow in the gap (Equations (183) and (186)).

The convective heat gain to the zone air due to the gap airflow when the airflow destination is indoor air is

$$q_v = \dot{m}(C_{p,out}T_{gap,out} - C_{p,i}T_i) \quad (\text{W})$$

where

$C_{p,i}$ = heat capacity of the indoor air (J/kg-K)

$C_{p,out}$ = heat capacity of the gap outlet air (J/kg-K)

and where the air mass flow rate in the gap is

$$\dot{m} = \rho F \quad (\text{kg/s})$$

Fan Energy

The fan energy used to move air through the gap is very small and is ignored.

Airflow Window with Between-Glass Shading Device

Figure 93 shows the case of a double-glazed airflow window with a between glass shading device. The heat balance equations in this case are the same as those for the between-glass shading device with natural convection (Figure 90 and following equations) except that now

$$v = \frac{F/2}{A_{gap}} \quad (\text{m/s})$$

where $A_{gap} = sW$ is the cross-sectional area of the gap on either side of the shading device. It is assumed that the shading device is centered between the two panes of glass so that the airflow, F , is divided equally between the two gaps.

The convective heat gain to the zone air due to the airflow through the two gaps when the airflow destination is indoor air is

$$q_v = \dot{m}(C_{p,ave,out}T_{gap,ave,out} - C_{p,i}T_i) \quad (\text{W})$$

where the average temperature of the two outlet air streams is

$$T_{gap,ave,out} = (T_{gap,1,out} + T_{gap,2,out})/2$$

and

$C_{p,ave,out}$ = heat capacity of the outlet air evaluated at $T_{gap,ave,out}$ (J/kg-K)

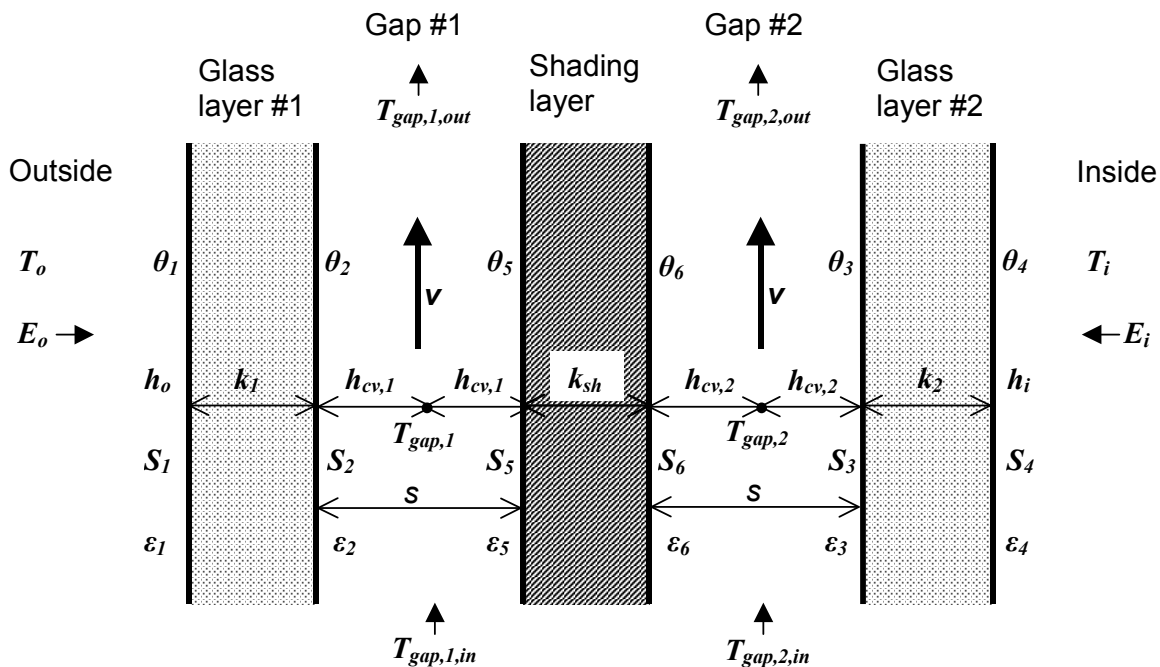


Figure 93. Airflow window with between-glass shading device showing variables used in the heat balance equations.

References

Arasteh, D.K., M.S. Reilly and M.D. Rubin. 1989. A versatile procedure for calculating heat transfer through windows. American Society of Heating, Refrigeration and Air-Conditioning Engineers, ASHRAE Transactions, Vol. 95, Pt. 2.

Finlayson, E.U., D.K. Arasteh, C. Huizenga, M.D. Rubin and M.S. Reilly. 1993. WINDOW 4.0: documentation of calculation procedures. Lawrence Berkeley National Laboratory report no. LBL-33943.

ISO. 2003. ISO 15099:2003. Thermal performance of windows, doors, and shading devices – Detailed calculations. International Organization for Standardization.

Munson, B.R, D.F. Young and T.H. Okiishi. 1998. "Fundamentals of Fluid Mechanics," Third Edition Update, John Wiley & Sons, Inc.

Simmler, H., U. Fischer and F. Winkelmann. 1996. Solar-Thermal Window Blind Model for DOE-2. Lawrence Berkeley National Laboratory, Simulation Research Group internal report, (unpublished).

Arasteh, D.K., J.C. Kohler and B.T. Griffith. Draft 2009. Modeling Windows in EnergyPlus with only U, SHGC, and optionally VT. LBNL report. Full reference to be determined.

Air Heat Balance Manager / Processes

Convection from Surfaces

This contribution is expressed using the convective heat transfer coefficient as follows:

$$q_{conv} = \sum_{i=1}^{nsurfaces} h_{c,i} A_i (T_a - T_{s,i}) \quad (205)$$

The inside heat transfer coefficient is modeled from a choice of correlations.

Convection from Internal Sources

This component is the companion part of the radiant contribution from internal gains described previously. It is added directly into the air heat balance. Such a treatment also violates the tenets of the heat balance since the surface temperature of the surfaces producing the internal loads exchange heat with the zone air through normal convective processes. However, once again, the details required to include this component in the heat balance are generally not available, and its direct inclusion into the air heat balance is a reasonable approach..

Infiltration/Ventilation

Any air that enters by way of infiltration is assumed to be immediately mixed with the zone air. The determination of the amount of infiltration air is quite complicated and subject to significant uncertainty. In the most common procedure, the infiltration quantity is converted from a number of air changes per hour (ACH) and included in the zone air heat balance using the outside temperature at the current hour.

EnergyPlus contains three models for infiltration. The first is the “Design Flow Rate” model that was inherited from EnergyPlus’s predecessor programs. It is accessed through the ZoneInfiltration:DesignFlowRate object and is based on environmental conditions modifying a design flow rate. The second is the “Effective Leakage Area” model based on Sherman and Grimsrud (1980) and accessed using the ZoneInfiltration:EffectiveLeakageArea input object. The third is the “Flow Coefficient” model based on Walker and Wilson (1998) and accessed using the ZoneInfiltration:FlowCoefficient input object. The model formulations for the Effective Leakage Area and Flow Coefficient models are from the ASHRAE Handbook of Fundamentals (2001 Chapter 26; 2005 Chapter 27) where they are referred to as “Basic” and “Enhanced”, respectively.

Infiltration Design Flow Rate

Infiltration (Ref Object: ZoneInfiltration:DesignFlowRate) is the unintended flow of air from the outdoor environment directly into a thermal zone. Infiltration is generally caused by the opening and closing of exterior doors, cracks around windows, and even in very small amounts through building elements. In this model, the user defines a design flow rate that can be modified by temperature differences and windspeed. The basic equation (Coblentz and Achenbach 1963) used to calculate infiltration with this model is:

$$Infiltration = (I_{design}) (F_{schedule}) \left[A + B | (T_{zone} - T_{odb}) | + C (WindSpeed) + D (Windspeed^2) \right]$$

More advanced infiltration calculations are possible using the EnergyPlus AirflowNetwork model for natural infiltration driven by wind when the HVAC system does not operate and/or

driven by wind and forced air for times when the HVAC system operates. Infiltration described by the equation shown above is entered into EnergyPlus using the following syntax. Exfiltration (the leakage of zone air to the outside) is generally handled better as zone exhaust air in the zone equipment description.

The question of typical values for these coefficients is subject to debate. Ideally, one should do a detailed analysis of the infiltration situation and then determine a custom set of coefficients using methods such as those laid out in Chapter 26 of the ASHRAE Handbook of Fundamentals. The EnergyPlus defaults are 1,0,0,0 which gives a constant volume flow of infiltration under all conditions.

BLAST (one of the EnergyPlus predecessors) used the following values as defaults: 0.606, 0.03636, 0.1177, 0. These coefficients produce a value of 1.0 at 0C deltaT and 3.35 m/s (7.5 mph) windspeed, which corresponds to a typical summer condition. At a winter condition of 40C deltaT and 6 m/s (13.4 mph) windspeed, these coefficients would increase the infiltration rate by a factor of 2.75.

In DOE-2 (the other EnergyPlus predecessor), the air change method defaults are (adjusted to SI units) 0, 0, 0.224 (windspeed), 0. With these coefficients, the summer conditions above would give a factor of 0.75, and the winter conditions would give 1.34. A windspeed of 4.47 m/s (10 mph) gives a factor of 1.0.

The source of the BLAST defaults is noted in the BLAST documentation as:

"Empirical equation and the coefficient default were determined from ASHRAE journal articles and other data on the effects of outdoor weather conditions."

The source of the DOE-2 defaults is based on examining the infiltration relationships described in the ASHRAE Handbook of Fundamentals.

The EnergyPlus example files use all of the above, the BLAST defaults in some (e.g., GeometryTest), the DOE-2 defaults in some (e.g., 5ZoneAirCooled), and the EnergyPlus defaults in some (e.g., LgOffVAVDetCoil).

Infiltration by Effective Leakage Area

The Effective Leakage Area model is based on Sherman and Grimsrud (1980) and accessed using the ZoneInfiltration:EffectiveLeakageArea input object. The model formulation used in EnergyPlus is from the ASHRAE Handbook of Fundamentals (2001 Chapter 26; 2005 Chapter 27) where it is referred to as the "Basic" model.

The Effective Leakage Area, or Sherman-Grimsrud, model is:

$$Infiltration = (F_{Schedule}) \frac{A_L}{1000} \sqrt{C_s \Delta T + C_w (WindSpeed)^2} \quad (206)$$

where,

$F_{schedule}$ is a value from a user-defined schedule,

A_L is the effective air leakage area in cm^2 that corresponds to a 4 Pa pressure differential,

C_s is the coefficient for stack-induced infiltration in $(L/s)^2/(cm^4 \cdot K)$,

ΔT is the absolute temperature difference between zone air and outdoor air,

C_w is the coefficient for wind-induced infiltration in $(L/s)^2/(cm^4 \cdot (m/s)^2)$, and

$WindSpeed$ is the local wind speed.

Infiltration by Flow Coefficient

The Flow Coefficient model is based on Walker and Wilson (1998) and accessed using the ZoneInfiltration:FlowCoefficient input object. The model formulation used in EnergyPlus is

from the ASHRAE Handbook of Fundamentals (2001 Chapter 26; 2005 Chapter 27) where it is referred to as the “Enhanced” or “AIM-2” model.

The Enhanced, or AIM-2, model is:

$$Infiltration = (F_{Schedule}) \sqrt{\left(c C_s \Delta T^n \right)^2 + \left(c C_w (s * WindSpeed)^{2n} \right)^2} \quad (207)$$

where,

$F_{schedule}$ is a value from a user-defined schedule,

c is the flow coefficient in $m^3/(s \cdot Pa^n)$,

C_s is the coefficient for stack-induced infiltration in $(Pa/K)^n$,

n is the pressure exponent,

C_w is the coefficient for wind-induced infiltration in $(Pa \cdot s^2/m^2)^n$, and

s is the shelter factor.

References:

Coblentz, C. W. and Achenbach, P. R. 1963. Field Measurement of Ten Electrically-Heated Houses. ASHRAE Transactions pp 358-365.

Sherman, M.H. and D.T. Grimsrud. 1980. Infiltration-pressurization correlation: Simplified physical modeling. ASHRAE Transactions 86(2):778

Walker, I.S., and D.J. Wilson. 1998. Field validation of equations for stack and wind driven air infiltration calculations. International Journal of HVAC&R Research 4(2).

ASHRAE Handbook of Fundamentals. 2005. Chapter 27. (and 2001 Chapter 26).

Ventilation

Ventilation (Ref Object: ZoneVentilation) is the purposeful flow of air from the outdoor environment directly into a thermal zone in order to provide some amount of non-mechanical cooling. Ventilation as specified by this input syntax is intended to model “simple” ventilation as opposed to the more detailed ventilation investigations that can be performed with the AirflowNetwork model. Simple ventilation in EnergyPlus can be controlled by a schedule and through the specification of minimum, maximum and delta temperatures. The temperatures can be either single constant values for the entire simulation or schedules which can vary over time. Specific details are given in the Input/Output reference document. As with infiltration, the actual flow rate of ventilation can be modified by the temperature difference between the inside and outside environment and the wind speed. The basic equation used to calculate ventilation in EnergyPlus is:

$$Ventilation = (V_{design}) (F_{schedule}) \left[A + B |T_{zone} - T_{odb}| + C(WindSpeed) + D(WindSpeed^2) \right]$$

More advanced ventilation calculations are possible using the EnergyPlus AirflowNetwork model.

Air Exchange

Air exchange and interchange between zones is treated as a convective gain.

Calculation of Zone Air Temperature

The zone air heat balance is the primary mechanism for linking the loads calculation to the system simulation. As such, the zone air temperature becomes the main interface variable. Its role in the integration process was described previously (“Basis for the Zone and System Integration”).

Building System Simulation System Manager / Processes

EnergyPlus uses a loop based HVAC system formulation. An example of the dual duct VAV system is shown below.

Air Loops

Definition of Air Loop

In EnergyPlus an air loop is a central forced air HVAC system. The term “loop” is used because in most cases some air is recirculated so that the air system forms a fluid loop. The air loop is just the “air side” of a full HVAC system. The input objects related to these air loops begin “AirLoopHVAC.”

For simulation purposes the air loop is divided into 2 parts: the primary air system (representing the supply side of the loop) and the zone equipment (representing the demand side of the loop). The primary air system includes supply and return fans, central heating and cooling coils, outside air economizer, and any other central conditioning equipment and controls. The zone equipment side of the loop contains the air terminal units as well as fan coils, baseboards, window air conditioners, and so forth. Supply and return plenums are also included on the zone equipment side of the loop.

Simulation Method

Simulating a forced air system and its associated zones can be done in a number of ways. EnergyPlus uses algebraic energy and mass balance equations combined with steady state component models. When the zone air and the air system are modeled with algebraic equations (steady state) then – in cases with recirculated air – there will be one or more algebraic loops. In other words it is not possible to solve the equations directly; instead iterative methods are needed. Typically a humidity ratio and a mass flow rate will be variables involved in the iterative solution.

In EnergyPlus the zone humidity ratios and temperatures are decoupled from the solution of the air system equations. The zone air is assigned heat and moisture storage capacities and the capacity effects are modeled by 1st order ordinary differential equations. During each system simulation time step new zone temperatures and humidities are predicted using past values. The zone temperatures and humidities are then held constant during the simulation of the air system (and the plant). Then the zone temperatures and humidity ratios are corrected using results from the system simulation. As a result the usual algebraic loops arising in steady state air system simulations are eliminated from the EnergyPlus system simulation. The zone temperatures, humidity ratios, and heating and cooling demands are known inputs to the system simulation.

The need for iteration can be reintroduced by the need for system control. If system setpoints are fixed, externally determined, or lagged and control is local (sensor located at a component outlet, actuator at a component inlet) then iteration can be confined to the components and the overall air system equations can be solved directly. However these requirements are too restrictive to simulate real systems. System setpoints are held fixed during a system time step. But controller sensors are allowed to be remote from the location of the actuator. Consequently EnergyPlus uses iteration over the entire primary air system in order to converge the system controllers.

Component Models

EnergyPlus component models are algorithmic with fixed inputs and outputs. They are embodied as Fortran90 subroutines within software modules. For each component several choices of inputs and outputs are possible. There is no one choice that will be most usable and efficient for all system configurations and control schemes. For reasons of consistency

and comprehensibility the requirement was imposed that all EnergyPlus models be forward models. That is, the component inputs correspond to the inlet conditions and the outputs to the outlet conditions. If another choice of inputs and outputs is needed it is obtained by numerical inversion of the forward model.

Iteration Scheme

The primary air system simulation uses successive substitution starting at the return air inlet node and proceeding in the flow direction to the supply air outlet nodes. The iteration proceeds until an individual actuator-controller has converged (the sensed value matches the setpoint within the specified tolerance). The system controllers are simulated in sequence. During this sequence of iterative solutions the air mass flow rates are held constant. The controllers are converged by the method of interval halving. This method was chosen (rather than for instance Newton-Raphson) for its robustness.

Determination of Air Mass Flow Rates

In most cases the air mass flow rate of the central air system is set by zone equipment downstream of the primary air system. The air terminal unit components with their built in dampers and controllers respond to the zone heating and cooling loads by setting air flow rates at their inlet nodes. These flow rates are passed back upstream to the primary air system, establishing the flow rates in the primary air system branches. These flow rates are held fixed while the primary air system is simulated.

Air Loop Simulation

A complete simulation of each primary air system – zone equipment air loop is done in the following manner.

- If this is the first simulation this system time-step, just call *ManageAirLoops* (simulates the primary air systems) and *ManageZoneEquipment* (simulates the zone equipment sets) once and quit. This initial pass is simulated with full design air flow rates and allows the zone equipment to set the flow rates for each zone that will meet the zone loads.
- Otherwise loop over primary air systems and zone equipment sets until the temperatures, flow rates, enthalpies, humidity ratios etc. agree to within tolerance at each primary air system – zone equipment gap.

```
DO WHILE ((SimAirLoops .OR. SimZoneEquipment) .AND. (IterAir.LE.MaxAir) )
  IF (SimAirLoops) THEN
    CALL ManageAirLoops(FirstHVACIteration,SimAirLoops,SimZoneEquipment)
    SimPlantDemandLoops = .TRUE
    SimElecCircuits = .TRUE.
  END IF

  IF (SimZoneEquipment) THEN
    CALL ResolveAirLoopFlowLimits(IterAir+1)
    CALL ManageZoneEquipment(FirstHVACIteration,SimZoneEquipment,SimAirLoops)
    SimPlantDemandLoops = .TRUE.
    SimElecCircuits = .TRUE.
  END IF

  IterAir = IterAir + 1

END DO

CALL ResolveLockoutFlags(SimAirLoops)
```

The logical flags *SimAirLoops* and *SimZoneEquipment* are used to signal whether the primary air systems or the zone equipment sets need to be resimulated. These flags are set by the subroutine *UpdateHVACInterface* which is called from within *ManageAirLoops* and *ManageZoneEquipment* at the end of each half-loop simulation. *UpdateHVACInterface* (when called from *ManageAirLoops*) passes the values at the outlet nodes of a primary air system on to the inlet nodes of the corresponding zone equipment half-loop and similarly (when called from *ManageZoneEquipment*) passes the values of the outlet nodes of a zone equipment half-loop on to the inlet nodes of its corresponding primary air system. Each time *UpdateHVACInterface* is called it also checks whether the values at the half-loop outlet nodes are in agreement with the values at the downstream half-loop inlet nodes. If they are not it sets the simulate flag of the downstream half-loop to *true*. The values checked by *UpdateHVACInterface* and their tolerances are as follows.

Quantities	Tolerances
specific enthalpy [J/kg}	10.0
mass flow rate [kg/s]	0.01
humidity ratio [kg H ₂ O / kg dry air]	0.0001
quality	0.01
air pressure [Pa]	10.0
temperature [C]	0.01
energy [J]	10.0

ResolveAirLoopFlowLimits is invoked to deal with zone equipment – primary air system flow mismatches. For instance the zone air terminal units (ATUs) may be asking for more air than the central fan can supply. In this case *ResolveAirLoopFlowLimits* takes the air flow that the fan can supply and apportions it among the ATUs in proportion to their design maximum air flow rates (*ResolveAirLoopFlowLimits* sets the $\dot{m}_{\max\,avail,\,node}$ at the entering node of each ATU in the system).

At the end of the air loop simulation *ResolveLockoutFlags* is called. This subroutine checks if any air system component has requested that the economizer be locked out. If such a request has been made and if the economizer is active, *ResolveLockoutFlags* sets *SimAirLoops* to *true* and the *EconoLockout* flag to *true* to ensure that the air loop will be resimulated with the economizer forced off.

Primary Air System Simulation

When the EnergyPlus HVAC simulation manager needs to simulate the primary air system side of the air loop it calls *ManageAirLoops*, the primary air system simulation manager subroutine.

Note that “air loop” is used inconsistently in the program: sometimes it means the full loop consisting of both supply & demand sides – primary air system and zone equipment; sometimes it means just the supply side – the primary air system.

Like the other manager routines in EnergyPlus, *ManageAirLoops* has a very simple structure:

Table 33. ManageAirLoop Code

```

IF (GetInputFlag) THEN !First time subroutine has been entered
    CALL GetAirPathData ! Get air loop descriptions from input file
    GetInputFlag=.false.
END IF

    ! Initialize air loop related parameters
CALL InitAirLoops(FirstHVACIteration)

    ! Call the AirLoop Simulation
IF (SysSizingCalc) THEN
    CALL SizeAirLoops
ELSE
    CALL SimAirLoops(FirstHVACIteration,SimZoneEquipment)
END IF

! No Update

    ! Report information at the Manage Air Loop Level
CALL ReportAirLoops

```

- 1) If the user input data has not been input, get the data and store it in the air loop data structures.
- 2) Perform air loop initialization calculations:
 - a) at the beginning of the simulation (one time initializations);
 - b) at the start of each environment (design day or simulation run period);
 - c) before each air loop simulation.
- 3) If automatic sizing of the loop flow rates is called for, do it.
- 4) Otherwise perform a simulation of the air loop.

Input data

The input data specifying an air loop consists of:

- 1) the loop configuration;
 - (a) Splitters, Mixers, and Branches;
 - (b) Components on the Branches
- 2) loop control;
 - (a) Controllers;
 - (b) System Availability Managers;
- 3) connection to zone equipment;
- 4) design flow rate.

These objects and their data are described in the EnergyPlus *Input Output Reference* document. The utility routines used to get and check the data are described in the EnergyPlus *Guide for Module Developers*, section Input Services.

Initialization Calculations

One Time Calculations

Zones Served by System

The main one time calculation involves figuring out what zones are served by each air loop. The EnergyPlus input does not explicitly describe which zones receive supply air from a given air loop. Instead that knowledge is embedded implicitly in the overall air loop – zone equipment network description. For sizing calculations it is important to have a data structure

that explicitly shows which zones each air loop serves. For instance, the air loop design supply air flow rate is obtained by summing the design heating or cooling air flow rates of the zones connected to the air loop.

For each air loop, the following calculation is performed.

- (1) For each air loop outlet branch, the corresponding zone equipment inlet node is identified.
- (2) This node number is compared to the inlet node of all AirLoopHVAC:SupplyPaths. When a match is found, the AirLoopHVAC:ZoneSplitter for this supply path is identified.
- (3) The outlet nodes of the AirLoopHVAC:ZoneSplitter are compared to the cooling inlet nodes of all the zone ZoneHVAC:AirDistributionUnits. When a match is found this zone is identified as being served by cooling supply air from the air loop.
- (4) Similarly the outlet nodes of the AirLoopHVAC:ZoneSplitter are compared with the heating inlet nodes of all ZoneHVAC:AirDistributionUnits. A match indicates that this zone is served by heating supply air from the air loop.
- (5) The case where there is no AirLoopHVAC:ZoneSplitter for a AirLoopHVAC:SupplyPath must be handled. In this case the program looks for a match between the zone equipment inlet node and an ZoneHVAC:AirDistributionUnit heating or cooling inlet node. When a match is found that zone is identified as being served with heating or cooling supply air from the air loop.
- (6) The list of cooled and heated zones are saved in the air loop data structure AirToZoneNodeInfo.

Branch Sizing

If this *not* an air loop sizing calculation, but is the first pass through the HVAC code in a normal simulation, loop over all the branches in all air loops and trigger the branch design air flow auto-sizing calculation. The actual calculation is described in the Sizing section of this document.

Begin Environment Initializations

For each air loop, loop over all the branches in the loop. Initialize each branch mass flow rate:

$$\dot{m}_{br,max} = \rho_{std} \cdot \dot{V}_{br,max}$$

$$\dot{m}_{br,min} = \rho_{std} \cdot \dot{V}_{br,min}$$

where ρ_{std} is the density of air at 20 degrees C, humidity ratio = 0, standard pressure.

For each branch, loop over all the nodes on the branch and set the node data to the following values:

$$T_{node} = 20^{\circ} C$$

$$W_{node} = W_{oa}$$

$$h_{node} = \text{PsyHFnTdbW}(20.0, W_{oa})$$

$$\dot{m}_{node} = \dot{m}_{br,max}$$

$$\dot{m}_{max,node} = \dot{m}_{br,max}$$

$$\dot{m}_{maxavail,node} = \dot{m}_{br,max}$$

$$\dot{m}_{\min,node} = 0.0$$

$$\dot{m}_{setpt,node} = 0.0$$

$$\dot{m}_{\min avail,node} = 0.0$$

$$P_{node} = P_{std,baro}$$

$$Qu_{node} = 0.0$$

where W_{oa} is the humidity ratio of the outside air; PsyHFnTdbW is the EnergyPlus psychrometric function for enthalpy h , given temperature and humidity ratio; and Qu is quality.

System Time Step Initializations

For each branch in each air loop, loop over all the nodes on the branch and set $\dot{m}_{setpt,node} = 0.0$; if it is the start of an HVAC solution sequence set $\dot{m}_{\max avail,node} = \dot{m}_{\max,node}$. Then set the mass flow rate setpoints for the air loop nodes.

- 1) On each air loop, loop over the outlet branches and find the loop outlet nodes. If it is the start of an HVAC solution sequence, set $\dot{m}_{setpt,outletnode} = \dot{m}_{outletbr,max}$. This will insure that during the first pass through the full loop that the mass flow rate will be at the maximum. Otherwise, set $\dot{m}_{setpt,outletnode} = \dot{m}_{zone eq inletnode}$. This sets the air loop flow rate to the total zone requirement.
- 2) Pass the mass flow rate setpoint upstream to the start of the outlet branches; through the splitter, if one exists; and upstream to the beginning node of the splitter inlet branch.
- 3) Sum the total return air mass flow rate and save it in the `AirLoopFlow` data structure.
- 4) For each air loop, loop over the inlet nodes and, at the start of each HVAC solution sequence, set the entering node mass flow rate equal to the primary air system design mass flow rate (subject to it not being larger than the entering node mass flow rate setpoint).

Central air system simulation

The subroutine `SimAirLoops` does the actual simulation the central air system equipment for all the air loops. The simulation of a full air loop (central equipment plus zone terminal units and equipment) requires the interaction of 2 managers: `ManageAirLoops` and `ManageZoneEquipment`. Thus a single call to `SimAirLoops` results in a simulation of all the central air system equipment, but this is one part of a larger iterative simulation of the full air loops involving the zone equipment as well.

`SimAirLoops` accomplishes its task using a set of nested loops.

- Loop over all of the central air systems (*Air Primary Loops*).
 - For each air system, make 1 or 2 simulation passes
 - Loop over each controller in the *Air Primary Loop*
 - For each controller, repeat the simulation of all the *Air Primary Loop* components until the controller has converged
 - Loop over each branch in the *Air Primary Loop*
 - On each branch, simulate in sequence each component

During and at the end of each loop some tests are performed.

At the end of the first pass of loop 2, a decision is made on whether a second pass is needed. The first pass has been performed assuming that there is a mass flow balance in the central air system simulation. This is usually the case. A call to *ResolveSysFlow* checks the mass balance and imposes a mass balance if there is not a balance. The lack of a system mass balance requires a resimulation of the central air system: i.e., a second pass in loop 2.

In loop 3 a call to *ManageControllers* simulates controller action and checks for controller convergence. If convergence is achieved loop 3 is exited.

After all the controllers on a loop are converged, steps 5 & 6 are repeated one more time to ensure all the components on the loop have final values.

At the end of the primary air system simulation a call to *UpdateHVACInterface* passes the primary air system outlet node data to the zone equipment inlet nodes. If the data across the supply side – demand side gap doesn't match to within a preset tolerance, the flag *SimZoneEquipment* is set to *true* to ensure that the zone equipment side gets resimulated. Finally a flag indicating whether the economizer is active is set. This flag is used at a higher level to decide whether the primary air system needs to be resimulated if an HVAC component is calling for economizer lockout.

Outdoor Air System

The *AirLoopHVAC:OutdoorAirSystem* is a subsystem of the primary air system (object *AirLoopHVAC*). To the primary air system it appears to be a single component. Within the subsystem are components and controllers very much like the primary air system. The purpose of the outside air system is to simulate the splitting of the return air into relief and recirculated air; mix the recirculated and outside air to obtain the mixed air (all accomplished by the *OutdoorAir:Mixer*); simulate the components in the outside air stream; control the components and outside air flow rate (economizer control). The outside air system differs from the primary air system in that it is not described in terms of splitters, mixers, and branches. Instead the components are simply listed in simulation order in an *AirLoopHVAC:OutdoorAirSystem:EquipmentList* input object. The components other than the *OutdoorAir:Mixer* are assumed to be in the outside air stream. The connectivity is defined by upstream component's outlet node = downstream component's inlet node. The *OutdoorAir:Mixer* should be the first component in the *AirLoopHVAC:OutdoorAirSystem:EquipmentList* object.

Simulation

The data for and simulation of the outside air system are contained in the *MixedAir* module. This includes the data and simulation of the *OutdoorAir:Mixer* and the *Controller:OutdoorAir*. The simulation of the outside air system is straightforward.

1. All controllers other than *Controller:OutdoorAir* are moved up to the primary air system level, where they are simulated with the primary air system controllers.
2. The *Controller:OutdoorAir* is simulated.
3. The components contained in the outside air system are simulated in the order of their input in the *AirLoopHVAC:OutdoorAirSystem:EquipmentList*.

Outdoor Air Mixer

The *OutdoorAir:Mixer* is the fundamental component of the outside air system. There should always be one *OutdoorAir:Mixer* in an *AirLoopHVAC:OutdoorAirSystem*. The outside air mixer splits the primary air system (*AirLoopHVAC*) return air into relief and recirculated air streams. Then it mixes the outside air stream with the recirculated air stream to obtain the mixed air stream. The outdoor air mixer has 2 inlet flow nodes: the return air node and the outside air node. It has 2 outlet flow nodes: the relief air node and the mixed air node.

Simulation

Given the air mass flow rates and conditions (humidity ratio, specific enthalpy) at the return air node, relief air node, and outside air node, the simulation uses mass and energy balance to calculate the air mass flow rate, humidity ratio, and specific enthalpy at the mixed air node. The mixed air temperature is calculated from the mixed air humidity ratio and specific enthalpy using the EnergyPlus psychrometric routine *PsyTdbFnHW*.

Zone Equipment Simulation

When the EnergyPlus HVAC simulation manager needs to simulate the zone equipment side of the air loop it calls *ManageZoneEquipment*, the zone equipment simulation manager subroutine. Like the other managers, *ManageZoneEquipment* has a very simple structure:

```

IF (GetInputFlag) THEN
  CALL GetZoneEquipment
  GetInputFlag = .FALSE.
END IF

CALL InitZoneEquipment(FirstHVACIteration)

IF (ZoneSizingCalc) THEN
  CALL SizeZoneEquipment
ELSE
  CALL SimZoneEquipment(FirstHVACIteration, SimAir)
END IF

CALL RecordZoneEquipment(SimAir)

CALL ReportZoneEquipment

SimZone = .False.

RETURN

```

- 1) If the user input data has not been input, get the data and store it in the zone equipment data structures
- 2) Perform zone equipment initialization calculations.
- 3) If calculation of the design zone air flow rates and loads needs to be done, do it. The results of this calculation are stored in the zone sizing data structures and used by the component automatic sizing algorithms and the central system sizing calculations.
- 4) Otherwise simulate all of the zone equipment.
- 5) Transfer the zone equipment outlet node data to the inlet nodes of the primary air systems and check for convergence (done in *RecordZoneEquipment* by calling *UpdateHVACInterface*).

Input data

The input specifying a set of zone equipment consists of:

- 1) the ZoneHVAC:EquipmentConnections object data;
 - (a) the zone connection to the air loop – air inlet nodes, exhaust nodes, outlet node, zone node;
 - (b) the components serving each zone – air terminal units, fan coils etc.;
- 2) zone supply air path data;
 - (a) zone splitters and supply plenums;
- 3) zone return air path data;

(a) zone mixer and return plenums;

Initialization Calculations

One Time Calculations

There are no one time calculations for zone equipment

Begin Environment Initializations

For each controlled zone initialize the zone inlet, exhaust and zone nodes to standard conditions. Namely:

$$T_{node} = 20^{\circ} C$$

$$W_{node} = W_{oa}$$

$$h_{node} = \text{PsyHFnTdbW}(20.0, W_{oa})$$

$$\dot{m}_{node} = 0$$

$$Qu_{node} = 1.0$$

$$P_{node} = P_{oa}$$

where W_{oa} is the humidity of the outside air; PsyHFnTdbW is the EnergyPlus psychrometric function for enthalpy h , given temperature and humidity ratio; p_{oa} is the pressure of the outside air; and Qu is quality.

System Time Step Initializations

For each controlled zone, set the exhaust nodes conditions equal to the zone node conditions; except set the exhaust nodes mass flow rate and min and max available mass flow rates to zero.

Simulation

The subroutine *SimZoneEquipment* does the actual simulation of all the zone equipment. Note that this includes components that are part of the demand side of an air loop as well as components that are independent of any air loop.

For zone equipment components that are part of the demand side of an air loop the simulation sequence is effectively performed in the direction of the air flow. First the supply air plenums and zone splitters are simulated in their input order. Then the air terminal units are simulated followed by the zone return plenums and zone mixer. Each air terminal unit sets its inlet node to the air mass flow rate required to satisfy its zone load. These mass flow rates are then passed back upstream to the air loop demand-side inlet nodes at the end of each zone equipment simulation sequence. These demand-side inlet mass flow rates are then used as mass flow rate setpoints in the air loop supply-side simulation.

If multiple air-conditioning components are attached to a zone, the components are simulated in the order specified by the user assigned priority given in the ZoneHVAC:EquipmentList object.

For each full air loop there should be 1 supply air path for each primary air system outlet (i.e. 1 for single duct systems, 2 for dual duct systems). For each full air loop there should be one return air path. The supply air paths consist of any combination of zone splitters and zone supply air plenums as long as it forms a tree structure with the supply air path inlet node the

root and the air terminal unit inlet nodes as the leaves. The return air path configuration is limited to a single mixer; there may be multiple return plenums.

- (1) Loop over all the supply air paths.
 - (a) Loop over each component (supply plenum or zone splitter) on the supply air path and simulate each component. The components are simulated in input order.
- (2) Loop over all the controlled zones.
 - (a) Set the required system output.
 - (b) Loop over the components serving the zone in the user prioritized order.
 - (i) Simulate each component.
 - (ii) Increment the required system output.
- (3) Loop over all the supply air paths
 - (a) Loop over the components on each supply air path in reverse input order. This reverse order simulation passes the air terminal units inlet mass flows back upstream to the return air path inlet node.
 - (b) Check to see if the supply air path inlet node mass flow rate has changed. If it has set the *SimAir* flag to *true*. This signals the HVAC manager that the supply-side of the air loop needs to be resimulated.
- (4) Calculate the zone air flow mass balance – the zone inlet and exhaust mass flow rates are summed and the zone node and return air node mass flow rates are determined by a mass balance for each zone.
- (5) Calculate the conditions at each zone return air node. Here energy not included in the zone energy balance such as light-heat-to-return-air is added to the return nodes of the controlled zones.
- (6) Loop over all of the return air paths.
 - (a) Loop over each component (return plenum or zone mixer) on the return air path and simulate each component.

This completes a single simulation sequence of all the zone equipment.

Air Path Components

Overview

The air path from the outlet of an AirLoopHVAC (the supply side of a full air loop / central air system) to the zone air distribution units (terminal units) is specified in a AirLoopHVAC:SupplyPath. The AirLoopHVAC:SupplyPath consists of AirLoopHVAC:ZoneSplitters and AirLoopHVAC:SupplyPlenums hooked together outlets to inlet.

The air path from the zone return nodes to the AirLoopHVAC inlet node is described in a AirLoopHVAC:ReturnPath object. The AirLoopHVAC:ReturnPath consists of AirLoopHVAC:ZoneMixers and AirLoopHVAC:ReturnPlenums hooked together outlet to inlets.

Zone Supply Air Path

The AirLoopHVAC:SupplyPaths are simulated just before the zone equipment is simulated and just after the zone equipment is simulated (in module ZoneEquipmentManager, subroutine SimZoneEquipment). In both cases the simulation consists of looping over all the AirLoopHVAC:SupplyPaths and simulating each component (AirLoopHVAC:ZoneSplitter or AirLoopHVAC:SupplyPlenum) in the path. For the “just before” simulation a flag *FirstCall* is set to *True* which tells the zone splitter and zone supply plenum models to do a downstream simulation (pass component inlet conditions to component outlets, except for air mass flow rate). For the “just after” simulation, *FirstCall* = *False*, which tells the components to do an

upstream simulation (set the component inlet air mass flow rate to the sum of the component outlet air mass flow rates, do nothing with other properties).

Zone Splitter

The AirLoopHVAC:ZoneSplitter divides an inlet air stream into multiple outlet streams. Its data and simulation model are encapsulated in the module *SplitterComponent*. There are two simulation modes. If *SimAirLoopSplitter* is called with input parameter *FirstCall = True* then the splitter is simulated in downstream mode. The outlet air streams' humidity ratio, pressure, enthalpy, and temperature are set to the conditions on the inlet air stream. If *SimAirLoopSplitter* is called with input parameter *FirstCall = False*, the splitter is simulated in the upstream mode. In this case the inlet air stream air mass flow rate is set to the sum of the outlet air mass flow rates. The same calculation is done for the maximum available and minimum available air flow rates.

Zone Supply Plenum

The AirLoopHVAC:SupplyPlenum acts analogously to the AirLoopHVAC:ZoneSplitter. The only difference is that the AirLoopHVAC:SupplyPlenum is associated with a Zone for which it sets the supply air flow rate and from which it gets its outlet air conditions. The AirLoopHVAC:SupplyPlenum divides an inlet air stream into multiple outlet streams. Its data and simulation model are encapsulated in the module *ZonePlenum*. There are two simulation modes. If *SimAirZonePlenum* is called with input parameter *FirstCall = True* then the plenum is simulated in downstream mode. The outlet air streams' humidity ratio, pressure, enthalpy, and temperature are set to the zone conditions. If *SimAirZonePlenum* is called with input parameter *FirstCall = False*, the plenum is simulated in the upstream mode. In this case the inlet air stream air mass flow rate is set to the sum of the outlet air mass flow rates. The same calculation is done for the maximum available and minimum available air flow rates.

Zone Return Air Path

The AirLoopHVAC:ReturnPaths are simulated after all the zone equipment is simulated (in module *ZoneEquipmentManager*, subroutine *SimZoneEquipment*). The simulation is accomplished by a call to *SimReturnAirPath* (in module *ReturnAirPathManager*). The simulation just consists of looping over all the AirLoopHVAC:ReturnPaths and simulating the components (AirLoopHVAC:ZoneMixer or AirLoopHVAC:ReturnPlenum) in each path.

Zone Mixer

The AirLoopHVAC:ZoneMixer combines multiple inlet air streams into a single outlet air stream. Its data and simulation model are encapsulated in the module *MixerComponent*. The following mass and energy balance equations are used.

$$\dot{m}_{air,out} = \sum_{i=1,n} \dot{m}_{air,in,i}$$

$$\dot{m}_{air,out} \cdot W_{air,out} = \sum_{i=1,n} \dot{m}_{air,in,i} \cdot W_{air,in,i}$$

$$\dot{m}_{air,out} \cdot h_{air,out} = \sum_{i=1,n} \dot{m}_{air,in,i} \cdot h_{air,in,i}$$

$$\dot{m}_{air,out} \cdot P_{air,out} = \sum_{i=1,n} \dot{m}_{air,in,i} \cdot P_{air,in,i}$$

$$T_{air,out} = PsyTdbFnHW(h_{air,out}, W_{air,out})$$

Where \dot{m} is air mass flow rate, W is humidity ratio, h is specific enthalpy, P is pressure, and T is temperature. $PsyTdbFnHW$ is the EnergyPlus psychrometric function for drybulb temperature as a function of enthalpy and humidity ratio. The air mass flow rate calculation is also done for the maximum and minimum available mass flow rates.

Zone Return Plenum

The AirLoopHVAC:ReturnPlenum acts analogously to the AirLoopHVAC:ZoneMixer. The only difference is that the AirLoopHVAC:ReturnPlenum is associated with a Zone for which it sets the supply air flow rate and from which it gets its outlet air conditions. The AirLoopHVAC:ReturnPlenum combines multiple inlet air streams into a single outlet air stream. Its data and simulation model are encapsulated in the module *ZonePlenum*. The outlet air mass flow rate is obtained from

$$\dot{m}_{air,out} = \sum_{i=1,n} \dot{m}_{air,in,i}$$

The outlet air temperature, enthalpy, humidity ratio and pressure are set to the zone conditions. Finally, calculation loops over the air distribution units connected to the zones that feed air to the plenum and adds in to the outlet air mass flow rate the leakage air flow rates from the upstream leaks defined in the ZoneHVAC:AirDistributionUnit inputs.

Plant Load Profile

The LoadProfile:Plant object is used to simulate a scheduled demand profile. This can be useful when the building loads are already known. Demanded load and flow rate are schedules specified in the object definition. The load profile can specify heating and cooling loads. Cooling loads are entered as negative numbers. The actual load met is dependent on the performance of the supply loop components.

The LoadProfile:Plant object must be connected on the demand side of the plant loop. It should be located on a Branch with its control type set to "ACTIVE." If desired, multiple LoadProfile:Plant objects can be combined in series and/or parallel.

Calculation Model

The LoadProfile:Plant object calculates the outlet water temperature based on the inlet water temperature from the plant loop and user inputs for the scheduled plant load and the requested flow rate. The calculation can be expressed with the equation:

$$T_{out} = T_{in} - \frac{Q_{load}}{\dot{m}c_p} \quad (208)$$

where

T_{out} = the outlet water temperature

T_{in} = the inlet water temperature

Q_{load} = the scheduled plant load

\dot{m} = the water mass flow rate

c_p = the specific heat of water

The user requested flow rate is not always available from the plant loop. The actual flow rate used in the calculation is the lesser of the user requested value and the plant available value.

Note that the LoadProfile:Plant object can still request and receive flow even if the scheduled plant load is zero. In this case the outlet temperature will be the same as the inlet temperature. This allows users to drive the plant loop flow without necessarily affecting the loop temperature.

For reporting purposes the energy consumption of the object is calculated using the equation:

$$E = Q_{load} \Delta t \quad (209)$$

where

E = the energy consumption

Q_{load} = the scheduled plant load

Δt = the time step interval

Plant/Condenser Loops

Integration of System and Plant

In order to integrate the air handling system simulation with the zones simulation, methods were developed to model the system air loop and its interactions with the zones due to temperature controls and the relative difference between the zone and supply air temperatures. A similar situation is encountered when integrating the central plant simulation. Typically, the central plant interacts with the systems via a fluid loop between the plant components and heat exchangers, called either heating or cooling coils. In EnergyPlus the performance of the systems and plant are interdependent because the simulations are combined. The plant outputs must match the system inputs and vice versa. That is, the temperature of the chilled water leaving the plant must equal the temperature of the water entering the coils, and the chilled water flow rate must satisfy mass continuity. In addition, coil controls are usually necessary to ensure that the values of chilled water flow variables entering and leaving the coil remain in a reasonable range. The specific controls vary from application to application but two common possibilities are: maintaining a constant coil leaving air temperature or limiting the water temperature rise across the coil.

Current Primary System Modeling Methodology

There are three main loops within the HVAC simulation in the new program: an air loop, a plant loop, and a condenser loop. The air loop is assumed to use air as the transport medium as part of an air handling system while the plant and condenser loops may use a fluid of the user's choosing (typically water). A user may have any number of each type of loop in a particular input file. There are no explicit limits on the number of loops within the program—the user is only limited by computer hardware. Execution speed will naturally vary with the complexity of the input file.

Main loops are further divided into “sub-loops” or “semi-loops” for organizational clarity and simulation logistics (see Figure “Connections between the Main HVAC Simulation Loops and Sub-Loops”). These sub-loops are matched pairs that consist of half of a main loop. Plant and condenser loops are broken into supply and demand sides. The plant demand sub-loop contains equipment that places a load on the primary equipment. This might include coils, baseboards, radiant systems, etc. The load is met by primary equipment such as chillers or boilers on the plant supply sub-loop. Each plant supply sub-loop must be connected to a plant demand sub-loop and vice versa. A similar breakdown is present on condenser loops where the demand side includes the water side of condensers while the supply side includes condenser equipment such as cooling towers.

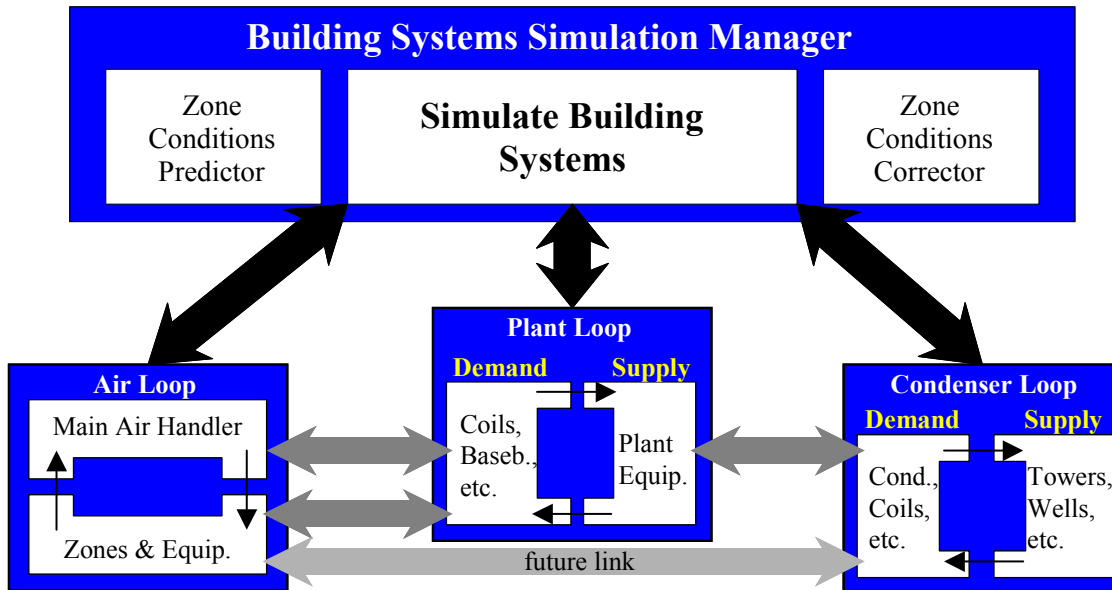


Figure 94. Connections between the Main HVAC Simulation Loops and Sub-Loops.

The breakdown into sub-loops allows for better handling and control of information and simulation flow throughout the program. Direct connections between the sub-loops of the air, plant, and condenser loops are enhanced by indirect connections between the various main loop types. For example, coils (heating or cooling) are in reality heat exchangers with an air and a water or refrigerant side. The air side of the coil is handled within the air loop where the control of the device is also maintained. The fluid side of the coil is handled within the plant demand side, which passes the energy requirements of the coil on to the plant supply side. All loops are simulated simultaneously, though sub-iteration loops are maintained between the two sides of any loop to speed convergence. Overall iterations ensure that the results for the current time step are balanced and updated information has been passed to both sides of the sub-loops as well as across to the other side of indirect connections such as coils.

Branches further divide the sub-loops into groups as they would appear within any HVAC system. Elements can be defined in series, in parallel, or both with some restrictions. Figure “Branch Layout for Individual HVAC Sub-Loops” provides an overview of a generic sub-loop representation. Branches are defined as individual legs within the loop structure. Thus, the segment between point A and point B is defined as a branch, as is the section between points E and F. There may be multiple sections (C1 to D1 through Cn to Dn) in between the splitter and mixer.

Each sub-loop may only have one splitter and one mixer. Thus, equipment may be in parallel between the mixer and splitter, however, within any branch, there can only be elements in series and not in parallel. The topology rules for individual sub-loops allow a reasonable amount of flexibility without requiring a complicated solver routine to determine the actual flow and temperature conditions. Note that since plant supply and demand are broken up into two separate sub-loops, chillers or boilers may be in parallel to each other in the supply side and coils may be in parallel to each other on the demand side. Thus, the restriction of only a single splitter and mixer on a particular sub-loop does not limit the normal configurations. Also, a sub-loop does not require a splitter or mixer if all equipment on the sub-loop is simply in series—this would correspond to a single branch that would define the entire sub-loop.

In addition, to avoid the need for complex solver routines, there are some restrictions on the placement of pumps within particular sub-loops. Currently, pumps are not allowed on the condenser demand sub-loop. In plant demand sub-loops pumps can be placed only in the

inlet branch. This will allow simulation of primary-secondary systems. A pump bank can be used and scheduled appropriately to simulate branch pumps in plant demand side. For condenser supply side sub-loops, a pump must be the first component on the branch between point A and B and may not be placed anywhere else within the sub-loop topology. For plant supply side sub-loops, pumps can either be the first component on this inlet branch (between point A and B in the figure below) or the first component on the parallel branches (between points Ci and Di). A pump that is the first component on the first branch (between A and B) is termed a “loop pump” while any pump in the parallel section (between Ci and Di) is termed a “branch pump”. Plant supply side sub-loops may have either a single loop pump or one or more branch pumps, but with the current release of EnergyPlus it is not yet possible to have both a loop pump and branch pumps on a single supply side sub-loop. For more information on pumps and pump placement rules, see the section on Pumps in this document.

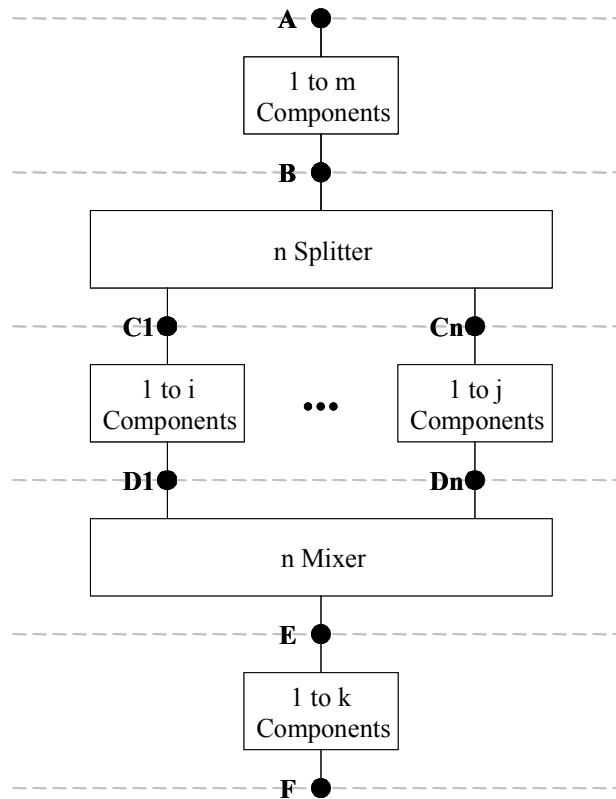


Figure 95. Branch Layout for Individual HVAC Sub-Loops

Essentially, each branch is made up of one or more components linked together in series. The branch has an information node containing properties of the loop (temperature, enthalpy, flow rate, etc.) at the beginning and end of the branch as well as between components. Components on the branch take the conditions of the node at their inlet and use that information as well as overall control information to simulate the component and write the outlet data to the node following the component. This information is then used either by the next component on the branch or establishes the outlet conditions for the branch.

Plant Flow Resolver

Overview of the Plant Flow Resolver Concept

One of the most important aspects of the solution procedure within the plant and condenser loops of the new program is the method used to solve the various sub-loops. This involves

making the supply side meet a particular load based on the simulation of the demand side loops. Load distribution is an issue that must be addressed as well as how flow rates are adjusted and temperatures are updated. These issues are discussed in the next several subsections, and the algorithms described are important to how the HVAC simulation functions.

In the first step, the loop manager would call the appropriate module to simulate (in order) all of the components on each branch of the loop except for splitters and mixers. In this step, each component would set the conditions at the outlet node including temperature, flow rate, maximum allowed (design) flow rate, minimum allowed (design) flow rate, maximum available flow rate, and minimum available flow rate. This would be based purely on the component's own control scheme and thus each component would be free to request as much (or as little) flow as desired.

In the second step, the loop manager would resolve the flow at all nodes and through all branches of the local loop. The components are then simulated with the corrected flows. For this iteration, the flow resolver sets the flow rate through each loop component.

Pump Control for Plant and Condenser Loops.

The pump is quite simply the component that drives the flow (also see Pumps). How it reacts depends on several different conditions. In total, there are three different decision variables, two of which are defined by user input. These three deciding factors are whether the pump is constant or variable speed, whether the pump operation is continuous or intermittent, and whether or not there is a load on the loop. The pump is simulated first on the supply side loop after the demand side loop has determined what the demand on the loop will be. The load is simply calculated by multiplying the requested flow rate from the demand side by the difference between the enthalpy at the supply side inlet and the enthalpy that corresponds to the current loop setpoint temperature. This setpoint temperature is the fluid temperature that one is attempting to maintain at the outlet of the supply side and can be scheduled to different values on an hourly basis.

The operation of a constant speed pump is fairly straightforward. If the user designates a constant speed pump that is operating continuously, the pump will run regardless of whether or not there is a load. This may have the net effect of adding heat to the loop if no equipment is turned on. If the pump is constant speed and operates intermittently, the pump will run at its capacity if a load is sensed and will shut off if there is no load on the loop.

A variable speed pump is defined with maximum and minimum flow rates that are the physical limits of the device. If there is no load on the loop and the pump is operating intermittently, then the pump can shutdown. For any other condition such as the loop having a load and the pump is operating intermittently or the pump is continuously operating (regardless of the loading condition), the pump will operate and select a flow somewhere between the minimum and maximum limits. In these cases where the pump is running, it will try to meet the flow request made by demand side components.

In many cases, the first estimate of flow requested by the demand side tends to be fairly accurate and the flow rate does not vary in subsequent iterations. However, because there is the possibility that the coils or some other component might request more flow in future iterations during the same time step, the program must not only set flow rates but also maintain a record of the current maximum and minimum flow rate limits. This information is important not only to the pump itself but also to other pieces of equipment which may control their flow rates and thus require knowledge of the limits within which they may work. In general, the decisions on what to set the maximum and minimum flow rates is directly related to the type of pump (constant or variable speed). For constant speed pumps, the maximum and minimum flow rate values are the same and thus if the flow requested does not match this, the other components must either deal with the flow or a bypass branch must be available to handle the excess flow. For variable speed pumps, the maximum and minimum flow rates are set by the user-defined limits.

Plant/Condenser Supply Side

Component models, such as boilers, chillers, condensers and cooling towers are simulated on the supply side of the plant and condenser loops. In order to allow specification of realistic configurations, the plant and condenser supply side loop managers were designed to support parallel-serial connection of component models on the loop. In addition, loop managers were designed to support both semi-deterministic models (e.g. the parameter estimation models of the ASHRAE Primary Toolkit [Pedersen 2001]) and “demand based” models (e.g. the performance map models of BLAST and DOE2.1E). As a result, the loop manager must be able to simulate models that require the mass flow rate as an input and models that calculate the mass flow rate as an output—sometimes in the context of a single loop configuration.

In order to achieve these design criteria without resorting to a pressure based flow network solver in the HVAC portion of the code, a rules-based “flow resolver” was developed for the EnergyPlus plant and condenser supply side managers. The flow resolver is based on the following assumptions and limitations:

- Each loop is only allowed to have a single splitter and a single mixer
- Due to the fact that there can only be one splitter and one mixer on a given loop, it follows logically that there can be at most one bypass on each loop
- No other components may be in series with a bypass, i.e., a branch that contains a bypass may have no other equipment on that branch
- Equipment may be in parallel only between the splitter and mixer component of a loop or between one of those types of equipment and the loop inlet/outlet nodes
- Equipment may be hooked together in series in each branch of the loop
- Flow rates on individual branches will be controlled using maximum and minimum available flow rate limits

The flow resolver employs a simple predictor-corrector algorithm to enforce mass continuity across the plant loop splitter as shown in the following figure.

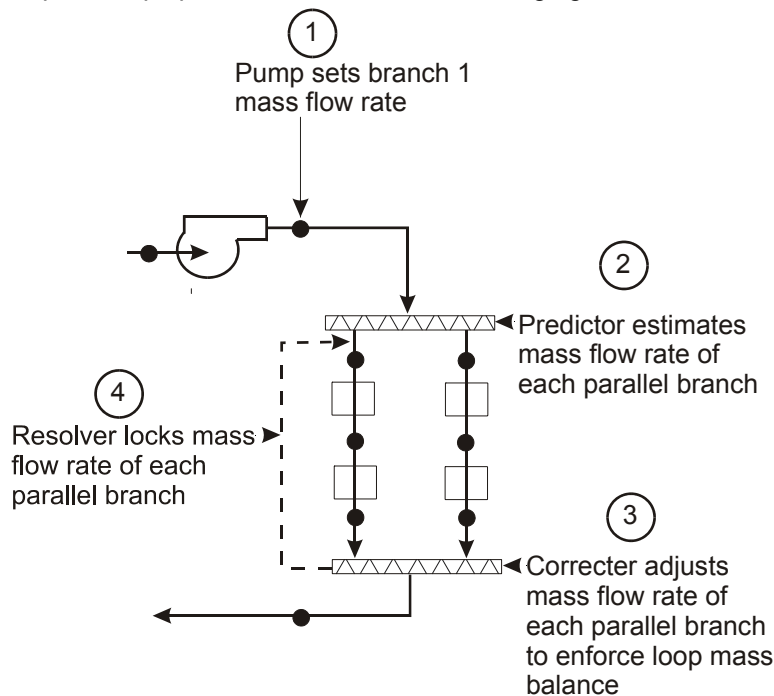


Figure 96. Plant/Condenser Supply Side Solution Scheme.

As previously discussed, the pump establishes the total loop mass flow rate by setting the flow in the first supply side branch. In the second step, a predictor algorithm polls each piece

of equipment on the loop and “predicts” branch mass flow rates based on the requested flow rate for each. The loop manager calls the appropriate module to simulate (in order) all of the components on each branch of the loop except for splitters and mixers. In this step, each component sets the conditions at its outlet node including temperature, flow rate, maximum allowed (design) flow rate, minimum allowed (design) flow rate, maximum available flow rate, and minimum available flow rate. These predicted values are based purely on the component’s own control scheme and thus each component is free to request as much (or as little) flow as desired.

Each component is tagged in the user input file as an ACTIVE, SERIESACTIVE, PASSIVE or BYPASS type of model. An ACTIVE type describes a demand based plant model that calculates mass flow rate as an output. An ACTIVE component when OFF will shut down the whole branch irrespective of the type of other components on the branch. A SERIESACTIVE component functions exactly like an ACTIVE component except that it will not shut down the whole branch when the component is OFF. The flow resolution algorithm is same for both ACTIVE and SERIESACTIVE components and in the rest of the document description of one type will fit the other type too. A PASSIVE type describes a semi-deterministic model that is simulated with the mass flow rate as an input. The BYPASS type designates a loop bypass.

The predictor algorithm first establishes the desired flow rate of each branch by searching for ACTIVE components on the branch. The first ACTIVE component in simulation order sets the desired branch flow. Branches with only PASSIVE components require a flow rate between the minimum and maximum allowable branch flow. Branches with a BYPASS component have a branch flow only when all other branches combined cannot handle the entire loop flow.

In the third step, the loop manager makes any necessary “corrections” to the requested branch flows in order to enforce overall continuity on the loop. If mass conservation allows all ACTIVE branches to be satisfied, then the remaining flow is divided between the PASSIVE branches and as a last resort, the BYPASS. If there is insufficient flow to meet the branch demand, ACTIVE branch requests are met first in the order that the branches appear in the branch list in the input file.

Plant/Condenser Demand Side

The plant and condenser demand side are simulated in a different manner than the supply sides because in reality there are no components to simulate or control. On the supply sides, there is a load management scheme and other constraints that must be resolved. On the demand sides, all of the components have already been simulated and controlled by the air loop, the zone equipment, or the plant supply side. Thus, the demand side management module only needs to resolve the actual flow rate through each section or branch of the sub-loop and also monitor the maximum and minimum flow rates that are available.

The flow rate is resolved first for each individual branch. For every branch, the program cycles through each node on the branch and determines what the flow requests and flow limits are. The most restrictive flow constraints are assumed to be valid for the entire branch regardless of component type. Since there may be several components in series on a particular branch, there is also a defined scheme for assigning priority to components that will have the ability to control the flow. The user may specify individual components as either active or passive. Active components are given highest priority for requesting a particular flow rate. If there is more than one active component on a particular branch, then it is assumed that the first active component on the branch is the highest priority and dictates the flow request.

Once all of the branches have set their flow rates and constraints, the splitter and mixer must resolve the various flow requests. In the demand side scheme, the mixer and any branch following the mixer is completely passive. Thus, all of the control happens at the splitter. The splitter first attempts to sum the maximum and minimum constraints from all of the active branches coming out of the device and compares those to the constraints that are valid for the branch leading into the splitter. When there is a mismatch between the outlet constraints

and the inlet constraints, the simulation will defer to the inlet constraints due to the fact that the pump is in reality controlling flow on the loop. Since the constraints of the pump would be passed across to the demand side from the supply side, an assumption is made that the coils or other demand side components must live within the bounds of the pump.

Once the flow has been resolved at the splitter, the branch flow rates and constraints between the splitter and mixer can be adjusted, if necessary. In some cases, this will be mandatory to maintain a mass balance at the splitter. When the flow rate coming out of the splitter does not match the active branch requests, individual branch flow rates must be adjusted to provide for the extra flow or the “flow deficit”. When there is extra flow, the excess flow is sent through any bypass branch first and then is sent to passive branches in reverse order of their appearance in the splitter outlet list. When all of these branches have been exhausted and there is still excess flow, flow will be increased to the active branches, also in reverse order. The reverse order guarantees that the branch appearing first has the highest priority to receive the flow rate it has requested. For the condenser loop, if there is not enough flow to meet all active branch requests (i.e., a “flow deficit”), then the flow rates through the bypass and passive branches are set to zero. The flow rates through the active branches will then be decreased in reverse order until the splitter outlet flow rate matches the available flow at the splitter inlet. For a plant loop flow deficit, the bypass and passive branch flows are also set to zero, and flow rates for each active branch are calculated as follows:

$$\dot{m}_{br} = \frac{\dot{m}_{br_request}}{\dot{m}_{tot_request}} * \dot{m}_{tot_available}$$

where:

\dot{m}_{br} = final resolved branch flow rate

$\dot{m}_{br_request}$ = requested branch flow rate

$\dot{m}_{tot_request}$ = total loop mass flow rate request

$\dot{m}_{tot_available}$ = total loop mass flow rate available

It is also necessary to monitor the flow constraints at the branches and components since once the flow rates are changed, the components must be resimulated by the controlling loop (air loop, zone equipment, or plant supply side). The controllers for these components must know if the constraints have been modified so that the simulation does not toggle between a component requesting a flow that the pump cannot meet and the pump then resetting the flow to what it can provide. Note that once a flow rate for any component has changed that this signals the need to resimulate any sub-loop to which it might have an indirect connection. Currently, this means that if a flow rate on the plant demand side changes, the simulation must recalculate the conditions on both the air loop and zone equipment sub-loops since coils and other equipment could be on either side of the main air loop. Similarly, if the condenser demand side simulation results in a change in flow rate through a chiller condenser, then the plant supply side must be triggered to perform its calculations again. Care has been taken to avoid cases where the various sub-loops might simply keep triggering the resimulation of their indirect connections in an infinite loop.

Temperature Resolution

The transition from load or energy based plant models to a loop based arrangement makes variables of both the flow rate and the fluid temperature. This means there are more degrees of freedom that must be controlled. The flow resolver concept discussed previously controls the flow rates through the components and maintains an overall mass flow balance through the loop. However, the temperatures still need to be controlled. A purely iterative procedure can be expected to converge to the appropriate loop temperatures, but the procedure can become slow to converge under conditions where the demand changes rapidly or the supply

components may not have enough capacity to meet the system demand. This situation is somewhat analogous to that existing in the link between the zone and the air system. In that case, the convergence and stability of the iterative solution was greatly improved by adding the thermal capacitance of the zone air and other fast responding mass within the zone. Based on that experience, it was decided to add thermal capacitance to the plant loop and benefit from the added stability. Because the thermal capacitance in the zone/system interaction is relatively small, it was necessary to use a third order numerical solution there. Since the plant loop thermal capacitance is higher, a simple first order solution has been found to be satisfactory.

To implement the capacitance, each loop is assigned a fluid volume as user input. This is used to determine a capacitance concentrated in the supply side outlet node. If the loop setpoint cannot be maintained, this node becomes an energy storage location and its temperature reflects the current capability of the supply side. The size of the thermal capacitance affects the speed of recovery from situations where the setpoint was not maintained. The user must estimate a fluid volume based on the size of the pipes in the loop. However rough estimates seem to be sufficient. The supply side outlet node temperature and the demand side inlet temperature proceed through smooth paths from one time step to the next. No energy is lost or gained because of storage in the loop capacitance. Once setpoint temperature is reached, the storage effects are not involved.

Loop Capacitance

In realistic conditions there is often some delay between changes in supply conditions and corresponding changes at demand side components due to the transport of fluid round the loop having a finite velocity. Plant and condenser loops include a simple loop capacitance model to simulate this effect. A time delay effect is achieved by including a 'well-stirred tank' between the supply outlet and the demand inlet nodes as indicated in Figure 97. The delay effect depends on the loop capacitance parameter (loop volume, m^3) and the loop flow rate.

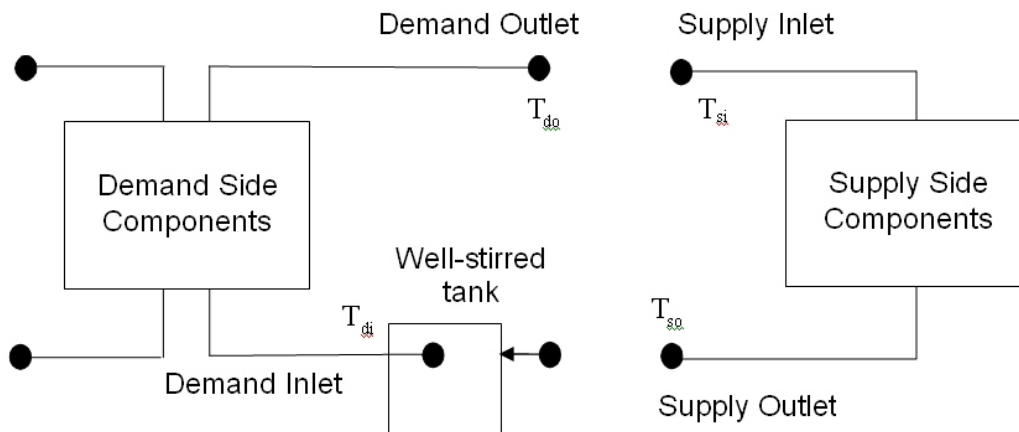


Figure 97. Demand and Supply Side Loops

The demand inlet temperature, (T_{di}) is calculated at the end of each iteration of the HVAC simulation, based on the loop mass (density x loop capacitance), the calculated supply side outlet, (T_{so}) and the demand inlet temperature from the previous timestep ($T_{di,old}$). The 'well-stirred tank' model is formulated from a mass balance as follows,

$$M(T_{di,new} - T_{di,old}) = \dot{m}_s t_{sys} 3600.(T_{di,new} - T_{di,old})$$

Where:

T_{di-old} Previous system time-step demand inlet temperature [°C]

T_{di-new} Current demand side inlet temperature [°C]

\dot{m}_s Maximum expected supply side mass flow rate [Kg/s]

t_{sys} System time step [hr]

T_{so} Supply side outlet temperature [°C]

M Mass of the water in the loop [Kg]

Loop capacitance (m^3) could be calculated from pipe size data but this is not usually known. If zero capacitance is specified the above formulation reduces to an instantaneous update in demand update temperature and the demand inlet temperature becomes the supply outlet temperature at the previous time step. If a very large capacitance is specified unrealistic time delay may result and there may be poor response to changes in loop setpoint temperature. The loop capacitance 'autocalculate' option gives reasonable values and is calculated as follows,

$$Capacity = \frac{\dot{m}_s * t_{sys} * 3600}{0.8\rho}$$

The Plant Flow Resolver Input

The input specifically related to the flow resolver consists of the plant BranchList and the plant ConnectorList as shown in the Input Output Reference. User defined names link the plant loop to its branches (contained in the BranchList) and define the loop splitters and mixers contained in the ConnectorList. The Connector:Splitter and Connector:Mixer syntax in turn define the relative connection of the branches to each other on the loop.

The Branch definition is input in simulation and connection order for all of the components on the branch. The simulation assumes that the inlet node of the first component listed on the branch is the branch inlet node and the outlet node of the last component listed on the branch is the branch outlet node. Examples of all the input syntax is shown in the Input/Output Reference for the appropriate object.

The flow control algorithm described in the next section determines the flow through the loop uses the branch control types: ACTIVE, PASSIVE, and BYPASS. An ACTIVE control type describes a demand based plant model that calculates mass flow rate as an output of the model. A PASSIVE control type describes a model that is simulated with the mass flow rate as a model input. The BYPASS control type designates a loop bypass.

The Plant Flow Resolver Algorithm

The flow resolver is called from the plant and condenser loop managers as shown below. All branches are simulated with desired flow rates, then the flow resolver sets the branch flows and the loops are simulated again. Finally the loop mixer is updated and the mixer outlet branch is simulated.

- Simulate Components with the flow rate "requested" to meet loop demand.
- Check Branch flows (downstream of splitter) to ensure that continuity is not being violated.
- Adjust branch flows if necessary to achieve a mass balance on the loop
- Resimulate Components with adjusted flows.

```
DO All Branches
  CALL Sim Plant Equipment
  IF (Branch is Splitter inlet ) CALL Update Splitter
END DO
```

CALL Solve Flow Network

```

DO All Branches
  CALL Sim Plant Equipment
  IF (Branch is Splitter inlet ) CALL Update Splitter
END DO

CALL Update Mixer

CALL Sim Plant Equipment on Mixer Outlet Branch

```

The flow resolver algorithm first establishes the desired flow rate of each branch by searching for ACTIVE components on the branch. The first ACTIVE component in simulation order sets the desired branch flow. Branches with only PASSIVE components require a flow rate between the minimum and maximum allowable branch flow. Branches with a BYPASS component have a branch flow only when all other branches combined cannot handle the entire loop flow.

If mass conservation allows all ACTIVE branches to be satisfied, then remaining flow is divided between the remaining PASSIVE branches and as a last resort, the BYPASS. If there is insufficient flow to meet the branch demand, ACTIVE branch requests are met first in the order that the branches appear in the branch list in input.

Summary of Load Distribution Schemes

Three load distribution schemes are employed in EnergyPlus. The figure below illustrates the plant load distribution algorithm. The total loop demand is calculated and used in the **ManagePlantLoopOperation** routine to determine which equipment is available based on the supervisory control scheme specified by the user. Once all available components have been identified the loop demand is distributed to the available components based on the user specified load distribution scheme.

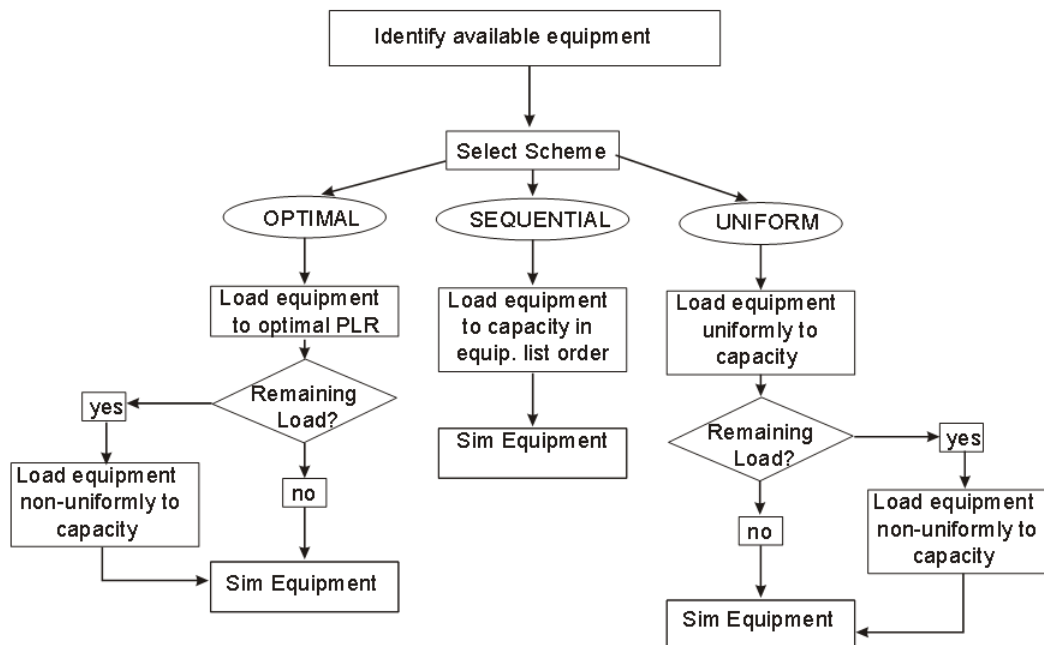


Figure 98. Load Distribution Scheme

The OPTIMAL scheme first loads each component to its optimal part load ratio (specified in input). Any remaining loop demand is distributed evenly to all the components. The

SEQUENTIAL scheme loads each component one at a time to capacity until the loop demand is met. The components are loaded up in the order that they appear in the equipment list specified in input. The UNIFORM scheme first divides the load evenly among all available components. If some components do not have the capacity to meet the uniformly distributed load, the remaining load is divided among the remaining components.

Summary of Plant Loop Demand Calculation Schemes

There are two plant loop demand calculations schemes in EnergyPlus. There is a **SingleSetPoint** and a **DualSetPointDeadband**; the **SingleSetPoint** is the default if that field is left blank in the PlantLoop object. In the SingleSetPoint scheme the Plant Loop requires that a Setpoint Manager set a single setpoint value that sets Node%TempSetPoint. Examples of this Setpoint Manager would be: the objects SetpointManager:Scheduled, SetpointManager:OutdoorAirReset, etc. For the DualSetPointDeadband scheme the Plant Loop requires that a Setpoint Manager that sets the high and low setpoint values for Node%TempSetPointHi and Node%TempSetPointLo. Examples of this setpoint manager would be: SetpointManager:Scheduled:DualSetpoint. Look in the Input Output Reference for the correct usage of these SetpointManagers.

The Plant Loop Demand Calculation Scheme determines the amount of heating or cooling necessary to bring the temperature of the Plant Loop to its setpoint(s). When this value is determined then the Load Distribution scheme explained in the previous section takes this value and distributes the load to the appropriate equipment. The demand calculation scheme determines how the load is calculated. In the next section is a summary of the 2 algorithms and how they are used.

Loop Demand Calculation Scheme SingleSetPoint

The SingleSetPoint scheme for the PlantLoop takes the value that is placed on the Node%TempSetPoint and calculates the heating or cooling load necessary to obtain that setpoint.

$$\begin{aligned} \text{DeltaTemp} &= \text{LoopSetPoint} - \text{LoopTempIn} \\ \text{LoopDemand} &= \dot{m} \cdot C_p \cdot \text{DeltaTemp} \end{aligned}$$

The sign of the Loop Demand determines if the loop has a cooling or heating load. Then the Load Distribution scheme distributes this calculated load to the appropriate equipment.

Loop Demand Calculation Scheme DualSetPointDeadband

The DualSetPointDeadband scheme for the PlantLoop takes the value that is placed on the Node%TempSetPointHi and Node%TempSetPointLo calculates the heating or cooling load necessary to obtain that setpoint; if in the DeadBand then no load is calculated. The pseudo code below shows the basis of the algorithm.


```

!Calculate the demand on the loop
IF (mdot > 0.0) THEN
  LoadToHeatingSetPoint = mdot*Cp*(LoopSetPointLo - LoopTempIn)
  LoadToCoolingSetPoint = mdot*Cp*(LoopSetPointHi - LoopTempIn)
  ! Possible combinations:
  ! 1 LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint > 0 --> Heating required
  ! 2 LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint < 0 --> Cooling Required
  ! 3 LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint > 0 --> Dead Band Operation
  ! 4 LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint < 0 --> Not Feasible
  IF (LoadToHeatingSetPoint .GT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
    LoopDemand = LoadToHeatingSetPoint
  ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .LT. 0.0) THEN
    LoopDemand = LoadToCoolingSetPoint
  ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
    LoopDemand = 0.0
  ELSE
    CALL ShowSevereError
  END IF
ELSE
  LoopDemand = 0.0
END IF

IF (ABS (LoopDemand) < LoopDemandTol) LoopDemand = 0.0

```

The sign of the Loop Demand determines if the loop has a cooling or heating load. Then the Load Distribution scheme distributes this calculated load to the appropriate equipment, if there is any.

Plant and Condenser Equipment Operation Schemes

Plants and condenser loops must have some mechanism for controlling the operation of the loop and which equipment is available under different operating conditions. Once the Loop load is calculated by the return conditions from the demand side and using the loop setpoint, this load needs to be allocated to the supply equipment according to the users input. This is mainly done by the operation schemes.

Each operation scheme must have the type of operation scheme, its identifying name, and the schedule that defines its availability. The first scheme appearing in the list is given the highest priority; the second scheme has second highest priority, etc. In other words, if according to its schedule, the first operation scheme is available, then it is used by the simulation to define how the plant or condenser loop operates. If it is not available, the second operation scheme in the list is checked to see if it is available until the highest priority scheme that is also available is found. See the Input Output Reference for input field details.

Plant Operation Schemes

See the Input Output Reference for input field details. The options for plant control schemes are:

Uncontrolled Loop Operation

The PlantEquipmentOperation:Uncontrolled scheme takes the full capacity of the supply equipment and cools or heats the loop accordingly. An example would be a cooling tower where the cooling tower would cool the condenser loop with all of its available capacity and not be limited by a capacity range or setpoint. Uncontrolled loop operation simply specifies a group of equipment that runs 'uncontrolled'. If the loop runs, this equipment will run also, unless turned off by the loop flow resolver to maintain continuity in the fluid loop.

Cooling Load Range Based Operation or Heating Load Range Based Operation

The PlantEquipmentOperation:CoolingLoad (or PlantEquipmentOperation:HeatingLoad) defines the different ranges and which equipment list is valid for each range. In each trio, there is a lower limit for the load range, an upper limit for the load range, and a name that links to an equipment availability list (PlantEquipmentList). Load range operation is used when the loop load is calculated and then the equipment is selected in the proper range.

This allows for the most efficient operation of the plant equipment or for the user to determine the most efficient plant configuration. When the equipment list has been determined then the load is allocated to the equipment in a manner selected by the user with “Optimal or Sequential” load distribution scheme. The load range based operation scheme has two statements associated with it: a main statement that defines the ranges that individual priority settings are valid and the lists of equipment that may be used for each range.

Condenser Operation Schemes

This is very similar to the plant operation schemes, but there are several more options available with the CondenserLoop. The condenser operation schemes apply to the equipment on the ‘supply side’ of the condenser loop—pumps, cooling towers, ground coupled heat exchangers, etc. The keywords select the algorithm that will be used to determine which equipment is available for each time step. The ‘*Range Based Operation*’ schemes select a user specified set of equipment for each user specified range of a particular simulation variable. ‘*Load Range Based*’ schemes compare the demand on the condenser supply side with specified load ranges and associated equipment lists. ‘*Outdoor...Range Based*’ schemes compare the current value of an environmental parameter with user specified ranges of that parameter. See the Input Output Reference for input field details.

Uncontrolled Loop Operation

The PlantEquipmentOperation:Uncontrolled scheme takes the full capacity of the supply equipment and cools or heats the loop accordingly. An example would be a cooling tower where the cooling tower would cool the condenser loop with all of its available capacity and not be limited by a capacity range or setpoint. Uncontrolled loop operation simply specifies a group of equipment that runs ‘uncontrolled’. If the loop runs, this equipment will run also, unless turned off by the loop flow resolver to maintain continuity in the fluid loop.

Cooling Load Range Based Operation or Heating Load Range Based Operation

The PlantEquipmentOperation:CoolingLoad (or PlantEquipmentOperation:HeatingLoad) statement defines the different ranges and which equipment list is valid for each range. In each trio, there is a lower limit for the load range, an upper limit for the load range, and a name that links to an equipment availability list (CondenserEquipmentList). Load range operation is used when the loop load is calculated and then the equipment is selected in the proper range. This allows for the most efficient operation of the plant equipment or for the user to determine the most efficient plant configuration. When the equipment list has been determined then the load is allocated to the equipment in a manner selected by the user with “Optimal or Sequential” load distribution scheme. The load range based operation scheme has two statements associated with it: a main statement that defines the ranges that individual priority settings are valid and the lists of equipment that may be used for each range.

Outdoor Drybulb Range Based Operation, Outdoor Wetbulb Range Based Operation, Outdoor RHPercent Range Based Operation

The various “PlantEquipmentOperation:Outdoor*” statements define the different ranges of the various environmental parameters and which equipment list is valid for each range. After the keyword and the identifying name, a series of data trios is expected. In each trio, there is a lower limit for the load range, an upper limit for the load range, and a name that links to an equipment availability list (the “CondenserEquipmentList”).

Outdoor Drybulb Temperature Difference Based Operation,. Outdoor Wetbulb Temperature Difference Based Operation

The various “PlantEquipmentOperation:Outdoor*Difference” statements control strategies help to control any condenser equipment based on the difference between a reference node temperature and any environmental temperature. For example a cooling tower can be

controlled by a strategy, which looks at the difference between the tower inlet temperature and wet-bulb temperature. A difference range is specified for each equipment list.

Primary-Secondary Loop Systems

Currently there are two methods to simulate a primary-secondary system in EnergyPlus:

- Plant Connection Component
- Common Pipe

Each of the method is described in some detail below.

Plant Connection Component

Plant Loop Connection Components (PlantLoopConnection and PlantLoopConnection:Controlled) are used to connect two plant loops together, typically to create a Primary-Secondary plant loop system. This system normally has multiple chillers on the primary constant volume loop, with a variable volume flow system to provide flow to multiple cooling coils. This keeps the flow through the chillers at a fixed level and allows for variable flow in the secondary systems. The hydraulic coupling is provided between the primary and secondary loop by the plant loop connection component. The PlantLoopConnection is a control volume around some piping and bypass valve. Its main function is to resolve mass flow between the two nested plant loops based upon the control modes entered by the user.

The PlantLoopConnection was designed to use the existing component/loop/solution structure to facilitate the simulation with the existing demand side manager and the supply side manager. The connection component does the mass flow resolution between the nested loops and does the proper mixing with the bypass as necessary. This should facilitate the majority of the primary secondary systems being used today, a diagram is shown below and a sample input file is included in the test suite.

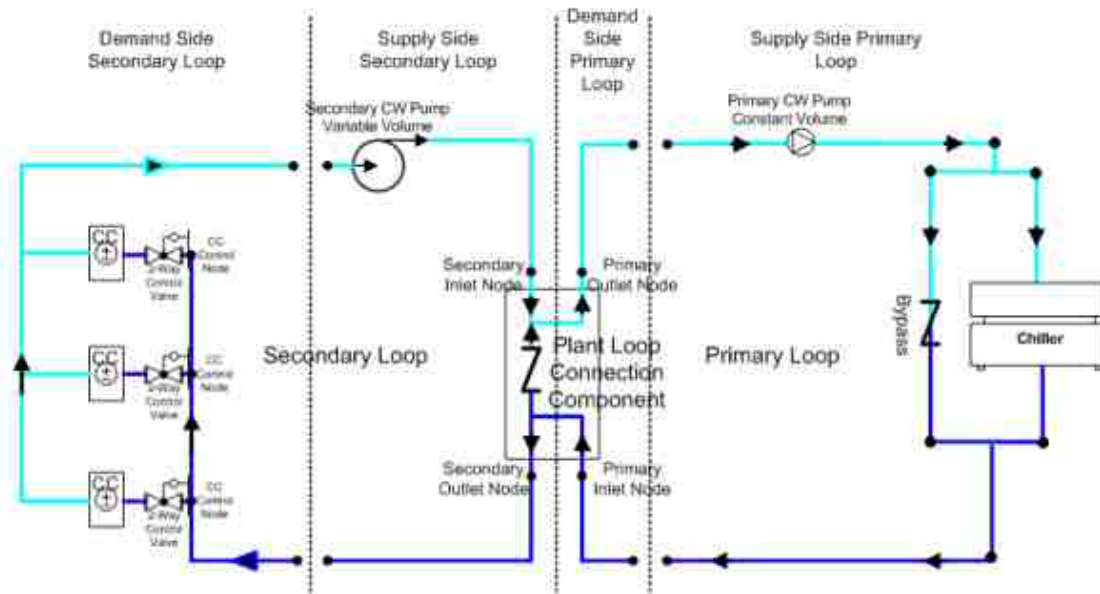


Figure 99. Example of a Primary-Secondary Nested Loop Simulation

The PlantLoopConnection is simulated on the Supply Side of the Secondary Loop and the results are provided to the Demand Side of the Primary Loop. Since the demand side of the primary loop is not simulated until the supply side of the secondary loop is simulated, there is an additional iteration through the demand side that is necessary. Only the

PlantLoopConnection triggers this additional demand-side iteration. This was necessary to keep the iterated information current and to ensure that an energy balance is achieved. The differences between the two types of connection component are described below.

PlantLoopConnection

This connection component is uncontrolled and is entirely passive. The following assumptions apply:

- The initial implementation has a constant flow primary loop (Loop with Chillers) and a variable flow secondary loop (Loop with the Coils).
- The secondary flow must be less than or equal to the primary flow.
- The secondary outlet flow is always taken completely from the primary inlet flow, so the secondary outlet temperature
- When the secondary side shuts down, either by no demand or the secondary pump scheduled off, the primary side will also shut down for that time step.
- If either the secondary or primary pump is scheduled, the same schedule needs to be applied to both of the pumps.
- Currently, only two loops can be simulated together, i.e. only one level of nesting.

PlantLoopConnection:Controlled

This connection component models an implied 3-way valve to control the leaving temperature on the secondary side. It will take the flows set by the respective loops and mix primary supply with secondary return to supply the specified secondary setpoint temperature if possible. The following assumptions apply:

- The secondary flow may be less than, equal to, or greater than the primary flow.
- The mass flow rate at the Primary Side Outlet Node is always equal to the mass flow rate at the Primary Side Inlet Node.
- The mass flow rate at the Secondary Side Outlet Node is always equal to the mass flow rate at the Secondary Side Inlet Node.
- If the primary or secondary side mass flow rate is less than VerySmallMassFlow (1.0D-30), the other side will simply bypass all flow from its inlet to its outlet.
- Currently, only two loops can be simulated together, i.e. only one level of nesting.
- The control algorithm is as follows:
 - 1) Determine the maximum possible primary to secondary mass flow rate. If the primary flow is greater than or equal to the secondary flow for the current time step, then this will be the mass flow rate at the Secondary Side Inlet Node. If the secondary flow is greater than the primary flow for the current time step, then this will be the mass flow rate at the Primary Side Inlet Node.
 - 2) Determine the primary to secondary mass flow rate required to meet the temperature setpoint on the Secondary Side Setpoint Node. The minimum primary to secondary flow rate is zero, meaning that the primary and secondary inlet flows are 100% bypassed to the primary and secondary outlet flows, and there is no exchange between the primary and secondary loops. The maximum possible primary flow to secondary is the result of Step 1.
 - 3) If the resulting secondary side temperature difference at the maximum possible primary to secondary mass flow rate is less than SmallTempDiff (1.0D-5) then the primary to secondary mass flow rate is set to zero and both sides bypass all flow.
 - 4) Calculate the primary and secondary outlet temperatures using the primary to secondary mass flow rate determined in Steps 2 and 3.

Common Pipe

Common pipe feature eliminates the need of specifying two different EnergyPlus loops each for Primary and Secondary half loops. Instead the user can set up the system as it is used in real life applications. A common pipe simulation requires that pumps be placed on both Demand (Secondary) and Supply (Primary) sides of the loop. A typical Common Pipe layout as used in EnergyPlus is shown in figure 92. The major assumptions in the common pipe implementation are as follows:

1. Pumps are placed on both demand and supply side of the loop.
2. Secondary pump flow rate can be less than, equal to or greater than the primary pump flow rate.
3. The flow at the inlet node of the half loop is equal to the flow at the outlet node of the half loop.
4. The pumps can have different schedules and any loop can be shut off when the other loop is still running.

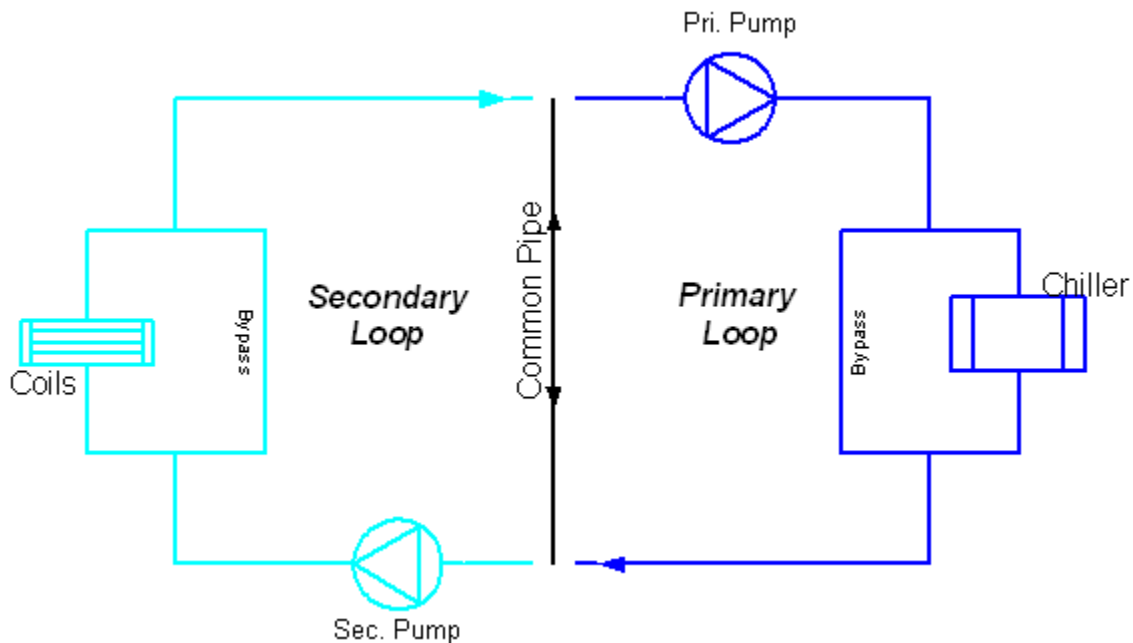


Figure 92: Common Pipe layout in EnergyPlus

Common pipe simulation is done during the interface update call at both Supply-to-Demand and Demand-to-Supply. Appropriate checks are used to make sure that the effect of flow reversal in between iteration is taken care of. Moreover, the common pipe keeps track of the flow rates and temperatures at all the four nodes linked to it; namely, the inlet and outlet nodes of each sub loop. This record will help to decide if loops have converged or not. In situations where the primary component meets the setpoint and the coil controls does not change its flow request, the common pipe converges quickly. The simple description of the control algorithm for common Pipe implementation is as follows:

- At FirstHVACiteration, the common pipe flow is initialized to zero.
- Common pipe is simulated at interfaces and thus we will have 2 different flows handle on either side of interface.
- Loops and corresponding flow rates are assigned inlet or outlet (to common pipe) depending on the interface which calls it. So when common pipe is called from demand to supply interface, the inlet loop is demand side and outlet loop is supply side and vice versa.

- Inlet flow is compared to outlet flow and the difference is set as the common pipe flow.
- At each interface the common pipe flow is assigned a direction which can be into the interface (Inlet flow < Outlet flow) or away from interface (Inlet flow > Outlet flow).
- Outlet temperature is calculated depending on the flow rate and flow direction. When flow is away from interface outlet flow temperature is same as inlet flow temperature. For a common pipe flow into the interface, the outlet flow temperature is calculated as mixed temperature of inlet flow and the common pipe flow.
- At demand to supply interface, the supply side inlet node temperature and flow rate are updated every iteration. At supply to demand interface, only flow is updated. The temperature is updated only at the end of timestep.
- Loops iterate till the flow and temperatures at all the 4 concerned nodes do not change.

Two-Way Common Pipe

In a typical EnergyPlus plant loop simulation, the only half loop inlet/outlet node that can be controlled is the Supply Side Outlet Node. In some cases this requirement becomes a limitation in analyzing different options. A good example is the Thermal storage application, where during charging phase, the coil setpoint can be different from the Ice storage equipment setpoint. This requirement is relaxed by using the PlantLoopConnection:Controlled object. However the modeling is done with some assumptions, which are not always useful. While connection component object can be used effectively for cases where secondary inlet is controlled, it will not work where the primary inlet has to be controlled (for example controlling the chiller return water temperature). A Two-Way common pipe tries to remove this limitation by providing a more generic interface to Primary-Secondary systems. Another benefit by using the Two-Way common pipe is the reduction in the user input in specifying the Primary-Secondary system. With the use of Two-Way common pipe, Primary and Secondary sides can be specified as a single Plant Loop. Though the Two-Way common pipe is designed to be generic some assumptions apply in modeling the component. The assumptions are as follows

- The secondary flow may be less than, equal to, or greater than the primary flow.
- The mass flow rate at the Primary Side Outlet Node is always equal to the mass flow rate at the Primary Side Inlet Node.
- The mass flow rate at the Secondary Side Outlet Node is always equal to the mass flow rate at the Secondary Side Inlet Node.
- Only one node (either Primary Inlet or Secondary Inlet) along with the Primary outlet node can be controlled. The system of equations that describe the loop interface will be under specified if both the Primary and Secondary Inlet nodes have to be controlled.

Figure 100 shows a schematic of a typical Primary-Secondary system with Two-Way common pipe. The flow in the common pipe legs (broken lines) can be in either direction. The flow depends on the following factors:

- Which node is controlled? If Primary inlet node is controlled then the flow in the common pipe leg on the supply side is from bottom to top. If Secondary inlet node is controlled then the flow in the common pipe leg on the demand side is from top to bottom.
- Once the flow in the one of the common pipe leg is fixed, then the direction of flow in the other leg depends on the difference between primary and secondary flows.
- Is the specified setpoint achievable with current secondary and primary outlet conditions? If the setpoint is not achievable, then the flow in each common pipe leg is reduced to its minimum possible value.

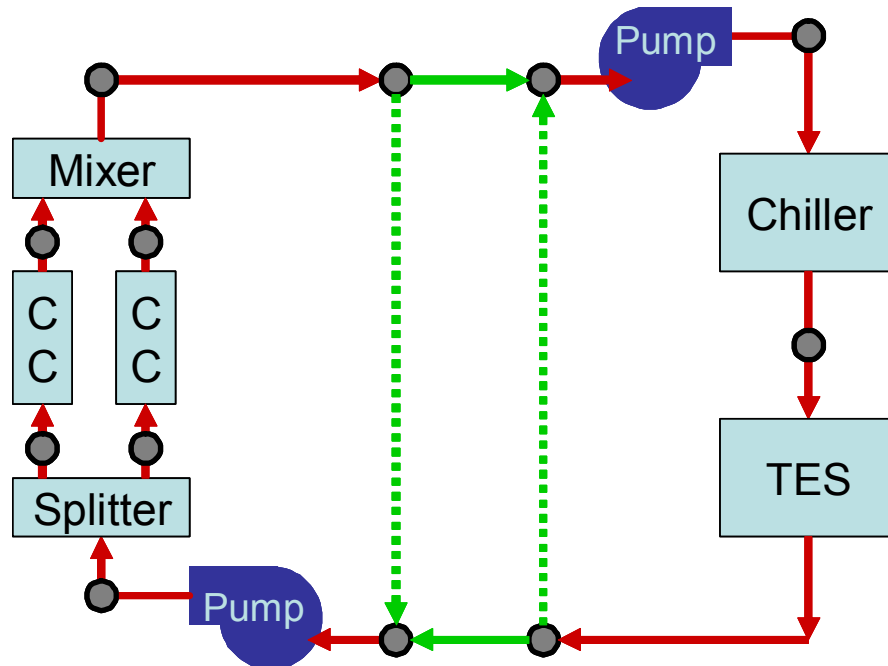


Figure 100. Schematic of a Two-Way Common Pipe used in Primary-Secondary System.

Figure 101 shows the Two-way common pipe interface in more detail. Mass flow rates and temperatures at different nodes are specified. Algorithm to solve for the flow rates in the two-way common pipe interface is briefly described below.

- Depending on the node that has to be controlled, the direction of flow in its corresponding common pipe leg is obtained.
- At the controlled node, with known demand outlet temperature, supply outlet temperature, primary flow rate and secondary flow rate calculate how much flow has to be recirculated in that particular half loop, so that setpoint is achieved.
- With a known flow in one common pipe leg, the flow on Primary to Secondary (or secondary to primary) is easily obtained.
- A simple mass balance on the remaining nodes will provide the flow and direction of flow in the remaining common pipe legs.

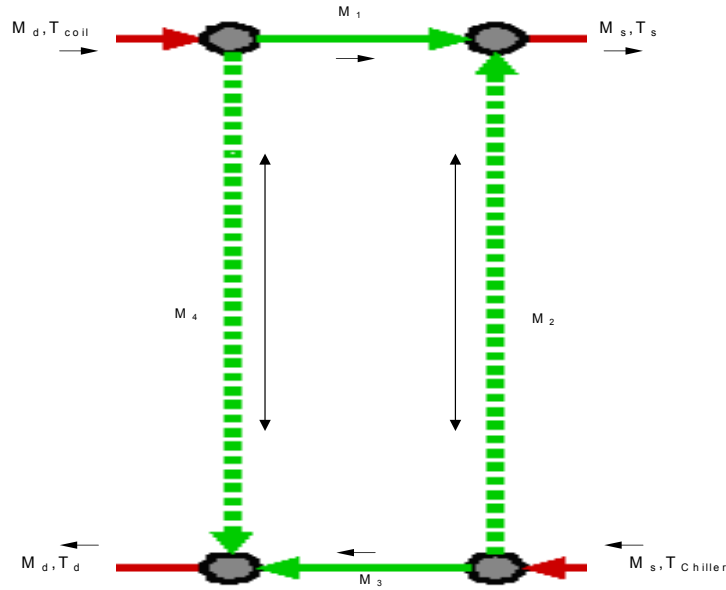


Figure 101. Two-Way Common Pipe Interface in Detail

Heat Recovery Loop Systems

Heat Recovery is accomplished by specifying another set of supply and demand loops. Each of the heat recovery components, i.e. engine driven and combustion turbine chillers, and internal combustion and combustion turbine generators is designed to use the existing component/loop/solution structure to facilitate the simulation with the existing demand side manager and the supply side manager. Heat recovery normally contains components that produce heat that can be recovered, and the ability to store or use that heat elsewhere in the system. The component that can store the excess heat and allow it to be used elsewhere in the system or for domestic hot water is the Water Heater:Simple and is defined in the Input/Output Reference.

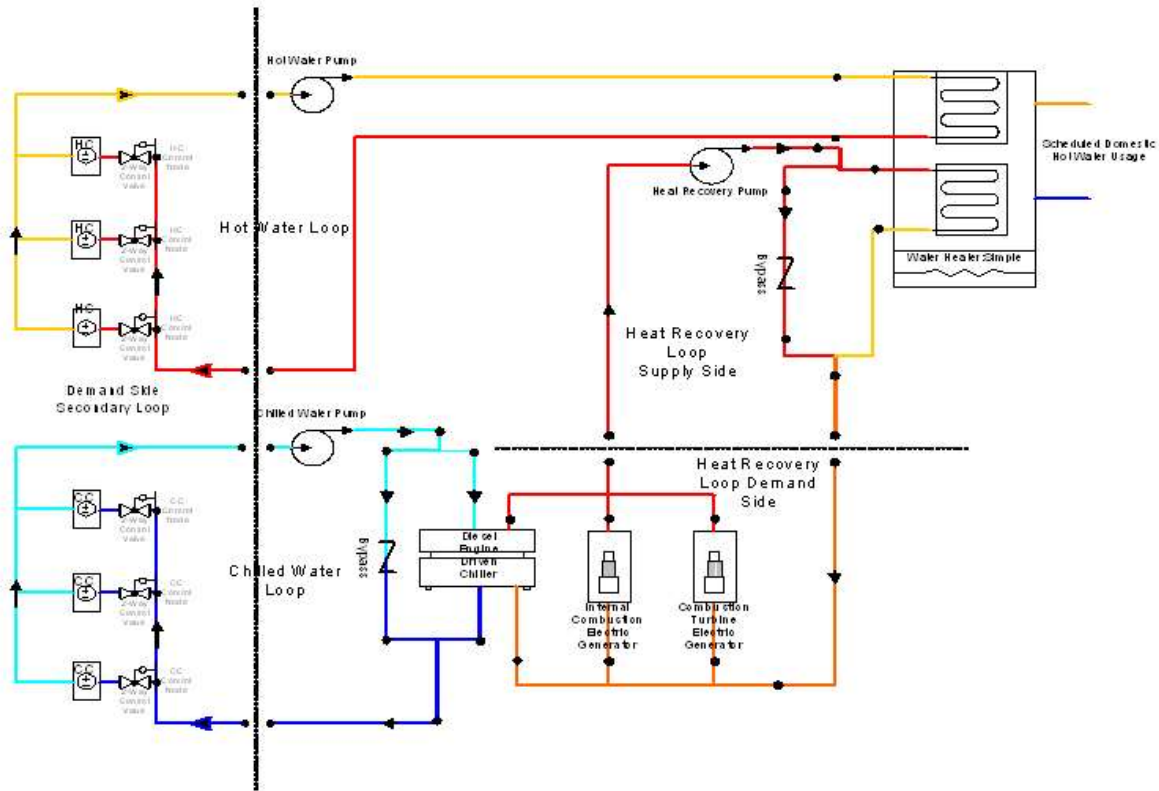


Figure 102. Example of a Heat Recovery Loop Simulation

In the example above there is a chilled water Loop with chilled water supplied by a diesel engine driven chiller. There is a hot water Loop that is being supplied by the water heater: simple. There is also scheduled domestic hot water usage on the water heater which excess demand can be met by a number of user-specified heating sources. Then on the demand side of the heat recovery loop there is the engine driven chiller, internal combustion, and combustion turbine electric generators with specified mass flows to recover the heat. This hot water is pump on the supply side by the heat recovery pump and provides the heat to the water heater to meet the water heater setpoint. This is probably one of the more complex configurations and interactions that would take place in heat recovery, but using the Plant supply and demand side configurations this can be extended to meet most user configurations. The plant water heater can also be used to just meet scheduled domestic hot water use, provide a hot water source for PlantLoop equipment, or provide a hot water storage tank for heat recovery as a single function. Or any combination of the above can be configured. Example files of some of these configurations are provided in the internal test suite.

Steam Systems and Component Models

A steam system uses the vapor phase of water to supply enthalpy or kinetic energy through the piping network. In case of EnergyPlus, the steam system is designed to provide energy solely for the building heating requirements. Hot steam from the boiler or steam generator in buildings can be used to heat a conditioned space with suitable heat transfer equipment such as fan-coils units, unit-heaters, radiators and convectors or steam can also heat water through shell and tube heat exchangers, and hot water can be supplied to the terminal units to provide the zone heating requirements.

The advantages that steam system offer over hot water or other heating systems are:

1. Steam flows through the system unaided by external energy source such as pumps; pressure difference moves steam across the system.
2. Steam, because of its low-density, can be used in high-rise buildings where water systems create excessive pressure.
3. Terminal units such as heating coils can be added or removed without making any changes to the system.
4. Steam components can be repaired or replaced by closing the steam supply without the difficulties associated with draining and refilling like in the water systems.
5. Steam is pressure-temperature dependent, therefore the system temperature can be controlled by varying either steam pressure or temperature.
6. Steam can be distributed through out the system without any change in temperature.

In view of the advantages mentioned, the steam systems are suitable for applications where heat is required for process and comfort heating such as in industrial plants, hospitals, restaurants, dry cleaning plants laundries and commercial buildings. They are also suitable in places where the heating medium has to travel great distances such as in facilities with scattered building locations or where the building height would result in excessive pressure in a water system, or locations where the load changes occur intermittently. Thus steam system is an essential and necessary development step for EnergyPlus.

From EnergyPlus simulation point of view, the advantage associated with a steam system is that steam can be distributed through out the system without change in temperature. This means that the boiler outlet temperature can simply be set equal to the heating coil inlet temperature for a steam system.

Another simulation-based advantage associated with the steam system is Steam Quality, which does not change from boiler outlet to coil inlet. Actual building steam systems are equipped with condensate drains through out the system, these drains remove, almost immediately any condensate that is formed during steam transportation, thereby maintaining the steam quality at constant value of 1.0 through out the high-pressure steam side.

The HVAC steam system implementation includes simulation models for two phase steam equipment like steam boiler, steam to air heating coils, steam pipes and condensate pumps, which can be connected to the generic loop framework.

Steam Loop Assumptions

To replicate the working of an actual building steam system in a satisfactory manner with simulation, it was necessary to make certain assumptions. These assumptions help simplify the loop complexity and increase usability. The effects of the assumptions made are described in detail below.

The following figure describes the Temperature Entropy Ts diagram based on which the steam system operates in EnergyPlus. The steam side of the loop operates on constant saturation pressure of steam: P_{Steam} , the waterside of the loop operates at atmospheric pressure P_{atm} .

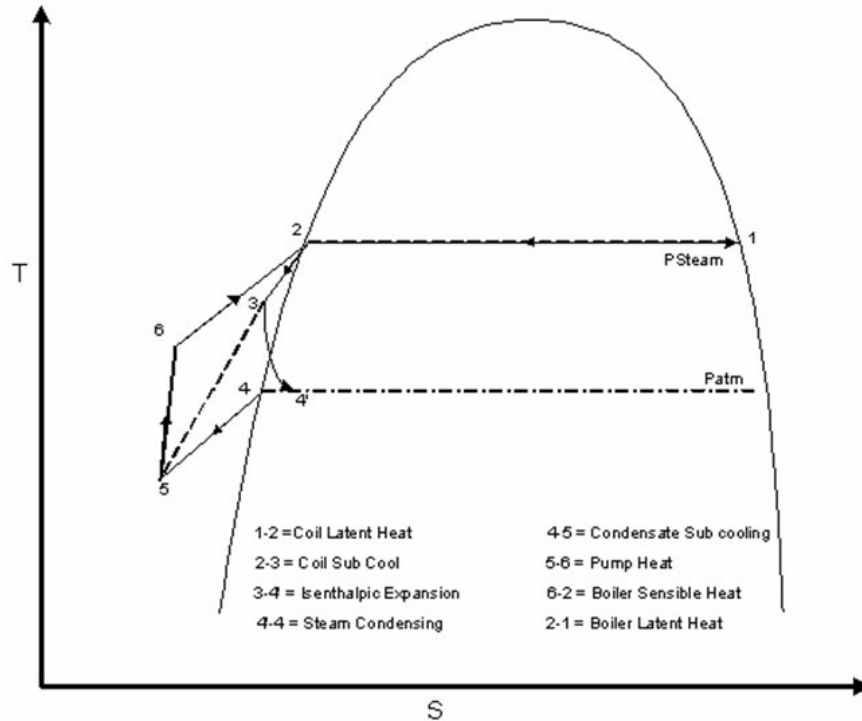


Figure 103. Schematic of Temperature – Entropy Diagram for Steam loop

It should be noted that the figure is simply a schematic and not a scaled representation of the process on a Mollier Chart. For the following descriptions, please refer to the schematic figure above.

- Process 1–2 on the Ts diagram, represents condensation of steam in the coil at constant pressure; this is where the steam gives up latent heat to the zone.
- Process 2–3 represents the subcooling of condensed steam at higher pressure, this subcooling takes place inside the steam coil, just before the steam trap. The delta temperature represented by 2-3 is the degree of subcooling in the steam coil, and is a user input to the steam coil. This subcool generally accounts for 1 to 2 % of the total heat transfer in the steam coil.
- Process 3–4' represents the isenthalpic expansion of water from high-pressure steam side to atmospheric pressure across the steam trap. As steam gives up its latent heat at the steam coil the condensate that forms in the steam coil still exists at higher pressure. This condensate is discharged to a lower pressure across the steam trap, this condensate contains more heat than necessary to maintain the liquid phase at the lower pressure, this excess heat causes some of the condensate to vaporize or flash to steam at lower pressure at some quality. The amount of water that flashes to steam can be calculated by the following equation

$$\%Flash\ Steam = \frac{h_{4'} - h_4}{h_{fg}} \times 100$$

Where $h_{4'}$ is Enthalpy of liquid at steam pressure just before condensate is supposed to leave the coil. Enthalpy at Point 3 is equal to enthalpy at point 4', since it is an isenthalpic process, and h_{fg} is the latent enthalpy of the fluid at atmospheric pressure.

For example, water at 102°C and 120 Kpa flashes to steam at 100°C and atmospheric pressure, with quality equal to 0.003. This results in loss of some latent capacity of steam and is one of the terms contributing to loop loss in steam system.

- Process 4'-4 represents the condensation of the flashed steam, which has exited from steam trap into the condensate drain. Condensation occurs at atmospheric pressure P_{atm} , there is loss in latent capacity due to this unavoidable process, only condensate can be returned back to the boiler in a steam system.
- Process 4-5 represents the loop sub cooling at atmospheric pressure; this is the sub cooling of the condensate that takes place during condensate return to the boiler because the return loop is not insulated, loop sub cooling is of the order of 20°C to 30°C. This is a user-defined input in every steam coil, because the variability in location of steam coils in a building will result in different condensate return temperatures for each of the coils.
- Process 5-6 represents the temperature and pressure rise in condensate due to pump heat addition. The pumping process generates heat, which is added to the condensate. The condensate is pumped back to the boiler at higher pressure.
- Process 6-2 represents the sensible heat addition by the boiler to the return condensate.
- Process 2-1 represents the latent enthalpy of steam, added by the boiler to the water to convert it to steam at saturation pressure.
- Point 3, which is outlet of the coil and Point 5, which is inlet of the pump are specified directly by the user, subsequently the loop losses in EnergyPlus are directly summed up as the enthalpy difference between point 3 and 5, which is calculated by fluid property routines in EnergyPlus. This helps to maintain flexibility and at the same time helps negate the intermediate points calculation in the system.

Aspects of the steam loop such as quality of steam, steam pressure, and steam generation which play an important role in EnergyPlus simulation are described in following sections.

Constant Pressure Steam loop

The steam loop in EnergyPlus is pressure driven and it is assumed to operate at constant pressure on the gaseous-steam part, while the condensate return loop is assumed to operate at atmospheric pressure. The steam loop essentially operates at saturation pressure corresponding to the steam temperature; the steam boiler serves to maintain the loop temperature. The boiler model determines the amount of energy required to generate the required amount of steam.

Factors such as friction in pipes, which tend to cause small amount of pressure drop in steam loop are neglected in the model. It is assumed that the steam pipes are fairly well insulated to prevent heat loss and subsequent condensation of steam in the pipes. In actual systems small quantities of steam, which condenses due to heat loss during transportation is removed immediately from the system by steam drains. This helps eliminate water hammer, degradation of steam quality and heat transfer capability.

Steam Generation at Saturated Conditions:

Building steam heating systems avoid supplying superheated steam because superheat damages the building HVAC equipment. Superheated steam is generated only if there is a cogeneration power system in building such as steam turbine, which requires superheated steam. The purpose of having superheated steam is redundant for building steam heating systems because the amount of energy carried by the same is negligible compared to the latent heat. A simple enthalpy calculation has been provided in this section to describe the negligible effect of superheat.

Assuming the Mass flow rate of steam as unity.

Case 1: Loop Operating at Saturation Conditions (no superheat), calculating enthalpy of saturated steam at boiler outlet temperature.

Boiler Outlet Temperature = 105°C,

$$h_{Case1:NoSuperheat} = [h_{f105^{\circ}C}]$$

$$h_{Case1:NoSuperheat} = 2684000 \quad J$$

Case 2: Loop operating with Superheated steam, calculating enthalpy of superheated steam for 5°C superheat.

Degree of Superheat = 5°C

Boiler Outlet Temperature = 110°C,

$$h_{Case2:Superheat} = [h_{f110^{\circ}C}]$$

$$h_{Case2:Superheat} = 2691000 \quad J$$

The energy difference between the superheated state and the saturated state as calculated in the following equation. A 5°C superheat provides only a 0.2608% increase in heat transfer. The advantage of this additional increase in heat transfer is negligible, especially when considering the economic aspect of sizing a bigger heat exchanger to accommodate additional superheat transfer. The detrimental effects of superheated steam on the building HVAC system also come into effect once superheat is used.

$$\Delta Q_{2-1} = \frac{2691000 - 2684000}{2684000} \times 100 = 0.2608 \text{ percent}$$

Based on the reasoning above the steam loop in EnergyPlus is designed and implemented to operate at saturated conditions.

Steam Quality

The boiler operation is assumed capable to generate steam at quality equal to 1.0 every time. This is a reasonable assumption, since in practice the variability in generated steam quality would only occur if the boiler operation were not properly controlled.

The steam loop is assumed to have perfect transport mechanism. There is no transportation losses due to friction and heat transfer with surroundings. This assumption helps maintain the quality of steam through out the system constant value of either 0 or 1.

Steam enters the coils at boiler outlet conditions. Steam coils are designed with steam traps, which only allow condensed steam to leave the coil; hence the steam always condenses and leaves the coil at quality of 0.0.

With the above simplifying assumption enables the EnergyPlus steam loop to be solved without problems.

Steam Traps

Steam traps are essential part of the steam system; they are indirect flow controllers of the loop. Purpose of steam trap is to allow only condensate out of the coil from higher-pressure steam to lower atmospheric pressure condensate return line. Points 3-4, in schematic Figure 103, represent this process across the steam trap on the Ts diagram. The expansion process across the steam trap is assumed to be isenthalpic. There is possibility of flashing of high-pressure condensate across the trap because of pressure drop, resulting in generation of steam at lower pressure, this steam generated at lower pressure subsequently condenses

in the return piping, and heat is lost to the atmosphere. This heat lost is a part of the steam loop losses.

Steam traps are modeled in the EnergyPlus steam coil by assigning the condensate outlet from the coil a quality of 0.0 and sub cooling the condensate at lower pressure.

Although condensate from the steam coil contains valuable heat, attempting to utilize this heat by holding the condensate in the coil reduces the heat transfer area. It causes operational problems because it retains air, which further reduces heat transfer and non-condensable gases such as carbon dioxide, which cause slow corrosion of the steam coil. Steam moves rapidly in mains and supply piping so when condensate accumulates to the point where the steam can push a slug of it, serious damage can occur from the resulting water hammer, hence the condensate should be immediately removed from the steam coil. This is achieved with steam traps.

Hence an ideal steam trap should remove all condensate, air, and non-condensable gases that might be in the system, with little or no loss of steam.

Loop Losses

Subcooling of condensate in condensate return piping and flash steam condensing across the steam trap constitute the unavoidable loop losses in the EnergyPlus simulation steam system. These losses can be inferred from Figure 103 and are summed up by calculating enthalpy difference between points 3 and 5 on the schematic Ts diagram.

Unavoidable losses in the EnergyPlus steam loop occur due to pressure drop across the steam trap, which causes flashing of steam and loss in some percentage of latent heat of steam, process 3-4' and 4'-4 on the Ts diagram in Figure 103. The condensate is pumped backed to the boiler through return pipe network, which is not insulated. Sub cooling of the condensate occurs in the return network, represented by process 4-5 on the Ts diagram in Figure 103. This loop sub cool contributes to significant percentage of loop losses.

In practical systems the return pipeline to the boiler is not insulated despite the condensate containing some valuable heat, however due to low mass flow rate of steam, this amount is negligible and only recovered if separate heat recovery systems are used by coupling them to the loop. The condensate is occasionally collected in a receiver and then pumped back to the boiler. EnergyPlus simulation mimics practical systems by assumed that the return pipeline is not insulated and accounts for this by calculating loop losses.

The loop losses are calculated in the steam coil simulation model rather than the steam pipe simulation, because the condensate sub cool in the return loop is a direct function of the location of the steam coil in the building. In building energy software like EnergyPlus the user would have a fair idea about location of steam coils rather than the location of condensate return piping.

Steam To Air Heat Exchanger

Description of Model:

The steam to air heat exchanger (Coil:Heating:Steam input object) is the terminal equipment in the steam loop on the demand side that satisfies the heating requirements of the various zones. The steam-to-air heat exchanger simulation model in EnergyPlus calculates the mass flow rate of steam desired to meet the heating demand.

A heating coil can be used either as a zone coil or a coil in the air loop depending on the user and application. The steam coil simulation model is designed to take these two locations into consideration. An air-loop steam heating coil is temperature controlled and the zone coil is zone load controlled. This relatively simple and straightforward concept of coil control is preferred over the iterative method to determine mass flow rates using various numeric techniques. The assumptions made in the coil model are described in the section below.

This model accounts for the latent heat transfer and sensible cooling of water; the question of superheat is eliminated because steam is assumed to be saturation conditions. Steam enters the coil at quality equal to 1.0, at saturation temperature and leaves the coil with

desired degree of sub cooling. The user inputs the desired degree of subcooling, which determines the condensate outlet condition from the coil.

EnergyPlus heat balance methods determine the amount of heating required in the zone to maintain the zone at the desired conditions. Based on this value of heating load, the zone coil determines the mass flow rate of steam required since the heating coil load is the control variable in a zone coil. The following equation describes this calculation to determine steam mass flow rate required for the desired heating capacity.

$$\dot{m}_{zc} = \frac{Q_{zc}}{h_{fg} + c_{p,w} \times \Delta T_{sc}}$$

In case of the air loop-heating coil, the load on the coil is calculated within the coil simulation routine. The air loop coil is setpoint controlled and heats the air to maintain the air stream at the desired setpoint, the setpoint is a user input, generally in the range of 12°C to 16°C. The following equation describes the air loop coil load.

$$Q_{al} = \dot{m}_a \times c_{p,a} \times [T_{sp} - T_a]$$

The following equation is used to determine the steam mass flow rate required by the air loop coil to meet the heating requirements.

$$\dot{m}_{al} = \frac{Q_{al}}{h_{fg} + c_{p,w} \times \Delta T_{sc}}$$

Each of the zone coils and air loop coils are simulated independently and the steam mass flow rates for each is added over every time step of simulation. This value of total mass flow rate is reported to the boiler, which in turns supplies this required amount of steam.

The control of the steam to air coil is a complex issue. The loop splitter-splits total steam flow from the boiler and delivers the required amount of steam to each of the coils connected to the loop through the steam pipe network. In cases where the system is undersized, the coils demand more mass flow rate of steam than the boiler can generate. The splitter in this case cannot provide all the coils with requested steam mass flow. Subsequently the coils are starved of steam and the zone temperatures fall. In some cases the user might schedule off the coil, they should then not operate. These issues need to be taken care of in the implementation of the steam coil simulation model. The control algorithm for the steam coil operation under various situations is best explained with the help of pseudo-code using standard IF THEN ELSE blocks.

*****PSEUDO CODE SECTION STARTS*****

Steam coil is zone load controlled.

IF (Coil = LoadControlled) THEN

Check for operational conditions only then continue simulation further. The operational conditions are the inlet mass flow rates of steam and air to the coil, the user schedule to the coil and heating load on the coil. The coil is simulated only if these conditions are met.

IF [($\dot{m}_s > 0$) . and. ($\dot{m}_a > 0$) .and. (Schedule = ON) .and. ($Q_{zc} > 0.0$)] THEN

If the heating demand from the zone-heating coil is greater than coil capacity, then the heating coil is undersized, and the coil can only deliver its maximum heating capacity to the zone. In this case the heating demand on the coil is set equal to this lower value of maximum heating capacity. If the above is not true then the simulation ignores this statement and proceeds to the next one.

$$IF \left[(Q_{ZC} > Q_{ZC_{Max}}) \right] \quad THEN \quad \left[Q_{ZC} = Q_{ZC_{Max}} \right]$$

The following equation calculates the steam mass flow rate required by the coil. This flow rate is required to meet the heating requirements for the zone. This value of mass flow is requested from the splitter outlet.

$$\dot{m}_{zc} = \frac{Q_{zc}}{h_{fg} + c_{p,w} \times \Delta T_{sc}}$$

If the calculated value of steam mass flow with the previous equation is greater than the maximum inlet steam flow that the splitter can provide to the coil at that time step. Then the requested coil flow rate is set equal to the inlet steam flow rate. This is the maximum amount of steam that can be supplied to the coil at this moment. The coil can provide heating capacity equal to this limited amount of steam. If the requested flow rate is less than what the splitter can provide then the program ignores the logic of the IF Loop Below

$$IF (\dot{m}_{zc} > \dot{m}_{in}) \quad THEN \quad (\dot{m}_{zc} = \dot{m}_{in})$$

Re-Calculating the coil heating capacity with the lower value of steam mass flow rate.

$$Q_{ZC} = \dot{m}_{zc} \times (h_{fg} + c_{p,w} \times \Delta T_{sc})$$

END IF

The following equations calculate the outlet condensate-water and outlet air temperatures to the zone based on the amount of heating capacity provided by the coil.

$$T_{W_{out}} = T_{S_{in}} - \Delta T_{sc}$$

$$T_{a_{out}} = T_{a_{in}} + \frac{Q_{zc}}{\dot{m}_a \times c_{p,a}}$$

ELSE Else the coil is not running and in this case set outlets to inlets.

$$\dot{m}_s = 0, \quad \dot{m}_a = 0, \quad Q_{zc} = 0, \quad T_{W_{out}} = T_{S_{in}}, \quad T_{a_{out}} = T_{a_{in}}$$

END IF

END IF End IF for the zone load controlled coil.

Steam coil is temperature controlled.

IF (Coil=TemperatureControlled) THEN

Check for operational conditions and continue simulation further. The operational conditions are the inlet mass flow rates of steam and air to the coil, the user schedule to the coil and delta temp exists between the setpoint and air inlet temperature. The coil is simulated only if these conditions are met.

IF $\left[(\dot{m}_s > 0) .and. (\dot{m}_a > 0) .and. (Schedule = ON) .and. \right]$ THEN

Calculate the heating load on the coil using setpoint and inlet air temperatures.

$$Q_{al} = \dot{m}_a \times Cp_a \times [T_{sp} - T_a]$$

The logic loop for temperature-controlled coil begins here. In case the heating load on the coil is negative, which might occur if the setpoint is below the air inlet temperature, the coil operation needs to be shut off.

IF $(Q_{al} < 0.0)$ THEN

Assigning the inlet to outlet and mass flows to zero shuts off the coil operation.

$$\dot{m}_s = 0, \quad \dot{m}_a = 0, \quad Q_{al} = 0, \quad Tw_{out} = Ts_{in}, \quad Ta_{out} = Ta_{in}$$

If air loop coil load is greater than maximum coil load calculated at maximum steam mass flow rate, in such case the coil is undersized, the coil can only deliver to the air loop its maximum heating capacity. Setting the air loop coil load equal to maximum load on coil. If this is not the case then the program ignores this ELSE IF block and proceeds to the next one.

ELSE IF $\left[(Q_{al} > Q_{al_{Max}}) \right]$ THEN $\left[Q_{al} = Q_{al_{Max}} \right]$

If the heating coil is under sized then it can only provide its maximum heating capacity, in this case the air temperature will be below the setpoint, and is calculated based on this maximum allowed value of heat transfer. Calculating the air and water outlet temperatures.

$$Ta_{out} = Ta_{in} + \frac{Q_{al}}{\dot{m}_a \times c_{p,a}}$$

$$Tw_{out} = Ts_{in} - \Delta T_{sc}$$

Determining the mass flow rate of steam required by the undersized coil. This value of mass flow is requested from the splitter outlet.

$$\dot{m}_{al} = \frac{Q_{al}}{h_{fg} + c_{p,w} \times \Delta T_{sc}}$$

A check is introduced to determine if this requested mass flow rate is greater than what the splitter outlet can provide to the coil at that particular time step of simulation. In this case the requested value of steam mass flow is greater than what the splitter can provide to that coil, subsequently set the requested coil flow rate equal to the inlet steam flow rate, delivered to the coil by the splitter. This is the maximum amount of steam that can be supplied to the coil at this moment. If the requested flow rate is less than what the splitter can provide then the program ignores the logic of the IF Loop.

IF $(\dot{m}_{al} > \dot{m}_{in})$ THEN $(\dot{m}_{al} = \dot{m}_{in})$

Re-Calculating the coil heating capacity and air outlet temperature with the lower value of steam mass flow rate provided by the splitter.

$$Q_{al} = \dot{m}_{al} (h_{fg} + c_{p,w} \times \Delta T_{sc})$$

$$Ta_{out} = Ta_{in} + \frac{Q_{al}}{\dot{m}_a \times c_{p,a}}$$

End IF

If the above two IF ELSE block are not true, then the coil is perfectly sized, the splitter can provide the required mass flow rate to the coil, and the setpoint temperature can be maintained as desired.

ELSE

The ideal case where the coils can meet the required setpoint temperature. Setting the outlet air temperature to the setpoint, calculating the water outlet temperature and the required steam mass flow rate.

$$T_{a_{out}} = T_{SP}$$

$$T_{W_{out}} = T_{S_{in}} - \Delta T_{sc}$$

$$Q_{al} = \dot{m}_a \times c_{p,a} \times [T_{sp} - T_a]$$

END IF End IF statement, for the air coil heating loop.

END IF End IF statement for the operating condition loop

END IF End IF statement for the Temperature Setpoint Controlled Coil

The steam coil model encapsulates the above described control logic along with the other necessary simulation code for reading the user inputs and the code for reporting the simulation results.

*****PSEUDO CODE SECTION ENDS*****

The two main types of coil control discussed above are followed by common simulation code in the coil model. This code calculates the loop losses occurring due to flashing of steam across the steam trap, isenthalpic expansion occurring across the steam trap due to pressure difference, and loss occurring due to condensate sub cooling returning back to the boiler. The above-mentioned two processes are explained in Figure 103 as process 3-4' and 4-5.

The loop loss calculation is included in the steam coil simulation model, because the degree of subcooling in the return piping for the condensate is solely a function of the coil location. In practical applications a coil, which is further away from boiler would return back condensate at much lower temperatures compared to coil, which is closer to boiler. Hence for user ease it makes perfect sense to include this input into the coil and calculate the pump inlet conditions in the steam coil simulation model itself.

The loop losses in the EnergyPlus steam system is calculated by determining the enthalpy difference between point 3 and 5. The simulation code that determines the loop loss is common to both the coil models, this helps determine the condensate pump inlet conditions.

The following equation is used to calculate condensate enthalpy at coil outlet, point 3 in Figure 103. Point 2 represents condensed steam; enthalpy at this point is calculated directly by EnergyPlus property routines.

$$h_{f3} = h_{f2} - c_{p,w} \times \Delta T_{SC}$$

Point 4 is at atmospheric pressure, enthalpy at this is calculated directly by EnergyPlus property routines. It is saturation enthalpy of steam at quality equal to 0.0 and saturation temperature at atmospheric pressure.

Point 5 is inlet to the pump; enthalpy at this point is calculated with the following equation. The delta temperature represents the degree of loop subcooling occurring during condensate return back to the pump.

$$h_{f5} = h_{f4} - c_{p,w} \times \Delta T_{LSC}$$

Subsequently loop loss for each coil would be enthalpy difference between 3 and 5 and is calculated using the following equation

$$\Delta Q_{loss} = \dot{m}_s \times (h_{f3} - h_{f5})$$

The total loop loss would be a summation of the individual losses occurring for each of the steam coils, this is the unavoidable loss in current steam system.

A simple schematic describing the coil framework, inlet and outlet conditions to the coil and the flow rate resolution is shown in Figure 104. Five zone coils and one air loop coil are described in the picture, Qzone is calculated by EnergyPlus heat balance and it is the input to the zone coils, while in air loop coil the Qal is calculated within the simulation model

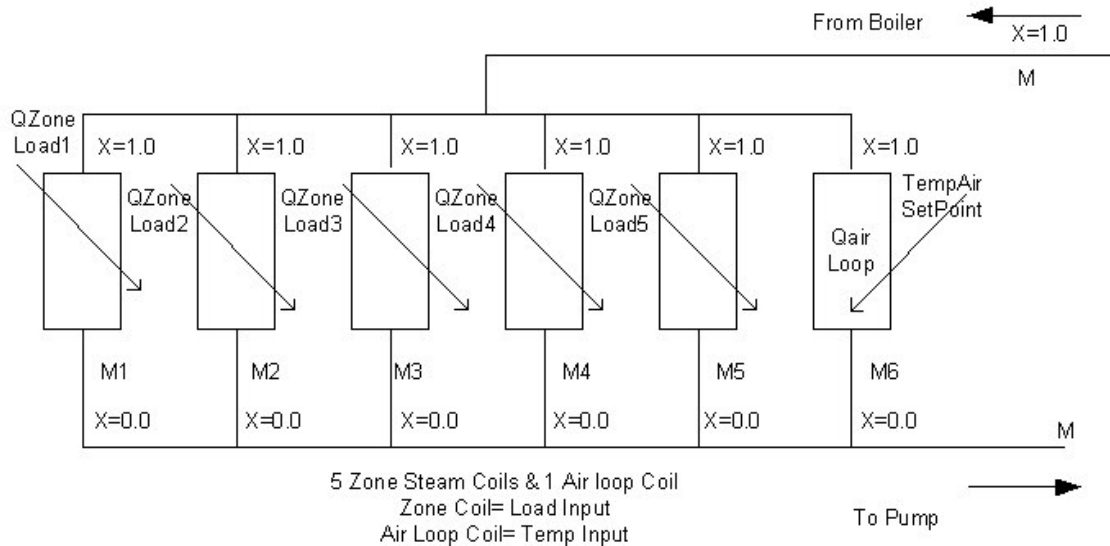


Figure 104. Schematic of Steam Coil Connection to the Steam Loop

As depicted in the figure above the steam condenses on entering the coils, sub cools by the specified amount and leaves the heat exchanger as water. The steam in the heat exchanger at any moment has to condense eventually since steam trap at the outlet to coil permits only water to leave the coil. Steam traps being an essential part of the loop are modeled by controlling the coil outlet condition at quality equal to 0.0. Subsequently the amount of heat transferred to air is a direct function of latent heat and the degree of sub cooling desired by the user.

Model Assumptions

The steam coils works on two basic assumptions, firstly, it's assumed that perfect latent heat transfer takes place over every time step, secondly, the user specified degree of subcooling occurs in the coil. Steam coils always use steam traps- purpose of which is to only let water out of the coil, hence the modeler knows with certainty the outlet dryness fraction of steam coil is at quality equal to 0.0, because water leaves coil. Hence the heat transfer is equal to latent heat of steam, which is independent of UA value of coil when averaged over time step of EnergyPlus simulation.

In practice there is 1% to 2 % sub cooling of the outlet water stream, the user with desired degree of subcool input accounts for this, subsequently UA calculation for the coil model become superfluous and is neglected in the EnergyPlus model.

Ideally for sub-cooling UA would play an important role, however sub cooling in steam coil constitutes negligible amount of heat transfer, hence this model avoids the UA calculation. This is a very reasonable assumption since sensible heat transfer is barely 1% to 2 % of the total heat transfer in steam coils, simply due to large latent heat capacity of coil. NOTE in his thesis he gives a little example here which is probably not necessary for the Docs.

Condensate Pump

The steam loop operates at a pressure differential over the gaseous and liquid part of the loop; subsequently a condensate pump is required to pump the condensed steam back to the boiler at the required pressure. Two main reasons for condensed steam to be returned to the boiler is energy savings in reheating water since the water is at high temperature, secondly the water is treated by chemicals to prevent corrosion in the pipes and equipment. This is an expensive process and therefore it's economical to reuse the chemically treated water.

Description of Pump Model

The pump model designed in EnergyPlus is a variable speed condensate pump. Condensation of steam produces water, this takes place at variable rate hence the return water flow rate would be variable but constant when averaged over a time step. Condensate pumps operate intermittently; the pump will run at its capacity if a load/flow rate is sensed and will shut off if there is no load on the loop.

The condensate pump essentially operates between maximum and minimum flow rates, which are the physical limits of the device. The pump is designed to meet the flow request made by demand side components, which are the coils in case of steam system.

The main difference between the variable volume pump and the constant volume pump is the Part Load Performance Curve. The fraction of full load power is determined by the third order equation, which follows:

$$P_{Frac} = C_1 + C_2 \times PLR + C_3 \times PLR^2 + C_4 \times PLR^3$$

In preceding equation, PLR stands for part load ratio while C1 to C4 are pump- part load coefficients. The following five equations describe the pump operation and calculation of pump output variables such as total power, shaft power and pump heat to fluid etc.

$$\dot{V} = \frac{\dot{m}}{\rho_w}$$

Using the previous equation the pump volume flow rate is determined; the user enters the value of the maximum and minimum volume flow rate.

The pump part load ratio is a function of the pump volume flow rate at any instance determined by the loop and the pump nominal volume flow rate, which is a user input. The following equation calculates the Part Load Ratio (PLR).

$$PLR = \frac{\dot{V}}{\dot{V}_{nom}}$$

The pump power is calculated as described in following equation. Pump power is a product of fractional full load power and pump nominal power use. Fractional full load power is calculated in a preceding equation while pump nominal power is a user input to the model.

$$P = P_{Frac} \times P_{Nom}$$

The shaft power is simply the product of the pump power and motor efficiency, this is required to calculate the heat generated and delivered to the fluid being pumped. The following equation is used to calculate pump shaft power.

$$P_S = P \times \eta_m$$

The model assumes that all heat generated and lost ends up in the fluid to the loop, this assumption is necessary since EnergyPlus operates on a closed loop. The following equation is used to calculate the pump heat to the fluid, which raises the condensate temperature. The pump motor efficiency is defined by the user input and the fractional motor loss to fluid is the amount of heat generated by the pump motor that is added to the fluid loop (as opposed to being lost to the environment where the pump is located). FracMotorLossToFluid is also a user input

$$P_H = P_S + (P - P_S) \times F_{mf}$$

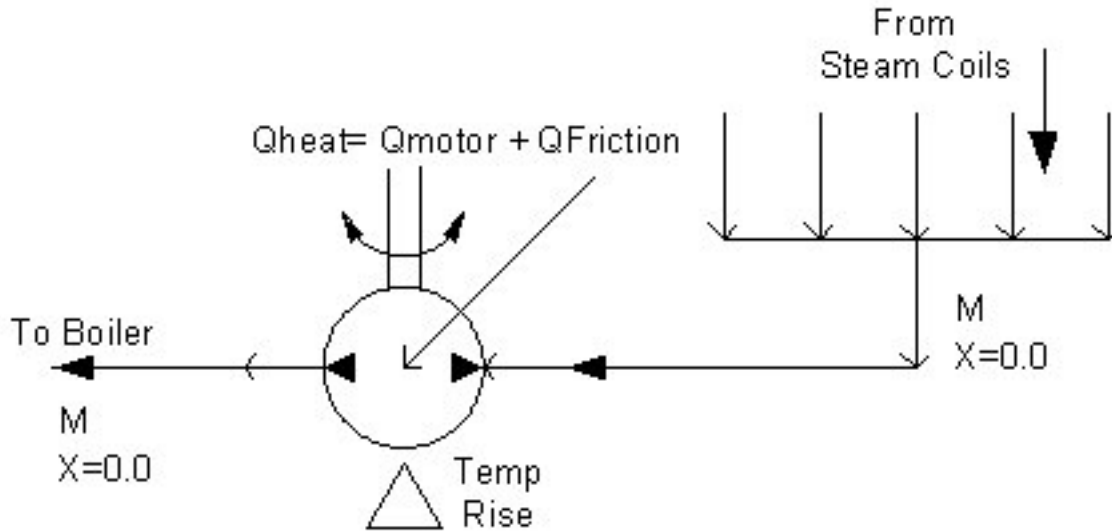
The shaft power relates to the increase in head through the pump to the loop operating pressure. The head lost through the piping network due to frictional heat, represents the heat gain by the fluid throughout the network. For model simplicity, this heat is added along with the heat resulting from the pump motor. The difference between the pump power and the shaft power is the inefficiency of the pump, or the amount of energy input into the pump that the motor converts to heat rather than mechanical energy. Some of this heat is added to the fluid being pumped. These two above-mentioned terms are used in the PumpHeatToFluid equation for calculating P_H shown above.

A simple energy balance over the pump based on the pump inlet conditions and flow rate is used to calculate the pump outlet temperature. The condensate outlet temperature from the pump is slightly higher than inlet due to the heat dissipation to the fluid steam during pumping action. This is calculated in the following equation. The pump water outlet temperature is the boiler inlet temperature.

$$T_{w_{out}} = T_{w_{in}} + \frac{P_H}{\dot{m} \times c_{p,w}}$$

Pump control is an important part of the steam loop. Existing control structure from EnergyPlus has been utilized to operate the condensate pump. The pump is simulated first on the supply side of the loop after the demand side loop has determined what the demand on the loop will be.

A simple schematic describing the flow across the pump is shown in the following figure



Condensate Pump Operation

Figure 105. Schematic of Condensate Pump in Steam Loop

Model Assumptions

Due to the fact that a pump is a mechanical device that acts on the fluid it is circulating, it causes the fluid temperature rise. The EnergyPlus model assumes that all pressure increase caused by the pump will eventually be lost due to friction, and that friction will be added as heat to the fluid. Although the plant and condenser loops in steam systems are simple pressure-based models, a simplifying assumption has been made in EnergyPlus to assume the heat resulting from the pump itself and from friction throughout the loop is added at the fluid being pumped. In case of steam, this assumption is easily justified because the volume flow rate of water is very small in the loop.

Steam Pipe

Description of Model

The steam pipe essentially serves as “energy carrier” and transfers the node conditions from one point of the pipe to another. It’s simply a node inlet to node outlet connection, transferring values from inlet to outlet. The pipe forms an important part of the framework connecting various equipments from the supply to demand side and inlet and outlets of the equipments.

The steam pipe supports two additional properties, which are pressure and quality, unlike its water counterpart. Pipe simulation model in EnergyPlus is hardwired to water as a fluid type; this necessitated the development of similar model supporting pressure and quality for the steam system.

Model Assumptions

The piping network in the developed steam system is assumed to perfectly distribute steam, return the condensate, and remove air and non-condensable gases. It is also assumed that the pipes are sized to distribute steam not only at full load but also at partial loads and excess loads that can occur during system warm up.

The steam pipe is perfect and there is no losses occurring in transportation. This assumption was necessitated since if pipe losses were accounted for, the loss would have to be

distributed into the zone, which would be a very complex issue in itself, since in EnergyPlus the pipes are unaware of their locations and simply serve as connectors.

Loop and Equipment Sizing

The importance of correct equipment sizing is often ignored in discussions of building simulation methods. The lack of reliable, efficient and flexible sizing calculations can present a serious barrier to the adoption and acceptance of building simulation programs. This section describes the sizing methodology implemented in EnergyPlus. This method includes:

1. A zone by zone heat balance load and air-flow calculation for multiple design days;
2. Significant user control with modest input requirements;
3. Zone, system and plant level calculations of design heating and cooling capacities and fluid flow rates;
4. Modular, component-specific sizing algorithms for each HVAC component.

Sizing Manager

The sizing calculations in EnergyPlus are managed by a sizing manager contained in the software module *SizingManager*. The main sizing manager routine *ManageSizing* is called from *ManageSimulation* before the annual simulation sequence is invoked. *ManageSizing* performs the following tasks.

- (1) By calling *GetSizingParams*, *GetZoneSizingInput*, *GetSystemSizingInput* and *GetPlantSizingInput* reads in all the user sizing input contained in objects *Sizing:Parameters*, *Sizing:Zone*, *Sizing:System* and *Sizing:Plant*. These objects and their data are described in the [EnergyPlus Input Output Reference](#), Group – Design Objects.
- (2) Set the *ZoneSizingCalc* flag equal to *true*.
- (3) Loop over all the design days. **This starts the zone design calculations.**
 - (a) Call *UpdateZoneSizing(BeginDay)* to initialize zone design load and flow rate sequences.
 - (b) Loop over hours in the day
 - (i) Loop over zone time steps in each hour
 1. Call *ManageWeather* to obtain outside conditions for this time-step.
 2. Call *ManageHeatBalance* to do a full heat balance calculation for each zone. The call to *ManageHeatBalance* also brings about an HVAC simulation. *ZoneSizingCalc = true* signals the *HVACManager* to ignore the real HVAC system and instead run the ideal zonal system (described below) used to calculate design loads and flow rates. *HVACManager* also calls *UpdateZoneSizing(DuringDay)* to save the results of the ideal zonal system calculation in the design load and flow rate sequences.
 - (c) Call *UpdateZoneSizing(EndDay)* to calculate peaks and moving averages from the zone design sequences for each design day.
 - (4) Call *UpdateZoneSizing(EndZoneSizingCalc)* to calculate for each zone the peak heating & cooling loads and flow rates over all the design days. The corresponding design load and flow rate sequences are saved for use in the system design calculations. **This ends the zone design calculations.**
 - (5) Set the *SysSizingCalc* flag equal to *true*.
 - (6) Call *ManageZoneEquipment* and *ManageAirLoops* to read in the zone and central system inputs needed for the system design calculations. The program needs enough information to be able to figure out the overall air loop connectivity.
 - (7) Loop over all the design days. **This starts the system design calculations.**
 - (a) Call *UpdateSysSizing(BeginDay)* to initialize system design load and flow rate sequences.

- (b) Loop over hours in the day
 - (i) Loop over zone time steps in each hour
 1. Call *ManageWeather* to obtain outside conditions for this time-step.
 2. Call *UpdateSysSizing(DuringDay)* to save the results of the system design calculations in the system design load and flow rate sequences.
 - (c) Call *UpdateSysSizing(EndDay)* to calculate peaks and moving averages from the system design sequences for each design day.
- (8) Call *UpdateSysSizing(EndSysSizingCalc)* to calculate for each system the peak heating & cooling loads and flow rates over all the design days. The corresponding design load and flow rate sequences are saved for use in the system design calculations. **This ends the system design calculations.** And this ends the tasks of the Sizing Manager.

Zone Design Loads and Air Flow Rates

Overview

There is no single best way to establish design HVAC flow rates and size HVAC equipment. Different building designs, climates, and HVAC systems will impose varying constraints on the designer. The method used to size an HVAC system in a hot, moist climate such as Miami will be different than the method used for a building in Albuquerque. The type of building is also relevant – a simple watts per square foot loads estimate could be adequate for a building containing a network server farm while a detailed, dynamic loads simulation would be necessary for a passive solar building. In the end the designer's experience and engineering judgement will play an important role in any sizing calculation.

HVAC equipment sizing begins with the calculation of space heating and cooling loads. A space cooling (heating) load is defined as the rate at which heat must be removed (added) to a space to maintain a constant temperature. The current industry standard method for calculating space loads is the *heat balance method* [ASHRAE Fundamentals (2001), page 29.1; Pedersen et al., (1997); Pedersen (2001)]. Since EnergyPlus is a heat balance based simulation program it is straightforward for the program to use this method for calculating zone loads.

Zone Design Data Arrays

The zone design data arrays are:

ZoneSizingInput(i) stores the input data from the Sizing:Zone objects.

CalcZoneSizing(i,j) stores the results of the zone design calculations for all zones and all design days. The index *i* is for the controlled zones, *j* for design days.

CalcFinalZoneSizing(i) stores the results of the zone design calculations for the peak heating and cooling cases for each zone. The index *i* is for the controlled zones.

ZoneSizing(i,j) corresponds to *CalcZoneSizing* but includes the effect of the user specified sizing factor or user specified zone design flow rate.

FinalZoneSizing(i) corresponds to *CalcFinalZoneSizing* but includes the effect of the user specified sizing factor or user specified zone design flow rate.

The data stored in *CalcZoneSizing*, *CalcFinalZoneSizing*, *ZoneSizing* and *FinalZoneSizing* includes the following data items.

Table 34. Zone Sizing Data

Name	Description
All the data from <i>ZoneSizingInput</i>	

<i>DesHeatMassFlow</i>	the zone design heating air mass flow rate in [kg/s]
<i>DesCoolMassFlow</i>	the zone design cooling air mass flow rate in [kg/s]
<i>DesHeatLoad</i>	the zone design heating load in [W]
<i>DesCoolLoad</i>	the zone design cooling load in [W]
<i>DesHeatDens</i>	the zone design heating air density [kg/m ³]
<i>DesCoolDens</i>	the zone design cooling air density [kg/m ³]
<i>DesHeatVolFlow</i>	the zone design heating air volume flow rate [m ³ /s]
<i>DesCoolVolFlow</i>	the zone design cooling air volume flow rate [m ³ /s]
<i>DesHeatCoilInTemp</i>	zone heating coil design air inlet temperature [C]
<i>DesCoolCoilInTemp</i>	zone cooling coil design air inlet temperature [C]
<i>DesHeatCoilInHumRat</i>	the zone heating coil design air inlet humidity ratio [kg/kg]
<i>DesCoolCoilInHumRat</i>	the zone cooling coil design air inlet humidity ratio [kg/kg]
<i>HeatMassFlow</i>	current zone heating air mass flow rate at the HVAC time step [kg/s]
<i>CoolMassFlow</i>	current zone cooling air mass flow rate at the HVAC time step [kg/s]
<i>HeatLoad</i>	Current zone heating load [W]
<i>CoolLoad</i>	Current zone cooling load [W]
<i>HeatZoneTemp</i>	Current zone temperature during heating [C]
<i>HeatZoneRetTemp</i>	current zone return temperature during heating [C]
<i>CoolZoneTemp</i>	Current zone temperature during cooling [C]
<i>CoolZoneRetTemp</i>	current zone return temperature during cooling [C]
<i>HeatZoneHumRat</i>	Current zone humidity ratio during heating [C]
<i>CoolZoneHumRat</i>	Current zone humidity ratio during cooling [C]
<i>ZoneTempAtHeatPeak</i>	zone temperature at maximum heating [C]
<i>ZoneRetTempAtHeatPeak</i>	zone return temperature at maximum heating [C]
<i>ZoneTempAtCoolPeak</i>	zone temperature at maximum cooling [C]
<i>ZoneRetTempAtCoolPeak</i>	zone return temperature at maximum cooling [C]
<i>ZoneHumRatAtHeatPeak</i>	zone humidity ratio at maximum heating [kg/kg]
<i>ZoneHumRatAtCoolPeak</i>	zone humidity ratio at maximum cooling [kg/kg]
<i>TimeStepNumAtHeatMax</i>	zone time step number (in the day) at the heating peak
<i>TimeStepNumAtCoolMax</i>	zone time step number (in the day) at the cooling peak

<i>HeatDDNum</i>	design day index of design day causing heating peak
<i>CoolDDNum</i>	design day index of design day causing cooling peak
<i>MinOA</i>	design minimum outside air [m3/s]
<i>HeatFlowSeq(i)</i>	daily sequence of zone heating air mass flow rates (zone time step) [kg/s]
<i>CoolFlowSeq(i)</i>	daily sequence of zone cooling air mass flow rates (zone time step) [kg/s]
<i>HeatLoadSeq(i)</i>	daily sequence of zone heating loads (zone time step) [W]
<i>CoolLoadSeq(i)</i>	daily sequence of zone cooling loads (zone time step) [W]
<i>HeatZoneTempSeq(i)</i>	daily sequence of zone temperatures (heating, zone time step) [C]
<i>HeatZoneRetTempSeq(i)</i>	daily sequence of zone return temperatures (heating, zone time step) [C]
<i>CooltZoneTempSeq(i)</i>	daily sequence of zone temperatures (cooling, zone time step) [C]
<i>CoolZoneRetTempSeq(i)</i>	daily sequence of zone return temperatures (cooling, zone time step) [C]
<i>HeatZoneHumRatSeq(i)</i>	daily sequence of zone humidity ratios (heating, zone time step) [kg/kg]
<i>CoolZoneHumRatSeq(i)</i>	daily sequence of zone humidity ratios (cooling, zone time step) [kg/kg]

Zone Design Load Calculation

As described in the preceding section, the Sizing Manager initiates the zone design calculation by looping over all of the design days and calling the Heat Balance Manager for each zone time-step in each design day. The Heat Balance manager then causes the HVAC Manager to be called in a manner identical to a normal simulation. The *ZoneSizingCalc* set to *true* signals the HVAC Manager to ignore the actual HVAC system and instead calculate the design zone loads and air flow rates using an ideal zonal system.

In module *HVACManager*, subroutine *ManageHVAC* calls *SimHVAC*. *SimHVAC* checks *ZoneSizingCalc*. If it is *true*, *SimHVAC* calls *ManageZoneEquipment* and returns, rather than simulating the actual system. In turn *ManageZoneEquipment* checks if *ZoneSizingCalc* is *true*; if it is it calls *SizeZoneEquipment* rather than *SimZoneEquipment*.

SizeZoneEquipment assumes that each controlled zone is served by an ideal air conditioning unit. This unit supplies heating or cooling air at a fixed, user input temperature and humidity (specified in the Sizing:Zone objects). The units have infinite capacity – the flow rate can be any amount. The calculation steps are as follows.

- 1) Loop over all the controlled zones.
- 2) If the system is active (zone temperature not in the deadband and zone load greater than 1 watt) the sign of the zone load is used to determine whether heating or cooling is required and T_{in} and W_{in} are set to the appropriate values from the Sizing:Zone input. The system output Q_{sys} is simply set equal to the zone demand – it is assumed that the ideal system can always meet the zone load. The air flow rate corresponding to the load is just

$$\dot{m}_{sys} = Q_{sys} / (C_{p,air} \cdot (T_{in} - T_z))$$

If the system is not active, the mass flow rate is set to zero and the system output is left at zero.

- 3) The results for each zone are stored in the zone sizing data arrays.

Updating and Adjusting the Zone Results

The results from *SizeZoneEquipment* are at the system time-step and are for all design days. These results then need to be summed or averaged over the zone time-step, peak values calculated for each design day, a heating & a cooling load sequence chosen for each zone from all the design day results, possible further smoothing of results done, zone coil loads calculated, and user sizing multipliers or user specified design flows taken into account. These tasks are accomplished by the subroutine *UpdateZoneSizing*. It is called at the start of each design day (*CallIndicator* = *BeginDay*), at the zone time-step (*CallIndicator* = *DuringDay*), at the end of the design day (*CallIndicator* = *EndDay*) and at the end of the zone design calculation (*CallIndicator* = *EndZoneSizingCalc*).

BeginDay

The environment (in this case, a design day) name and number are stored in the zone sizing data structures

DuringDay

The calculated and stored sequences are summed or averaged over the zone time-step.

EndDay

- (1) Smooth the design sequences by applying a moving, fixed-width averaging window to the sequences. The width of the window is user specified in the *Sizing:Parameters* input object. The sequences that are smoothed are:
 - (a) *CoolFlowSeq*
 - (b) *CoolLoadSeq*
 - (c) *HeatFlowSeq*
 - (d) *HeatLoadSeq*
 - (e) *CoolZoneRetTempSeq*
 - (f) *HeatZoneRetTempSeq*
- (2) The peak heating and cooling loads and mass & volume flow rates are extracted from each set of design sequences.
- (3) Using the time of the peak and the design outside air fraction the design zone heating and cooling coil inlet temperatures and humidity ratios are calculated.
- (4) For each zone, looking at the results for all of the design days, the design days that cause the peak heating and peak cooling for that zone are chosen and the corresponding design sequences and peak loads and flow rates are saved in the *CalcFinalZoneSizing* array. This finishes the calculated – unmodified by the user – portion of the zone design calculation.

EndZoneSizingCalc

- (1) Write out onto a comma-separated file the calculated design sequences for each zone: *HeatLoadSeq*, *CoolLoadSeq*, *HeatFlowSeq*, *CoolFlowSeq* and the corresponding peaks and volumetric flow peaks.
- (2) The data in *CalcZoneSizing* and *CalcFinalZoneSizing* is moved to *ZoneSizing* and *FinalZoneSizing*. The user modifications to the calculated sizing will be applied to and stored in *ZoneSizing* and *FinalZoneSizing*.

- (3) The user can modify the calculated zone design results by specifying a sizing factor at the global or zone level or by specifying an actual design heating or cooling zone design volumetric flow rate. All of this input is treated as a sizing factor. If the user inputs a cooling design volumetric flow rate for a zone it is divided by the calculated cooling design volumetric flow rate for the zone to give a zone cooling sizing factor. Note that the user can input a zone sizing factor or a zone design flow rate – not both – so there is never a conflict.
- (4) Once the zone heating and cooling sizing factors are established, the design flow and load sequences as well as peak loads and flows are multiplied by the appropriate sizing factor and stored in *ZoneSizing* and *FinalZoneSizing*. This is the data that will be used for sizing zone HVAC equipment and in the system sizing calculation.
- (5) The outside air fractions are recalculated using the new user-modified design flow rates and new design zone coil inlet conditions calculated and stored. At this point the condition that the design flow rates are never allowed to be less than the minimum outside air flow rate is imposed.

If *outside air method* is *flow/zone*, the input *outside air flow per zone* value will be used, even if it is zero or blank. If *outside air method* is *sum*, the sum of the *outside air flow per person* * *DesignNumberOfPeople* + *outside air flow per area* * *ZoneArea* will be used. If *outside air method* is *maximum*, the maximum of the *outside air flow per person* * *DesignNumberOfPeople* and *outside air flow per area* * *ZoneArea* will be used. If *outside air method* is *flow/person*, *outside air flow per person* will be used to calculate the design minimum outside airflow rate.

If *cooling design air flow method* is *flow/zone*, then *cooling design air flow rate* will be used for the design max cooling air flow rate. If *cooling design air flow method* is *design day*, then the design day calculation will set the design max cooling air flow rate. If *cooling design air flow method* is *design day with limit*, then the maximum from *cooling min flow per area* and *cooling min flow* will set a lower limit on the design max cooling air flow rate. In all cases, the maximum from *cooling min flow per area*, *cooling min flow*, and *cooling min flow fraction* will set a minimum zone cooling air flow rate. In all cases the maximum design cooling air flow rate must be \geq to the ventilation requirement.

If *heating design air flow method* is *flow/zone*, then *heating design air flow rate* will be used for the design max heating air flow rate. If *heating design air flow method* is *design day*, then the design day calculation will set the design max heating air flow rate. If *heating design air flow method* is *design day with limit*, then the maximum from *heating max flow per area*, *heating max flow* and *heating max flow fraction* will set an upper limit on the design max heating air flow rate. The design max heating air flow rate must always be \geq the ventilation requirement.

This concludes the calculation of the zone design flow rates and loads.

System Design Loads and Air Flow Rates

Overview

The purpose of the system design calculation is to estimate design heating and cooling loads and air flow rates for each air loop in the simulation problem. The calculation sequence for system level design loads and air flow rates resembles the calculation sequence for zone loads and air flow rates. There is an update subroutine *UpdateSysSizing* called at the beginning, during, and end of a loop in the Sizing Manager over all the design days. The major difference is that this calculation is done at the zone time-step only. There is no idealized component calculation triggered at the system time-step as in the zone calculation. The system design calculation operates at the zone time step using the design environment weather data and the data stored in the zone sizing arrays. The results of the system design calculation are stored in the system sizing arrays described below.

System Design Data Arrays

The system design data arrays are:

SysSizInput(i) stores the input data from the Sizing:System objects.

SysSizing(i,j) stores the results of the system design calculations for all systems and all design days. The index *i* is for air loops, *j* for design days.

CalcSysSizing(i) stores the results of the system design calculations for the peak heating and cooling cases for each air loop. The index *i* is for the air loops.

FinalSysSizing(i) corresponds to *CalcSysSizing* but includes the effect of the user specified sizing factor or user specified system design flow rate.

The data stored in *SysSizing*, *CalcSysSizing* and *FinalSysSizing* includes the following data items.

Table 35. System Sizing Data

Name	Description
All the data from <i>SysSizInput</i>	
<i>CoinCoolMassFlow</i>	coincident peak cooling mass flow rate [kg/s]
<i>CoinHeatMassFlow</i>	coincident peak heating mass flow rate [kg/s]
<i>NonCoinCoolMassFlow</i>	noncoincident peak cooling mass flow rate [kg/s]
<i>NonCoinHeatMassFlow</i>	noncoincident peak heating mass flow rate [kg/s]
<i>DesMainVolFlow</i>	design main supply duct volume flow [m ³ /s]
<i>DesHeatVolFlow</i>	design heat supply duct volume flow [m ³ /s]
<i>DesCoolVolFlow</i>	design cool supply duct volume flow [m ³ /s]
<i>SensCoolCap</i>	design sensible cooling capacity [W]
<i>HeatCap</i>	design heating capacity [W]
<i>PreheatCap</i>	design preheat capacity [W]
<i>CoolMixTemp</i>	design mixed air temperature for cooling [C]
<i>CoolMixHumRat</i>	design mixed air humidity ratio for cooling [kg water/kg dry air]
<i>CoolRetTemp</i>	design return air temperature for cooling [C]
<i>CoolRetHumRat</i>	design return air humidity ratio for cooling [kg water/kg dry air]
<i>CoolOutTemp</i>	design outside air temperature for cooling [C]
<i>CoolOutHumRat</i>	design outside air humidity ratio for cooling [kg water/kg dry air]
<i>HeatMixTemp</i>	design mixed air temperature for heating [C]
<i>HeatMixHumRat</i>	design mixed air humidity ratio for heating [kg water/kg dry air]
<i>HeatRetTemp</i>	design return air temperature for heating [C]
<i>HeatRetHumRat</i>	design return air humidity ratio for heating [kg water/kg dry air]
<i>HeatOutTemp</i>	design outside air temperature for heating [C]
<i>HeatOutHumRat</i>	design outside air humidity ratio for heating [kg water/kg dry air]

<i>HeatFlowSeq(i)</i>	daily sequence of system heating air mass flow rate (zone time step) [kg/s]
<i>CoolFlowSeq(i)</i>	daily sequence of system cooling air mass flow rate (zone time step) [kg/s]
<i>SensCoolCapSeq(l)</i>	daily sequence of system sensible cooling capacity (zone time step) [W]
<i>HeatCapSeq(i)</i>	daily sequence of system heating capacity (zone time step) [W]
<i>PreHeatCapSeq(i)</i>	daily sequence of system preheat capacity (zone time step) [W]
<i>SysCoolRetTempSeq(i)</i>	daily sequence of system cooling return temperatures (zone time step) [C]
<i>SysCoolRetHumRatSeq(l)</i>	daily sequence of system cooling return humidity ratios (zone time step) [kg water/kg dry air]
<i>SysHeatRetTempSeq(i)</i>	daily sequence of system heating return temperatures (zone time step) [C]
<i>SysHeatRetHumRatSeq(l)</i>	daily sequence of system heating return humidity ratios (zone time step) [kg water/kg dry air]
<i>SysCoolOutTempSeq</i>	daily sequence of system cooling outside temperatures (zone time step) [C]
<i>SysCoolOutHumRatSeq</i>	daily sequence of system cooling outside humidity ratios (zone time step) [kg water/kg dry air]
<i>SysHeatOutTempSeq</i>	daily sequence of system heating outside temperatures (zone time step) [C]
<i>SysHeatOutHumRatSeq</i>	daily sequence of system heating outside humidity ratios (zone time step) [kg water/kg dry air]

System Design Flow Rate and Load Summation and Adjustment

There is no system level subroutine corresponding to *SizeZoneEquipment*. Instead the system design loads and flow rates are calculated using the zone level results. The zone design flow rates for the zones served by an air loop are summed to obtain the system level design flow rates. These air flows are mixed with the system level design minimum outside air flow rate to obtain system design coil loads. These activities are all performed within the *UpdateSysSizing* subroutine in the *SimAirServingZones* module. It is called at the start of each design day (*CallIndicator = BeginDay*), at the zone time-step (*CallIndicator = DuringDay*), at the end of the design day (*CallIndicator = EndDay*) and at the end of the zone design calculation (*CallIndicator = EndSysSizingCalc*).

There is a logical flag *SysSizingCalc* corresponding to *ZoneSizingCalc*. It is used to allow the component routines to distinguish a normal simulation call from a being called during a system sizing calculation.

BeginDay

- (1) The environment (in this case, a design day) name is stored in the system sizing data structures.
- (2) Loop over the zones cooled by this air loop:

$$(a) \text{ NonCoinCoolMassFlow}_{\text{sys}} = \sum \text{DesCoolMassFlow}_{\text{zone}}$$

(3) Loop over the zones heated by this air loop:

$$(a) \text{ NonCoinHeatMassFlow}_{\text{sys}} = \sum \text{DesHeatMassFlow}_{\text{zone}}$$

DuringDay

(1) Loop over the zones cooled by this air loop:

$$\text{CoolFlowSeq}_{\text{sys}}(i) = \sum \text{CoolFlowSeq}_{\text{zone}}(i)$$

$$\text{SysCoolRetTemp}(i) = \frac{\sum (\text{CoolZoneRetTempSeq}(i) \cdot \text{CoolFlowSeq}_{\text{zone}}(i))}{\text{CoolFlowSeq}_{\text{sys}}(i)}$$

$$\text{SysCoolRetHumRat}(i) = \frac{\sum (\text{CoolZoneHumRatSeq}(i) \cdot \text{CoolFlowSeq}_{\text{zone}}(i))}{\text{CoolFlowSeq}_{\text{sys}}(i)}$$

$$\text{FracOA} = \rho_{\text{air}} \cdot \text{DesOutAirVolFlow}_{\text{sys}} / \text{CoolFlowSeq}_{\text{sys}}(i)$$

$$T_{\text{mix}} = T_{\text{outside}} \cdot \text{FracOA} + \text{SysCoolRetTemp}(i) (1 - \text{FracOA})$$

$$W_{\text{mix}} = W_{\text{outside}} \cdot \text{FracOA} + \text{SysCoolRetHumRat}(i) (1 - \text{FracOA})$$

$$\text{SysCoolOutTempSeq}(i) = T_{\text{outside}}$$

$$\text{SysCoolOutHumRatSeq}(i) = W_{\text{outside}}$$

Get the current (zone time-step) system cooling capacity:

$$\text{SysSensCoolCap}_{\text{cur}} = C_{p,\text{air}} \cdot \text{CoolFlowSeq}_{\text{sys}}(i) \cdot (T_{\text{mix}} - T_{\text{sup}})$$

$$\text{SensCoolCapSeq}(i) = \text{SysSensCoolCap}_{\text{cur}}$$

If $\text{SysSensCoolCap}_{\text{cur}}$ is the maximum for the day so far then save $\text{SysSensCoolCap}_{\text{cur}}$ as the design value:

$$\text{SensCoolCap}(i)_{\text{sys}} = \text{SysSensCoolCap}_{\text{cur}}$$

And save the corresponding mixed, return and outside conditions:

$$\text{CoolMixTemp}_{\text{sys}} = T_{\text{mix}}$$

$$\text{CoolMixHumRat}_{\text{sys}} = W_{\text{mix}}$$

$$\text{CoolRetTemp}_{\text{sys}} = \text{SysCoolRetTemp}(i)$$

$$\text{CoolRetHumRat}_{\text{sys}} = \text{SysCoolRetHumRat}(i)$$

$$\text{CoolOutTemp}_{\text{sys}} = T_{\text{outside}}$$

$$\text{CoolOutHumRat}_{\text{sys}} = W_{\text{outside}}$$

Here ρ_{air} is the density of dry air at 20C and standard elevation corrected pressure, [kg/m³];

FracOA is the outside air fraction;

$C_{p,\text{air}}$ is the specific heat of dry air at 20C, [J/kg-K];

T_{sup} is the user specified design cooling supply temperature [C];

T_{mix} is the current mixed air temperature [C];

W_{mix} is the current mixed air humidity ratio [kg water / kg dry air];

T_{outside} is the current outside air temperature [C];

W_{outside} is the current outside air humidity ratio [kg water / kg dry air].

(2) Loop over the zones heated by this air loop.

$$\text{HeatFlowSeq}_{\text{sys}}(i) = \sum \text{HeatFlowSeq}_{\text{zone}}(i)$$

$$\text{SysHeatRetTemp}(i) = \frac{\sum (\text{HeatZoneRetTempSeq}(i) \cdot \text{HeatFlowSeq}_{\text{zone}}(i))}{\text{HeatFlowSeq}_{\text{sys}}(i)}$$

$$\text{SysHeatRetHumRat}(i) = \frac{\sum (\text{HeatZoneHumRatSeq}(i) \bullet \text{HeatFlowSeq}_{\text{zone}}(i))}{\text{HeatFlowSeq}_{\text{sys}}(i)}$$

$$\text{FracOA} = \rho_{\text{air}} \bullet \text{DesOutAirVolFlow}_{\text{sys}} / \text{HeatFlowSeq}_{\text{sys}}(i)$$

$$T_{\text{mix}} = T_{\text{outside}} \bullet \text{FracOA} + \text{SysHeatRetTemp}(i) (1 - \text{FracOA})$$

$$W_{\text{mix}} = W_{\text{outside}} \bullet \text{FracOA} + \text{SysHeatRetHumRat}(i) (1 - \text{FracOA})$$

$$\text{SysHeatOutTempSeq}(i) = T_{\text{outside}}$$

$$\text{SysHeatOutHumRatSeq}(i) = W_{\text{outside}}$$

Get the current (zone time-step) system heating capacity:

$$\text{SysHeatCap}_{\text{cur}} = C_{p,\text{air}} \bullet \text{MinFlowRat}_{\text{sys}} \bullet \text{HeatFlowSeq}_{\text{sys}}(i) \bullet (T_{\text{sup}} - T_{\text{mix}})$$

$$\text{HeatCapSeq}(i) = \text{SysHeatCap}_{\text{cur}}$$

If $\text{SysHeatCap}_{\text{cur}}$ is the maximum for the day so far then save $\text{SysHeatCap}_{\text{cur}}$ as the design value:

$$\text{HeatCap}(i)_{\text{sys}} = \text{SysHeatCap}_{\text{cur}}$$

And save the corresponding mixed, return and outside conditions:

$$\text{HeatMixTemp}_{\text{sys}} = T_{\text{mix}}$$

$$\text{HeatMixHumRat}_{\text{sys}} = W_{\text{mix}}$$

$$\text{HeatRetTemp}_{\text{sys}} = \text{SysHeatRetTemp}(i)$$

$$\text{HeatRetHumRat}_{\text{sys}} = \text{SysHeatRetHumRat}(i)$$

$$\text{HeatOutTemp}_{\text{sys}} = T_{\text{outside}}$$

$$\text{HeatOutHumRat}_{\text{sys}} = W_{\text{outside}}$$

Here $\text{MinFlowRat}_{\text{sys}}$ is the user specified minimum supply flow ratio.

EndDay

If the user has specified *coincident* system sizing then:

$$\text{DesCoolVolFlow}_{\text{sys}} = \rho_{\text{air}} \bullet \text{CoinCoolMassFlow}_{\text{sys}}$$

$$\text{DesHeatVolFlow}_{\text{sys}} = \rho_{\text{air}} \bullet \text{CoinHeatMassFlow}_{\text{sys}}$$

$$\text{DesMainVolFlow}_{\text{sys}} = \text{Max}(\text{DesCoolVolFlow}_{\text{sys}}, \text{DesHeatVolFlow}_{\text{sys}})$$

If the user has specified *noncoincident* system sizing then:

$$\text{DesCoolVolFlow}_{\text{sys}} = \rho_{\text{air}} \bullet \text{NonCoinCoolMassFlow}_{\text{sys}}$$

$$\text{DesHeatVolFlow}_{\text{sys}} = \rho_{\text{air}} \bullet \text{NonCoinHeatMassFlow}_{\text{sys}}$$

$$\text{DesMainVolFlow}_{\text{sys}} = \text{Max}(\text{DesCoolVolFlow}_{\text{sys}}, \text{DesHeatVolFlow}_{\text{sys}})$$

EndSysSizingCalc

At this point all the calculations have been done in $\text{SysSizing}(i,j)$: we have results for each design day. Now these results need to be processed to find the heating and cooling design quantities for each system over all the design days.

For coincident sizing the task is quite easy.

(1) Loop over all of the air loops.

(a) Loop over all of the design days.

(i) If the value of DesCoolVolFlow in SysSizing for the current design day is greater than the value stored in CalcSysSizing , then move DesCoolVolFlow from SysSizing into CalcSysSizing along with CoolDesDay , CoinCoolMassFlow ,

SensCoolCap, *CoolFlowSeq(i)*, *SensCoolCapSeq(i)*, *CoolMixTemp*, *CoolRetTemp*, *CoolMixHumRat*, *CoolRetHumRat*, *CoolOutTemp*, *CoolOutHumRat*, *SysCoolRetTempSeq(i)*, *SysCoolRetHumRatSeq(i)*, *SysCoolOutTempSeq(i)* and *SysCoolOutHumRatSeq(i)*.

- (ii) If the value of *DesHeatVolFlow* in *SysSizing* for the current design day is greater than the value stored in *CalcSysSizing*, then move *DesHeatVolFlow* from *SysSizing* into *CalcSysSizing* along with *HeatDesDay*, *CoinHeatMassFlow*, *HeatCap*, *PreHeatCap*, *HeatFlowSeq(i)*, *HeatCapSeq(i)*, *PreHeatCapSeq(i)*, *HeatMixTemp*, *HeatRetTemp*, *HeatMixHumRat*, *HeatRetHumRat*, *HeatOutTemp*, *HeatOutHumRat*, *SysHeatRetTempSeq(i)*, *SysHeatRetHumRatSeq(i)*, *SysHeatOutTempSeq(i)* and *SysHeatOutHumRatSeq(i)*.

At the end of each design day loop the peak cooling and the peak heating data will be stored in *CalcSysSizing*. At this point we set *DesMainVolFlow* in *CalcSysSizing* equal to the maximum of *DesCoolVolFlow* and *DesHeatVolFlow*.

For noncoincident sizing the task is harder since we don't have a single time-step during which all the zone peaks occur. So there is no obvious value for outside air temperature at the peak, return air temperature at the peak and so forth. We must return to the zone sizing data and calculate average values for return and outside conditions.

(b) Loop over all of the zones cooled by this air loop

- (i) In *FinalZoneSizing* replace the value in *DesCoolCoilInTemp* with the user specified *CoolSupTemp_{sys}*. Do the same for *DesCoolCoilInHumRat* and *CoolSupHumRat*. This ensures that zone equipment connected to an air loop will use the system design supply air conditions as coil entering conditions.

- (ii) $NonCoinCoolMassFlow_{sys} = \sum DesCoolMassFlow_{zone}$

$$SysCoolRetTemp = \frac{\sum (ZoneRetTempAtCoolPeak \bullet DesCoolMassFlow_{zone})}{NonCoinCoolMassFlow_{sys}}$$

$$SysCoolRetHumRat = \frac{\sum (ZoneHumRatAtCoolPeak \bullet DesCoolMassFlow_{zone})}{NonCoinCoolMassFlow_{sys}}$$

$$SysCoolOutTemp = \frac{\sum (T_{OA,zone\ peak} \bullet DesCoolMassFlow_{zone})}{NonCoinCoolMassFlow_{sys}}$$

$$SysCoolOutHumRat = \frac{\sum (W_{OA,zone\ peak} \bullet DesCoolMassFlow_{zone})}{NonCoinCoolMassFlow_{sys}}$$

At the end of the zone loop calculate mixed air conditions and the system sensible cooling capacity.

$$FracOA = \rho_{air} \bullet DesOutAirVolFlow_{sys} / NonCoinCoolMassFlow_{sys}$$

$$T_{mix} = SysCoolOutTemp \bullet FracOA + SysCoolRetTemp \bullet (1 - FracOA)$$

$$W_{mix} = SysCoolOutHumRat \bullet FracOA + SysCoolRetHumRat \bullet$$

$$(1 - \text{FracOA})$$

$$\text{SysSensCoolCap} = C_{p,air} \bullet \text{NonCoinCoolMassFlow} \bullet (T_{mix} - T_{sup})$$

Then (for noncoincident sizing) the variables calculated in section (ii) are moved into the *CalcSysSizing* Array.

(c) Loop over all of the zones heated by this air loop.

- (i) In *FinalZoneSizing* replace the value in *DesHeatCoilInTemp* with the user specified *HeatSupTemp_{sys}*. Do the same for *DesHeatCoilInHumRat* and *HeatSupHumRat*. This ensures that zone equipment connected to an air loop will use the system design supply air conditions as coil entering conditions.

(ii) $\text{NonCoinHeatMassFlow}_{sys} = \sum \text{DesHeatMassFlow}_{zone}$

$$\text{SysHeatRetTemp} = \frac{\sum (\text{ZoneRetTempAtHeatPeak} \bullet \text{DesHeatMassFlow}_{zone})}{\text{NonCoinHeatMassFlow}_{sys}}$$

$$\text{SysHeatRetHumRat} = \frac{\sum (\text{ZoneHumRatAtHeatPeak} \bullet \text{DesHeatMassFlow}_{zone})}{\text{NonCoinHeatMassFlow}_{sys}}$$

$$\text{SysHeatOutTemp} = \frac{\sum (T_{OA,zone\ peak} \bullet \text{DesHeatMassFlow}_{zone})}{\text{NonCoinHeatMassFlow}_{sys}}$$

$$\text{SysHeatOutHumRat} = \frac{\sum (W_{OA,zone\ peak} \bullet \text{DesHeatMassFlow}_{zone})}{\text{NonCoinHeatMassFlow}_{sys}}$$

At the end of the zone loop calculate mixed air conditions and the system sensible cooling capacity.

$$\text{FracOA} = \rho_{air} \bullet \text{DesOutAirVolFlow}_{sys} / \text{NonCoinHeatMassFlow}_{sys}$$

$$T_{mix} = \text{SysHeatOutTemp} \bullet \text{FracOA} + \text{SysHeatRetTemp} \bullet (1 - \text{FracOA})$$

$$W_{mix} = \text{SysHeatOutHumRat} \bullet \text{FracOA} + \text{SysHeatRetHumRat} \bullet (1 - \text{FracOA})$$

$$\text{SysHeatlCap} = C_{p,air} \bullet \text{NonCoinHeatlMassFlow} \bullet (T_{sup} - T_{mix})$$

Then (for noncoincident sizing) the variables calculated in section (ii) are moved into the *CalcSysSizing* Array.

- (2) We now have the calculated system sizing data. This data needs to be altered to take into account the user input system design flow rates (if any). Note that user specified sizing ratios have already been applied to the zone sizing data which have been used in out preceding system sizing calculation. Thus the user specified sizing ratios do not have to be explicitly taken into account at the system level.

First we move the calculated system sizing data from *CalcSysSizing* array into the *FinalSysSizing* array. *FinalSysSizing* will contain the user modified system design data when we are all done.

Loop over the air loops.

- (i) As in the zone case, the user specified system design flow rates are turned into sizing ratios by dividing the user input value by the calculated value. For each air loop this gives us a $SizRat_{cool}$ and $SizRat_{heat}$.

$$CoinCoolMassFlow = SizRat_{cool} \bullet CoinCoolMassFlow_{calc}$$

$$NonCoinCoolMassFlow = SizRat_{cool} \bullet NonCoinCoolMassFlow_{calc}$$

$$DesCoolVolFlow = SizRat_{cool} \bullet DesCoolVolFlow_{calc}$$

Since the flow rates have been altered the outside air fraction will change. This will alter the design mixed air conditions and lead to an altered value for the cooling capacity. This must be done for the time-step sequence and for the peak value.

- (ii) Loop over the zone timesteps (index=i).

$$CoolFlowSeq_{sys}(i) = SizRat_{cool} \bullet CoolFlowSeq_{sys,calc}(i)$$

$$FracOA = \rho_{air} \bullet DesOutAirVolFlow_{sys} / CoolFlowSeq_{sys}(i)$$

$$T_{mix} = SysCoolOutTempSeq(i) \bullet FracOA + SysCoolRetTempSeq(i) \bullet (1 - FracOA)$$

$$SensCoolCapSeq(i) = C_{p,air} \bullet CoolFlowSeq_{sys}(i) \bullet (T_{mix} - T_{sup})$$

- (iii) Do the same calculation for peak cooling.

$$FracOA = \rho_{air} \bullet DesOutAirVolFlow_{sys} / DesCoolVolFlow$$

$$T_{mix} = CoolOutTemp_{sys} \bullet FracOA + CoolRetTemp_{sys} \bullet (1 - FracOA)$$

$$W_{mix} = CoolOutHumRat_{sys} \bullet FracOA + CoolRetHumRat_{sys} \bullet (1 - FracOA)$$

$$SensCoolCap_{sys} = C_{p,air} \bullet DesCoolVolFlow_{sys} \bullet (T_{mix} - T_{sup})$$

T_{mix} and W_{mix} are saved in *FinalSysSizing*.

- (iv) Do the same calculation for the heating case.

$$CoinHeatMassFlow = SizRat_{heat} \bullet CoinHeatMassFlow_{calc}$$

$$NonCoinHeatMassFlow = SizRat_{heat} \bullet NonCoinHeatMassFlow_{calc}$$

$$DesHeatVolFlow = SizRat_{heat} \cdot DesHeatVolFlow_{calc}$$

(v) Loop over the zone timesteps (index=i).

$$HeatFlowSeq_{sys}(i) = SizRat_{Heat} \cdot HeatFlowSeq_{sys,calc}(i)$$

$$FracOA = \rho_{air} \cdot DesOutAirVolFlow_{sys} / HeatFlowSeq_{sys}(i)$$

$$T_{mix} = SysHeatOutTempSeq(i) \cdot FracOA + \\ SysHeatRetTempSeq(i) \cdot (1 - FracOA)$$

$$HeatCapSeq(i) = C_{p,air} \cdot HeatFlowSeq_{sys}(i) \cdot (T_{sup} - T_{mix})$$

(vi) Do the same calculation for peak heating.

$$FracOA = \rho_{air} \cdot DesOutAirVolFlow_{sys} / DesHeatVolFlow$$

$$T_{mix} = HeatOutTemp_{sys} \cdot FracOA + HeatRetTemp_{sys} \cdot (1 - FracOA)$$

$$W_{mix} = HeatOutHumRat_{sys} \cdot FracOA + HeatRetHumRat_{sys} \cdot \\ (1 - FracOA)$$

$$HeatCap_{sys} = C_{p,air} \cdot DesHeatVolFlow_{sys} \cdot (T_{sup} - T_{mix})$$

T_{mix} and W_{mix} are saved in *FinalSysSizing*.

(vii) $DesMainVolFlow_{sys} = \text{MAX}(DesCoolVolFlow_{sys}, DesHeatVolFlow_{sys})$

This concludes the system design calculation.

Plant Loop Sizing

Introduction

The program needs to be able to autosize the fluid flow rate in each plant fluid loop. The design plant loop flow rates are set by the sum of the needs of the demanding components on each loop. For chilled water loops these components will be cooling coils. For hot water loops – hot water coils. And for condenser loops – various types of chiller that use condenser water for cooling. Each component that uses water for heating or cooling stores its design water flow rate (in its sizing routine) in the array *CompDesWaterFlow*, labeled by its inlet water supply node number. These individual component design water flow rates are then accessed, summed for each plant loop, and stored in the *PlantSizingData* array. This array also contains the user specified design values for each plant loop.

Hot and Chilled Water Loop Sizing

Maximum Loop Volumetric Flow Rate

The loop maximum volumetric flow rate (m^3) is just set equal to the value stored in the *PlantSizData* array for this loop.

Volume of the plant loop

Since the loop capacitance has a stability requirement of $(\dot{V} \cdot \Delta t_{step} / V) \leq 1$ the volume is set so that the stability requirement will be 0.8 at the zone time step, which is the largest time step encountered at the max flow rate the loop can reach.

$$V_{loop} = (\dot{V}_{loop, max} \cdot \Delta t_{step, zone} \cdot 3600) / 0.8$$

Condenser Loop Sizing**Maximum Loop Volumetric Flow Rate**

The loop maximum volumetric flow rate (m^3) is just set equal to the value stored in the *PlantSizData* array for this loop.

Volume of the plant loop

Since the loop capacitance has a stability requirement of $(\dot{V} \cdot \Delta t_{step} / V) \leq 1$ the volume is set so that the stability requirement will be 0.8 at the zone time step, which is the largest time step encountered at the max flow rate the loop can reach.

$$V_{loop} = (\dot{V}_{loop, max} \cdot \Delta t_{step, zone} \cdot 3600) / 0.8$$

Component Sizing**Introduction**

In EnergyPlus each HVAC component sizes itself. Each component module contains a sizing subroutine. When a component is called for the first time in a simulation, it reads in its user specified input data and then calls the sizing subroutine. This routine checks the autosizable input fields for missing data and calculates the data when needed.

A number of high-level variables are used in the sizing subroutines.

CurDuctType (in *DataSizing*) contains the information about the current duct type. The types can be *main*, *cooling*, *heating* or *other*.

CurZoneEqNum (in *DataSizing*) is the current zone equipment set index and indicates that the component is a piece of zone equipment and should size itself using the zone sizing data arrays.

CurSysNum (in *DataSizing*) is the current air loop index and indicates that the component is part of the primary air system and should size itself using the system sizing data arrays.

Fan Sizing

Fan sizing is done in subroutine *SizeFan*.

Max Flow Rate

If the fan is part of the central air system then check the duct type.

For duct type = *main*, *other* or default

$$\dot{V}_{fan, max} = DesMainVolFlow_{sys}$$

For variable volume fans the minimum flow rate is set to

$$\dot{V}_{fan, min} = DesMainVolFlow_{sys} \cdot MinFlowRat_{sys}$$

for duct type=*cooling*

$$\dot{V}_{fan, max} = DesCoolVolFlow_{sys}$$

For variable volume fans the minimum flow rate is set to

$$\dot{V}_{fan, min} = DesCoolVolFlow_{sys} \cdot MinFlowRat_{sys}$$

for duct type=*heating*

$$\dot{V}_{fan, max} = DesHeatVolFlow_{sys}$$

For variable volume fans the minimum flow rate is set to

$$\dot{V}_{fan, min} = DesHeatVolFlow_{sys} \cdot MinFlowRat_{sys}$$

If the fan is zone equipment then check whether it is part of a component that only does heating.

For heating only $\dot{V}_{fan, max} = DesHeatVolFlow_{zone}$;

Otherwise $\dot{V}_{fan, max} = \text{Max}(DesHeatVolFlow_{zone}, DesCoolVolFlow_{zone})$

If the max fan flow rate is less than *SmallAirVolFlow* the max flow rate is set to zero.

Coil:Cooling:Water

The sizing is done in subroutine *SizeWaterCoil* of module *WaterCoils*

Design Water Flow Rate (m³/s)

System Coils

The design water volumetric flow rate is calculated using:

$$WaterVolFlowRate_{coil, des} = \frac{Load_{coil, des}}{\rho_w \cdot c_{p,w} \cdot \Delta T_{w, des}}$$

$\Delta T_{w, des}$ is just the *Loop Design Temperature Difference* user input from *Sizing:Plant* (if the coil is in the outside air stream, 1/2 the *Loop Design Temperature Difference* is used). The design coil load $Load_{coil, des}$ is calculated from:

$$Load_{coil, des} = AirMassFlowRate_{coil, des} \cdot (h_{air, coil, des, in} - h_{air, coil, des, out})$$

The design air mass flow rate depends on the location of the coil. If the coil is in the outside air stream the flow rate is set to $\rho_{air} \cdot DesOutAirVolFlow_{sys}$ (the design outside air volumetric flow for the system). If the coil is in a cooling duct the flow rate is set to $\rho_{air} \cdot DesCoolVolFlow_{sys}$. If the coil is in a heating duct the flow rate is set to $\rho_{air} \cdot DesHeatVolFlow_{sys}$. If the coil is in the main duct (or any other kind of duct) the flow rate is set to $\rho_{air} \cdot DesMainVolFlow_{sys}$.

To obtain the inlet and outlet enthalpies, we need the inlet and outlet temperatures and humidity ratios. The inlet and outlet conditions depend on whether the coil is in the outside air stream and if it is not, whether or not there is outside air preconditioning.

1. Coil in outside air stream
 - a. $T_{air, in, des} = CoolOutTemp_{sys}$ (the outside air temperature at the design cooling peak)
 - b. $T_{air, out, des} = PrecoolTemp_{sys}$ (the specified Precool Design Temperature from the *Sizing:System* object).
 - c. $W_{air, in, des} = CoolOutHumRat_{sys}$ (the outside humidity ratio at the design cooling peak)
 - d. $W_{air, out, des} = PrecoolHumRat_{sys}$ (the specified Precool Design Humidity Ratio from the *Sizing:System* object)
2. Coil in main air stream, no preconditioning of outside air

- a. $T_{air,in,des} = CoolMixTemp_{sys}$ (the mixed air temperature at the design cooling peak)
 - b. $W_{air,in,des} = CoolMixHumRat_{sys}$ (the mixed air humidity ratio at the design cooling peak)
 - c. $T_{air,out,des} = CoolSupTemp_{sys}$ (the specified Central Cooling Design Supply Air Temperature from the *Sizing:System* object)
 - d. $W_{air,out,des} = CoolSupHumRat_{sys}$ (the specified Central Cooling Design Supply Air Humidity Ratio from the *Sizing:System* object)
3. Coil in main air stream, outside air preconditioned. The outside air fraction is calculated as $Frac_{oa} = DesOutAirVolFlow_{sys} / DesVolFlow$. $DesVolFlow$ is just $AirMassFlowRate_{coil,des} / \rho_{air}$.
- a. $T_{air,in,des} = Frac_{oa} \cdot PrecoolTemp_{sys} + (1 - Frac_{oa}) \cdot CoolRetTemp_{sys}$ (see Table 35. System Sizing Data)
 - b. $W_{air,in,des} = Frac_{oa} \cdot PrecoolHumRat_{sys} + (1 - Frac_{oa}) \cdot CoolRetHumRat_{sys}$
 - c. $T_{air,out,des} = CoolSupTemp_{sys}$ (the specified Central Cooling Design Supply Air Temperature from the *Sizing:System* object)
 - d. $W_{air,out,des} = CoolSupHumRat_{sys}$ (the specified Central Cooling Design Supply Air Humidity Ratio from the *Sizing:System* object)

With the inlet and outlet conditions established, we can obtain the inlet and outlet enthalpies:

$$h_{air,coil,des,in} = PsyHFnTdbW(T_{air,in,des}, W_{air,in,des})$$

$$h_{air,coil,des,out} = PsyHFnTdbW(T_{air,out,des}, W_{air,out,des})$$

where *PsyHFnTdbW* is the EnergyPlus function for calculating air specific enthalpy given the air temperature and humidity ratio. We now have all we need to calculate $Load_{coil,des}$ and $WaterVolFlowRate_{coil,des}$.

Zone Coils

If the coil is part of an *AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction* unit, the water flow rate is set equal to the terminal unit's chilled water flow rate. Otherwise (e.g., the zone-level coil is part of *ZoneHVAC:FourPipeFanCoil*, *ZoneHVAC:UnitVentilator* or *ZoneHVAC:VentilatedSlab*) the calculation is similar to that at the system level. A design load is calculated:

$$Load_{coil,des} = AirMassFlowRate_{coil,des} \cdot (h_{air,coil,des,in} - h_{air,coil,des,out})$$

Where:

$$AirMassFlowRate_{coil,des} = DesCoolMassFlow_{zone} \text{ (see Table 34. Zone Sizing Data)}$$

$$h_{air,coil,des,in} = PsyHFnTdbW(T_{air,in,des}, W_{air,in,des})$$

$$h_{air,coil,des,out} = PsyHFnTdbW(T_{air,out,des}, W_{air,out,des})$$

$$T_{air,in,des} = DesCoolCoilInTemp_{zone} \text{ (see Table 34)}$$

$$W_{air,in,des} = DesCoolCoilInHumRat_{zone} \text{ (see Table 34)}$$

$$T_{air,out,des} = CoolDesTemp_{zone} \text{ (user input from Zone:Sizing object)}$$

$$W_{air,out,des} = CoolDesHumRat_{zone} \text{ (user input from Zone:Sizing object)}$$

$$WaterVolFlowRate_{coil,des} = \frac{Load_{coil,des}}{\rho_w \cdot c_{p,w} \cdot \Delta T_{w,des}}$$

where $\Delta T_{w,des}$ is the *Loop Design Temperature Difference* user input from the *Sizing:Plant* object.

Design Air Flow Rate

System Coils

The design air volumetric flow rate depends on the location of the coil. If the coil is in the outside air stream the flow rate is set to $DesOutAirVolFlow_{sys}$. If the coil is in a cooling duct the flow rate is set to $DesCoolVolFlow_{sys}$. If the coil is in a heating duct the flow rate is set to $DesHeatVolFlow_{sys}$. If the coil is in the main duct (or any other kind of duct) the flow rate is set to $DesMainVolFlow_{sys}$.

Zone Coils

If the coil is part of an *AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction* unit, the design air volumetric flow rate is set equal to the flow rate of the terminal unit. For all other zone coils it is set equal to:

$$\text{Max}(DesCoolMassFlow_{zone}, DesHeatMassFlow_{zone}) / \rho_{air}$$

Design Inlet Air Temperature

System Coils

The inlet air temperature depends on whether the coil is in the outside air stream and if it is not, whether or not there is outside air preconditioning.

1. Coil in outside air stream: $T_{air,in} = CoolOutTemp_{sys}$ (the outside air temperature at the design cooling peak).
2. Coil in main air stream, no preconditioning of outside air: $T_{air,in} = CoolMixTemp_{sys}$ (the mixed air temperature at the design cooling peak).
3. Coil in main air stream, outside air preconditioned. The outside air fraction is calculated as $Frac_{oa} = DesOutAirVolFlow_{sys} / DesVolFlow$. $DesVolFlow$ is just $AirMassFlowRate_{coil,des} / \rho_{air}$. Then

$$T_{air,in} = Frac_{oa} \bullet PrecoolTemp_{sys} + (1 - Frac_{oa}) \bullet CoolRetTemp_{sys}$$

Zone Coils

If the coil is part of an *AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction* unit, the Design Inlet Air Temperature is set equal to $ZoneTempAtCoolPeak_{zone}$ (see Table 34. Zone Sizing Data). For all other zone coils, it is set equal to $DesCoolCoilInTemp_{zone}$ (see Table 34).

Design Outlet Air Temperature

System Coils

The outlet air temperature depends on whether the coil is in the outside air stream.

1. Coil in outside air stream: $T_{air,out,des} = PrecoolTemp_{sys}$ (the specified Precool Design Temperature from the *Sizing:System* object).
2. Coil in main air stream: $T_{air,out,des} = CoolSupTemp_{sys}$ (the specified Central Cooling Design Supply Air Temperature from the *Sizing:System* object)

Zone Coils

If the coil is part of an *AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction* unit, then:

$$Load_{coil,des} = AirMassFlowRate_{coil,des} \cdot \Delta T_{w,des} \cdot c_{p,w} \cdot \rho_w$$

$$T_{air,out,des} = T_{air,in,des} - Load_{coil,des} / (\rho_{air} \cdot c_{p,air} \cdot CoolVolFlow_{coil,air,des})$$

where $CoolVolFlow_{coil,air,des}$ is the user input or previously autosized coil Design Air Flow Rate. For all other zone coils the Design Outlet Air Temperature is set to $CoolDesTemp_{zone}$ (see Table 34. Zone Sizing Data).

Design Inlet Air Humidity Ratio

System Coils

The inlet air humidity ratio depends on whether the coil is in the outside air stream and if it is not, whether or not there is outside air preconditioning.

1. Coil in outside air stream: $W_{air,in,des} = CoolOutHumRat_{sys}$ (the outside humidity ratio at the design cooling peak).
2. Coil in main air stream, no preconditioning of outside air: $W_{air,in,des} = CoolMixHumRat_{sys}$ (the mixed air humidity ratio at the design cooling peak).
3. Coil in main air stream, outside air preconditioned. The outside air fraction is calculated as $Frac_{oa} = DesOutAirVolFlow_{sys} / DesVolFlow$. $DesVolFlow$ is just $AirMassFlowRate_{coil,des} / \rho_{air}$. Then

$$W_{air,in,des} = Frac_{oa} \cdot PrecoolHumRat_{sys} + (1 - Frac_{oa}) \cdot CoolRetHumRat_{sys}$$

Zone Coils

If the coil is part of an *AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction* unit, the Design Inlet Air Humidity Ratio is set equal to $ZoneHumRatAtCoolPeak_{zone}$ (see Table 34. Zone Sizing Data). For all other zone coils, it is set equal to $DesCoolCoilInHumRat_{zone}$ (see Table 34).

Design Outlet Air Humidity Ratio

System Coils

The outlet air humidity ratio depends on whether the coil is in the outside air stream.

1. Coil in outside air stream: $W_{air,out,des} = PrecoolHumRat_{sys}$ (the specified Precool Design Humidity Ratio from the *Sizing:System* object)
2. Coil in main air stream: $W_{air,out,des} = CoolSupHumRat_{sys}$ (the specified Central Cooling Design Supply Air Humidity Ratio from the *Sizing:System* object)

Zone Coils

The Design Outlet Air Humidity Ratio is set equal to $CoolDesHumRat_{zone}$ (user input from *Zone:Sizing*).

Design Inlet Water Temperature

System Coils

The Design Inlet Water Temperature is set to the *Design Loop Exit Temperature* specified in the *Sizing:Plant* object for the water loop serving this coil.

Zone Coils

The Design Inlet Water Temperature is set to the *Design Loop Exit Temperature* specified in the *Sizing:Plant* object for the water loop serving this coil.

Coil:Cooling:Water:DetailedGeometry Sizing

The sizing is done in subroutine *SizeWaterCoil*

Max Water Flow Rate of Coil

The calculation is identical to that done for *Coil:Cooling:Water*.

Number of Tubes per Row

$$N_{tube / row} = \text{Int}(13750 \cdot \dot{V}_{coil, water, max})$$

$$N_{tube/row} = \text{Max}(N_{tube/row}, 3)$$

Fin Diameter

Depending on the duct type, get the coil design air flow rate.

For duct type = *main, other* or default

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesMainVolFlow}_{sys}$$

for duct type=*cooling*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesCoolVolFlow}_{sys}$$

for duct type=*heating*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesHeatVolFlow}_{sys}$$

$$D_{fin} = 0.335 \cdot \dot{m}_{air, des}$$

Minimum Air Flow Area

Depending on the duct type, get the coil design air flow rate.

For duct type = *main, other* or default

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesMainVolFlow}_{sys}$$

for duct type=*cooling*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesCoolVolFlow}_{sys}$$

for duct type=*heating*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesHeatVolFlow}_{sys}$$

$$A_{MinAirFlow} = 0.44 \cdot \dot{m}_{air, des}$$

Fin Surface Area

Depending on the duct type, get the coil design air flow rate.

For duct type = *main, other* or default

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesMainVolFlow}_{sys}$$

for duct type=*cooling*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesCoolVolFlow}_{sys}$$

for duct type=*heating*

$$\dot{m}_{air, des} = \rho_{air} \cdot \text{DesHeatVolFlow}_{sys}$$

$$A_{FinSurf} = 78.5 \cdot \dot{m}_{air, des}$$

Total Tube Inside Area

$$A_{tube, total inside} = 4.4 \cdot D_{tube, inside} \cdot N_{tube rows} \cdot N_{tubes/row}$$

Where $D_{tube, inside}$ is the tube inside diameter.

Tube Outside Surf Area

$$A_{tube, outside} = 4.1 \cdot D_{tube, outside} \cdot N_{tube rows} \cdot N_{tubes/row}$$

Where $D_{tube, outside}$ is the tube outside diameter.

Coil Depth

$$Depth_{coil} = Depth_{tube\ spacing} \cdot N_{tube\ rows}$$

Coil:Heating:Water Sizing

The sizing is done in subroutine *SizeWaterCoil*.

Max Water Flow Rate of Coil

System Coils

With the coil load from the system design data array and the user specified (in a Sizing:Plant object) design hot water temperature fall, calculate the max water flow rate:

$$\dot{V}_{coil, water, max} = HeatCap_{sys} / (C_{p, water} \cdot \rho_{water} \cdot \Delta T_{plt, hw, des})$$

Zone Coils

Using the zone design coil inlet and supply air conditions calculate the design coil load.

If the coil is not part of an induction unit then obtain the coil inlet temperature from the zone design data array;

$$T_{in, air} = DesHeatCoilInTemp_{zone}$$

If the coil is part of an induction unit take into account the induced air:

$$Frac_{minflow} = MinFlowFrac_{zone}$$

$$T_{in, air} = DesHeatCoilInTemp_{zone} \cdot Frac_{minflow} + ZoneTempAtHeatPeak_{zone} \cdot (1 - Frac_{minflow})$$

$$T_{out, air} = HeatDesTemp_{zone}$$

$$W_{out, air} = HeatDesHumRat_{zone}$$

If the coil is part of a terminal unit the mass flow rate is determined by the volumetric flow rate of the terminal unit:

$$\dot{m}_{air, des} = \rho_{air} \cdot \dot{m}_{air, des, tu}$$

Otherwise the design flow is obtained from the zone design data array:

$$\dot{m}_{air, des} = DesHeatMassFlow_{zone}$$

$$Q_{coil, des} = c_{p, air} \dot{m}_{air, des} (T_{out, air} - T_{in, air})$$

Here $c_{p, air}$ is calculated at the outlet humidity and the average of the inlet and outlet temperatures.

With the coil load and the user specified (in a Sizing:Plant object) design hot water temperature decrease, calculate the max water flow rate:

$$\dot{V}_{coil, water, max} = Q_{coil, des} / (C_{p, water} \cdot \rho_{water} \cdot \Delta T_{plt, hw, des})$$

UA of the Coil

To obtain the UA of the coil, we specify the model inputs (other than the UA) at design conditions and the design coil load that the coil must meet. Then we numerically invert the coil model to solve for the UA that will enable the coil to meet the design coil load given the specified inputs.

System Coils

The design coil load is the system design sensible cooling capacity;

$$Q_{coil, des} = HeatCap_{sys}$$

The required inputs for the simple coil model are:

$$T_{in, air} = HeatMixTemp_{sys}$$

$$W_{in, air} = HeatMixHumRat_{sys}$$

$$T_{in,water} = ExitTemp_{plt,hw,des}$$

$$\dot{m}_{in,water} = \rho_{water} \cdot \dot{V}_{coil,water,max}$$

Depending on the duct type, get the coil design air flow rate.

For duct type = *main*, *other* or default

$$\dot{m}_{in,air} = \rho_{air} \cdot DesMainVolFlow_{sys}$$

for duct type=*cooling*

$$\dot{m}_{in,air} = \rho_{air} \cdot DesCoolVolFlow_{sys}$$

for duct type=*heating*

$$\dot{m}_{in,air} = \rho_{air} \cdot DesHeatVolFlow_{sys}$$

We now have all the data needed to obtain UA. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function. In this case it finds the UA that will zero the residual function – the difference between the design coil load and the coil output divided by the design coil load. The residual is calculated in the function *SimpleHeatingCoilUAResidual*.

Zone Coils

If the coil is not part of an induction unit then obtain the coil inlet temperature from the zone design data array;

$$T_{in,air} = DesHeatCoilInTemp_{zone}$$

If the coil is part of an induction unit take into account the induced air:

$$Frac_{minflow} = MinFlowFrac_{zone}$$

$$T_{in,air} = DesHeatCoilInTemp_{zone} \cdot Frac_{minflow} + ZoneTempAtHeatPeak_{zone} \cdot (1 - Frac_{minflow})$$

$$W_{in,air} = DesHeatCoilInHumRat_{zone}$$

$$T_{in,water} = ExitTemp_{plt,hw,des}$$

$$\dot{m}_{in,water} = \rho_{water} \cdot \dot{V}_{coil,water,max}$$

$$T_{out,air} = HeatDesTemp_{zone}$$

$$W_{out,air} = HeatDesHumRat_{zone}$$

If the coil is part of a terminal unit the mass flow rate is determined by the volumetric flow rate of the terminal unit:

$$\dot{m}_{air,des} = \rho_{air} \cdot \dot{m}_{air,des,tu}$$

Otherwise the design flow is obtained from the zone design data array:

$$\dot{m}_{air,des} = DesHeatMassFlow_{zone}$$

$$\dot{Q}_{coil,des} = c_{p,air} \cdot \dot{m}_{air,des} \cdot (T_{out,air} - T_{in,air})$$

Here $c_{p,air}$ is calculated at the outlet humidity and the average of the inlet and outlet temperatures.

We now have all the data needed to obtain UA. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function. In this case it finds the UA that will zero the residual function – the difference between the design coil load and the coil output divided by the design coil load. The residual is calculated in the function *SimpleHeatingCoilUAResidual*.

Sizing of Gas and Electric Heating Coils

The sizing calculation is done in subroutine *SizeHeatingCoil* in module *HeatingCoils*.

Nominal Capacity of the Coil

System Coils

The value is obtained from the system design array.

$$Cap_{nom} = HeatCap_{sys}$$

Zone Coils

The capacity is calculated from the design coil inlet and outlet conditions.

If the coil is not part of an induction unit then obtain the coil inlet temperature from the zone design data array;

$$T_{in,air} = DesHeatCoilInTemp_{zone}$$

If the coil is part of an induction unit take into account the induced air:

$$Frac_{minflow} = MinFlowFrac_{zone}$$

$$T_{in,air} = DesHeatCoilInTemp_{zone} \cdot Frac_{minflow} + ZoneTempAtHeatPeak_{zone} \cdot (1 - Frac_{minflow})$$

$$T_{out,air} = HeatDesTemp_{zone}$$

$$W_{out,air} = HeatDesHumRat_{zone}$$

$$Q_{coil,des} = C_{p,air} \cdot DesHeatMassFlow_{zone} \cdot (T_{out,air} - T_{in,air})$$

Here $c_{p,air}$ is calculated at the outlet humidity and the average of the inlet and outlet temperatures.

DX Coil Sizing

The sizing calculations are done in subroutine *SizeDXCoil* in module *DXCoils*. This section covers the sizing of the objects

1. Coil:Cooling:DX:SingleSpeed
2. Coil:Heating:DX:SingleSpeed
3. Coil:Cooling:DX:TwoSpeed

Rated Air Volume Flow Rate

System Coils

The rated air flow rate is obtained from the system design array.

$$\dot{V}_{air,rated} = DesMainVolFlow_{sys}$$

Zone Coils

The rated air flow rate is the maximum of the heating and cooling design flow rates from the zone design array.

$$\dot{V}_{air,rated} = \text{Max}(DesCoolVolFlow_{zone}, DesHeatVolFlow_{zone})$$

Rated Total Cooling Capacity

System Coils

The rated cooling capacity is obtained by dividing the peak cooling capacity by the *Cooling Capacity Modifier Curve* evaluated at peak mixed wetbulb and outdoor drybulb temperatures.

$$T_{mix} = CoolMixTemp_{sys}$$

$$W_{mix} = CoolMixHumRat_{sys}$$

$$T_{sup} = CoolSupTemp_{sys}$$

$$W_{sup} = CoolSupHumRat_{sys}$$

$$T_{outside} = CoolOutTemp_{sys}$$

$$\rho_{air} = PsyRhoAirFnPbTdbW(\rho_{air,std}, T_{mix}, W_{mix})$$

$$h_{mix} = \text{PsyHFnTdbW}(T_{mix}, W_{mix})$$

$$h_{sup} = \text{PsyHFnTdbW}(T_{sup}, W_{sup})$$

$$T_{mix,wb} = \text{PsyTwbFnTdbWPb}(T_{mix}, W_{mix}, p_{air,std})$$

$$\text{CapModFac} = \text{CurveValue}(\text{CCapFTemp}, T_{mix,wb}, T_{outside})$$

$$\text{CCap}_{peak} = \rho_{air} \cdot \dot{V}_{air, rated} \cdot (h_{mix} - h_{sup})$$

$$\text{CCap}_{rated} = \text{CCap}_{peak} / \text{CapModFac}$$

We check that the design volume flow per total capacity is within the prescribed range:

$$\text{FlowCapRatio} = \dot{V}_{air, rated} / \text{CCap}_{rated}$$

If $\text{FlowCapRatio} < \text{FlowCapRatio}_{min}$ then

$$\text{CCap}_{rated} = \dot{V}_{air, rated} / \text{FlowCapRatio}_{min}$$

If $\text{FlowCapRatio} > \text{FlowCapRatio}_{max}$ then

$$\text{CCap}_{rated} = \dot{V}_{air, rated} / \text{FlowCapRatio}_{max}$$

where

$$\text{FlowCapRatio}_{min} = 0.00004027 \text{ m}^3/\text{s per watt (300 cfm/ton)}$$

And

$$\text{FlowCapRatio}_{max} = 0.00006041 \text{ m}^3/\text{s per watt (450 cfm/ton)}$$

Zone Coils

The rated cooling capacity for zone coils is calculated in the same manner as for system coils.

$$T_{mix} = \text{DesCoolCoilInTemp}_{zone}$$

$$W_{mix} = \text{DesCoolCoilInHumRat}_{zone}$$

$$T_{sup} = \text{CoolDesTemp}_{zone}$$

$$W_{sup} = \text{CoolDesHumRat}_{zone}$$

$$T_{outside} = T_{outside, desday, peak}$$

$$\rho_{air} = \text{PsyRhoAirFnPbTdbW}(p_{air,std}, T_{mix}, W_{mix})$$

$$h_{mix} = \text{PsyHFnTdbW}(T_{mix}, W_{mix})$$

$$h_{sup} = \text{PsyHFnTdbW}(T_{sup}, W_{sup})$$

$$T_{mix,wb} = \text{PsyTwbFnTdbWPb}(T_{mix}, W_{mix}, p_{air,std})$$

$$\text{CapModFac} = \text{CurveValue}(\text{CCapFTemp}, T_{mix,wb}, T_{outside})$$

$$\text{CCap}_{peak} = \rho_{air} \cdot \dot{V}_{air, rated} \cdot (h_{mix} - h_{sup})$$

$$\text{CCap}_{rated} = \text{CCap}_{peak} / \text{CapModFac}$$

We check that the design volume flow per total capacity is within the prescribed range:

$$\text{FlowCapRatio} = \dot{V}_{air, rated} / \text{CCap}_{rated}$$

If $\text{FlowCapRatio} < \text{FlowCapRatio}_{min}$ then

$$\text{CCap}_{rated} = \dot{V}_{air, rated} / \text{FlowCapRatio}_{min}$$

If $\text{FlowCapRatio} > \text{FlowCapRatio}_{max}$ then

$$\text{CCap}_{rated} = \dot{V}_{air, rated} / \text{FlowCapRatio}_{max}$$

where

$$\text{FlowCapRatio}_{min} = 0.00004027 \text{ m}^3/\text{s per watt (300 cfm/ton)}$$

And

$$\text{FlowCapRatio}_{max} = 0.00006041 \text{ m}^3/\text{s per watt (450 cfm/ton)}$$

Rated Total Heating Capacity

For :Heating:DX:SingleSpeed the rated heating capacity is set equal to the cooling capacity.

Rated SHR

The rated sensible heat ratio is calculated to be the sensible cooling (from rated inlet conditions to user specified supply conditions) divided by the total cooling (from rated inlet to specified supply).

$$T_{in, rated} = 26.6667 \text{ } ^\circ\text{C} \text{ (80 } ^\circ\text{F)}$$

$$W_{in, rated} = 0.01125 \text{ (corresponds to 80 } ^\circ\text{F drybulb, 67 } ^\circ\text{F wetbulb)}$$

$$C_{p, air} = \text{PsyCpAirFnWTdb}(W_{in, rated}, T_{in, rated})$$

For system coils

$$T_{sup} = \text{CoolSupTemp}_{sys}$$

$$W_{sup} = \text{CoolSupHumRat}_{sys}$$

For zone coils

$$T_{sup} = \text{CoolDesTemp}_{zone}$$

$$W_{sup} = \text{CoolDesHumRat}_{zone}$$

Then

$$h_{rated} = \text{PsyHFnTdbW}(T_{in, rated}, W_{in, rated})$$

$$h_{sup} = \text{PsyHFnTdbW}(T_{sup}, W_{sup})$$

$$\Delta h_{rated, sup} = h_{rated} - h_{sup}$$

$$\Delta Qs_{rated, sup} = C_{p, air} \cdot (T_{in, rated} - T_{sup})$$

$$SHR_{rated} = \Delta Qs_{rated, sup} / \Delta h_{rated, sup}$$

Evaporative Condenser Air Volume Flow Rate

The evaporative condenser air volume flow rate (m^3/s) is set to $0.000114 \text{ m}^3/\text{s}$ per watt (850 cfm/ton) times the total rated cooling capacity.

Evaporative Condenser Air Volume Flow Rate, Low Speed

The evaporative condenser air volume flow rate, low speed (m^3/s) is set to 1/3 times $0.000114 \text{ m}^3/\text{s}$ per watt (850 cfm/ton) times the total rated cooling capacity.

Evaporative Condenser Pump Rated Power Consumption

The evaporative condenser pump rated power consumption is set equal to the total cooling capacity times 0.004266 watts pump power per watt capacity (15 W/ton).

Evaporative Condenser Pump Rated Power Consumption, Low Speed

The evaporative condenser pump rated power consumption, low speed, is set equal to 1/3 times the total cooling capacity times 0.004266 watts pump power per watt capacity (15 W/ton).

Rated Air Volume Flow Rate, low speed

The rated air volume flow rate, low speed, is set equal to 1/3 times the full rated air volume flow rate.

Rated Total Cooling Capacity, Low Speed

The rated total cooling capacity, low speed, is set equal to 1/3 times the full rated total cooling capacity.

Rated SHR, low speed

The rated sensible heat ratio, low speed, is set equal to the full speed SHR.

Resistive Defrost Heater Capacity

For the heat pump the resistive defrost heat capacity is set equal to the cooling capacity.

DX MultiSpeed Coil Sizing

The sizing calculations are done in subroutine *SizeDXCoil* in module *DXCoils*. This section covers the sizing of the objects

- 1) Coil:Heating:DX:MultiSpeed
- 2) Coil:Cooling:DX: MultiSpeed

The rated air volume flow rate, rated total cooling capacity, rated heating capacity, rated SHR, evaporative condenser air volume flow rate, evaporative condenser pump rated power consumption at the highest speed are sized in the same ways as DX Coil Sizing.

After the sizes are determined at the highest speed, the sizes in the rest of speeds are assumed to

$$Value_n = \frac{n}{NumberOfSpeed} * Value_{NumberOfSpeed}$$

where

Value_n = Any autosizable variable at Speed n, except SHR

SHR_n = SHR_{NumberOfSpeed}

n = Speed Index number from 1 to NumberOfSpeed-1

NumberOfSpeed = The highest speed number

Pump Sizing

The loop pumps' autosizable inputs are nominal volumetric flow rate and nominal power consumption. We have

$$Eff_{tot} = Eff_{mot} \cdot Eff_{impeller}$$

The motor efficiency is an input. Since we need the total efficiency to calculate the nominal power consumption we assume an impeller efficiency of 0,78 for purposes of sizing.

Rated Volumetric Flow Rate

This is just set equal to the design loop demand obtained from summing the needs of the components on the demand side of the loop.

Rated Power Consumption

$$\dot{Q}_{nom} = H_{nom} \cdot \dot{V}_{nom} / Eff_{tot}$$

H_{nom} , the nominal head, is an input.

Electric Chiller Sizing

Generally chillers will need nominal cooling capacity, evaporator flow rate and condenser flow rate. All 3 quantities can be straightforwardly obtained using the user specified loop sizing data and the loop design flow rates.

All chillers on a loop are sized to meet the full loop load. If there are multiple chillers on a loop that call for autosizing, they will all be assigned the same cooling capacity and evaporator flow rate.

Nominal Cooling Capacity

$$\dot{Q}_{chiller, nom} = C_{p,w} \cdot \rho_w \cdot \Delta T_{loop, des} \cdot \dot{V}_{loop, des}$$

where

$C_{p,w}$ is the specific heat of water at 5 °C;

ρ_w is the density of water at standard conditions (5.05 °C);

$\Delta T_{loop,des}$ is the chilled water loop design temperature rise;

$\dot{V}_{loop,des}$ is the loop design volumetric flow rate.

Design Evaporator Volumetric Water Flow Rate

$$\dot{V}_{evap,des} = \dot{V}_{loop,des}$$

Design Condenser Volumetric Water Flow Rate

$$\dot{V}_{cond,des} = \dot{Q}_{chiller,nom} \cdot (1 + 1 / COP_{chiller,nom}) / (\Delta T_{loop,des} \cdot C_{p,w} \cdot \rho_w)$$

where

$C_{p,w}$ is the specific heat of water at design condenser inlet temperature;

ρ_w is the density of water at standard conditions (5.05 °C);

$\Delta T_{loop,des}$ is the chilled water loop design temperature rise;

$COP_{chiller,nom}$ is the chiller nominal COP.

Boiler Sizing

Generally boilers will need nominal heating capacity and rate. Both quantities can be straightforwardly obtained using the user specified loop sizing data and the loop design flow rates.

All boilers on a loop are sized to meet the full loop load. If there are multiple boilers on a loop that call for autosizing, they will all be assigned the same heating capacity and flow rate.

Nominal Capacity

$$\dot{Q}_{boiler,nom} = C_{p,w} \cdot \rho_w \cdot \Delta T_{loop,des} \cdot \dot{V}_{loop,des}$$

where

$C_{p,w}$ is the specific heat of water at the boiler design outlet temperature;

ρ_w is the density of water at standard conditions (5.05 °C);

$\Delta T_{loop,des}$ is the hot water loop design temperature decrease;

$\dot{V}_{loop,des}$ is the loop design volumetric flow rate.

Design Evaporator Volumetric Water Flow Rate

$$\dot{V}_{des} = \dot{V}_{loop,des}$$

Cooling Tower Sizing

The quantities needed to autosize a cooling tower include the design water flow rate, the nominal fan power and air flow rate, and the tower UA. This data may be need to be given at more than one operating point:, for instance – high speed fan, low speed fan and free convection.

EnergyPlus provides two input choices: the user can input the design water flow rate and tower UA at each operating point or the tower nominal capacity (and let the program calculate the water flow rate and UA). Choice of input method will affect the sizing calculations in ways noted below.

Design Water Flow Rate

If *Tower Performance Input Method = UA and Design Water Flow Rate* then

$$\dot{V}_{tower,w,des} = \dot{V}_{loop,des}$$

If *Tower Performance Input Method = Nominal Capacity* then

$$\dot{V}_{tower, w, des} = 5.382E - 8 \cdot \dot{Q}_{tower, nom}$$

where $5.382 \cdot 10^{-08}$ is m^3/s per watt corresponds to the rule-of-thumb of sizing the tower flow rate at 3 gallons per minute per ton.

Fan Power at Design Air Flow Rate

The nominal fan power is sized to be 0.0105 times the design load.

If *Tower Performance Input Method = UA and Design Water Flow Rate* then

$$\dot{Q}_{tower, nom} = C_{p, w} \cdot \rho_w \cdot \dot{V}_{tower, w, des} \cdot \Delta T_{loop, des}$$

where

$C_{p, w}$ is the specific heat of water at the condenser loop design exit temperature;

ρ_w is the density of water at standard conditions (5.05 °C);

$\Delta T_{loop, des}$ is the condenser water loop design temperature rise;

Finally

$$\dot{Q}_{fan, nom} = 0.0105 \cdot \dot{Q}_{tower, nom}$$

Design Air Flow Rate

We assume a fan efficiency of 0.5 and a fan pressure rise of 190 Pascals. Then

$$\dot{V}_{tower, air, des} = \dot{Q}_{fan, nom} \cdot 0.5 \cdot \rho_{air} / 190$$

where

ρ_{air} is the density of air at standard conditions.

Tower UA Value at Design Air Flow Rate

To obtain the UA of the tower, we specify the model inputs (other than the UA) at design conditions and the design tower load that the tower must meet. Then we numerically invert the tower model to solve for the UA that will enable the tower to meet the design tower load given the specified inputs.

The design tower load is:

for *Tower Performance Input Method = UA and Design Water Flow Rate*

$$\dot{Q}_{tower, des} = C_{p, w} \cdot \rho_w \cdot \dot{V}_{tower, w, des} \cdot \Delta T_{loop, des}$$

for *Tower Performance Input Method = Nominal Capacity*

$$\dot{Q}_{tower, des} = 1.25 \cdot \dot{Q}_{tower, nom} \text{ (to allow for compressor heat)}$$

Then we assign the inputs needed for the model.

$T_{in, air} = 35$ °C (95 °F design air inlet temperature)

$T_{in, air, wb} = 25.6$ °C (78 °F design air inlet wetbulb temperature)

W_{in} is calculated from the entering air drybulb and wetbulb.

The inlet water mass flow rate is just the design water volumetric flow rate times the density of water.

The inlet water temperature is set slightly differently for the 2 input methods. For

UA and Design Water Flow Rate

$$T_{in, water} = T_{loop, exit, des} + \Delta T_{loop, des}$$

Nominal Capacity

$$T_{in, water} = 35$$
 °C (95 °F design inlet water temperature).

We now have all the data needed to obtain UA. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function. In this case it finds the UA that will zero the residual function – the difference

between the design tower load and the tower output divided by the design tower load. The residual is calculated in the function *SimpleTowerUAResidual*.

Air Flow Rate at Low Fan Speed

The nominal air flow rate at low fan speed is set to one-half of the full speed air flow rate.

Fan Power at Low Fan Speed

The fan power at low fan speed is set to 0.16 times the fan power at full speed.

Tower UA Value at Low Fan Speed

For *Tower Performance Input Method = UA and Design Water Flow Rate* the low speed UA is set to 0.6 times the full speed UA. For *Tower Performance Input Method = Nominal Capacity* the low speed UA is calculated in the same manner as the full speed UA using $\dot{Q}_{tower, nom, lowspeed}$ instead of $\dot{Q}_{tower, nom}$.

Air Flow Rate in Free Convection Regime

The free convection air flow rate is set to 0.1 times the full air flow rate.

Tower UA Value in Free Convection Regime

For *Tower Performance Input Method = UA and Design Water Flow Rate* the low speed UA is set to 0.1 times the full speed UA. For *Tower Performance Input Method = Nominal Capacity* the low speed UA is calculated in the same manner as the full speed UA using $\dot{Q}_{tower, nom, freeconv}$ instead of $\dot{Q}_{tower, nom}$.

Fan Coil Unit Sizing

Fan Coil units are compound components: each unit contains a fan, hot water coil, chilled water coil and outside air mixer. The inputs that may need to be autosized are the nominal unit air flow rate, the maximum hot and chilled water flow rates, and the design outside air flow rate. The data needed for sizing the units is obtained from the zone design arrays and the user specified plant sizing input.

Maximum Air Flow Rate

$$\dot{V}_{air, max} = \text{Max}(\text{DesCoolVolFlow}_{zone}, \text{DesHeatVolFlow}_{zone})$$

Maximum Outside Air Flow Rate

$$\dot{V}_{outsideair, max} = \text{Min}(\text{MinOA}_{zone}, \dot{V}_{air, max})$$

Maximum Hot Water Flow

$$T_{coil, in} = \text{DesHeatCoilInTemp}_{zone}$$

$$T_{coil, out} = \text{HeatDesTemp}_{zone}$$

$$\dot{Q}_{coil, des} = c_{p, air} \cdot \text{DesHeatMassFlow}_{zone} \cdot (T_{out, coil} - T_{in, coil})$$

$$\dot{V}_{max, hw} = \dot{Q}_{coil, des} / (c_{p, w} \cdot \rho_w \cdot \Delta T_{loop, des})$$

where

$c_{p, air}$ is evaluated at the average of the inlet & outlet temperatures and the coil outlet humidity ratio.

Maximum Cold Water Flow

$$T_{coil, in} = \text{DesColdCoilInTemp}_{zone}$$

$$T_{coil, out} = \text{ColdDesTemp}_{zone}$$

$$W_{coil, in} = \text{DesCoolCoilInHumRat}_{zone}$$

$$\begin{aligned}
 W_{coil,out} &= CoolDesHumRat_{zone} \\
 H_{coil,in} &= PsyHF_nTdbW(T_{coil,in}, W_{coil,in}) \\
 H_{coil,out} &= PsyHF_nTdbW(T_{coil,out}, W_{coil,out}) \\
 \dot{Q}_{coil,des} &= DesCoolMassFlow_{zone} \cdot (h_{in,coil} - h_{out,coil}) \\
 \dot{V}_{max,hw} &= \dot{Q}_{coil,des} / (c_{p,w} \cdot \rho_w \cdot \Delta T_{loop,des})
 \end{aligned}$$

where

$c_{p,air}$ is evaluated at the average of the inlet & outlet temperatures and the coil outlet humidity ratio.

Window Air Conditioner Sizing

Window air conditioners are compound components: each unit contains a fan, A DX coil and outside air mixer. The inputs that may need to be autosized are the nominal unit air flow rate and the design outside air flow rate. The data needed for sizing the units is obtained from the zone design arrays.

Maximum Air Flow Rate

$$\dot{V}_{air,max} = \text{Max}(DesCoolVolFlow_{zone}, DesHeatVolFlow_{zone})$$

Maximum Outside Air Flow Rate

$$\dot{V}_{outsideair,max} = \text{Min}(MinOA_{zone}, \dot{V}_{air,max})$$

Packaged Terminal Heat Pump Sizing

Packaged terminal heat pumps are compound components: each unit contains a supply air fan, a DX cooling coil, a DX heating coil, a GAS or ELECTRIC supplemental heating coil, and an outside air mixer. The inputs that may need to be autosized are the supply air and outside air volumetric air flow rates during cooling operation, heating operation, and when no cooling or heating is needed. In addition, the maximum supply air temperature from the supplemental heater can also be automatically selected. The data needed for sizing the units are obtained from the zone design arrays.

Supply air volumetric flow rate during cooling operation

$$\dot{V}_{SA,cooling} = \text{MAX}(DesCoolVolFlow_{zone}, DesHeatVolFlow_{zone})$$

Supply air volumetric flow rate during heating operation

$$\dot{V}_{SA,heating} = \text{MAX}(DesCoolVolFlow_{zone}, DesHeatVolFlow_{zone})$$

Supply air volumetric flow rate when no cooling or heating is needed

$$\dot{V}_{SA,no\,cool\,or\,heat} = \text{MAX}(DesCoolVolFlow_{zone}, DesHeatVolFlow_{zone})$$

Outside air volumetric flow rate during cooling operation

$$\dot{V}_{OA,cooling} = \text{MIN}(MinOA_{zone}, \dot{V}_{SA,cooling})$$

Outside air volumetric flow rate during heating operation

$$\dot{V}_{OA, heating} = \text{MIN}(\text{MinOA}_{zone}, \dot{V}_{SA, heating})$$

Outside air volumetric flow rate when no cooling or heating is needed

$$\dot{V}_{OA, no cool or heat} = \text{MIN}(\text{MinOA}_{zone}, \dot{V}_{SA, no cool or heat})$$

Maximum supply air temperature from supplemental heater

$$T_{SA, max} = \text{HeatDesTemp}_{zone}$$

MultiSpeed Heat Pump Sizing

MultiSpeed heat pumps are compound components: each unit contains a supply air fan, a multispeed DX cooling coil, a multispeed DX heating coil, and a GAS or ELECTRIC supplemental heating coil. The inputs that may need to be autosized are the supply air volumetric air flow rates during cooling operation, heating operation, and when no cooling or heating is needed. The data needed for sizing the units are obtained from the controlled zone design arrays.

Supply air volumetric flow rate during cooling operation at the highest speed

$$\dot{V}_{SA, cooling} = \text{MAX}(\text{DesCoolVolFlow}_{zone}, \text{DesHeatVolFlow}_{zone}) / \text{ZoneFraction}$$

Supply air volumetric flow rate during heating operation at the highest speed

$$\dot{V}_{SA, heating} = \text{MAX}(\text{DesCoolVolFlow}_{zone}, \text{DesHeatVolFlow}_{zone}) / \text{ZoneFraction}$$

Supply air volumetric flow rate when no cooling or heating is needed

$$\dot{V}_{SA, no cool or heat} = \text{MAX}(\text{DesCoolVolFlow}_{zone}, \text{DesHeatVolFlow}_{zone}) / \text{ZoneFraction}$$

where

ZoneFraction = Fraction of the total volume flow that goes through the controlling zone

Supply air volumetric flow rate during cooling operation at Speed n (1 to NumberOfSpeed-1)

$$\dot{V}_{SA, cooling, n} = \frac{n}{\text{NumberOfSpeed}} \dot{V}_{SA, cooling}$$

Supply air volumetric flow rate during heating operation at Speed n (1 to NumberOfSpeed-1)

$$\dot{V}_{SA, heating, n} = \frac{n}{\text{NumberOfSpeed}} \dot{V}_{SA, heating}$$

where

n = Speed Index number from 1 to NumberOfSpeed-1

NumberOfSpeed = The highest speed number

Single Duct Terminal Units

These are the AirTerminal:SingleDuct:ConstantVolume:Reheat, AirTerminal:SingleDuct:VAV:Reheat, AirTerminal:SingleDuct:VAV:NoReheat, AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat, and AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat objects. The inputs that may need to be autosized are the maximum air flow rate through the terminal unit and the maximum hot water flow rate through the reheat coil, if a hot water or steam reheat coil is present.

Maximum Air Flow Rate

$$\dot{V}_{air, max} = \text{Max}(\text{DesCoolVolFlow}_{zone}, \text{DesHeatVolFlow}_{zone})$$

Max Reheat Water Flow

$$T_{coil, in} = \text{DesHeatCoilInTemp}_{zone}$$

$$T_{coil, out} = \text{HeatDesTemp}_{zone}$$

The design air flow for heating depends on the terminal unit characteristics. If the terminal unit is VAV and reverse action, constant volume, or VAV:HeatAndCool then:

$$\dot{V}_{air, coil, heating} = \dot{V}_{air, max}$$

If the terminal unit is VAV and normal action then:

$$\dot{V}_{air, coil, heating} = \text{Frac}_{air, min} \cdot \dot{V}_{air, max},$$

where, $\text{Frac}_{air, min}$ corresponds to the minimum flow fraction of the terminal unit. This value is provided as user input, typically as the field "Zone Minimum Air Flow Fraction." For the VAV terminals that allow scheduling minimum flow fraction (e.g. AirTerminal:SingleDuct:VAV:Reheat), there are two ways that $\text{Frac}_{air, min}$ can be determined. If a value is entered in the input field Constant Minimum Air Flow Fraction, then it is always used for $\text{Frac}_{air, min}$. If the minimum air flow fraction method is "Schedule" and the Constant Minimum Air Flow Fraction is left blank, then the program uses the average of the minimum and maximum values in the schedule for $\text{Frac}_{air, min}$.

The coil load and max hot water flow rate are then:

$$\dot{Q}_{coil, des} = c_{p, air} \cdot \rho_{air} \cdot \dot{V}_{air, coil, heating} \cdot (T_{out, coil} - T_{in, coil})$$

$$\dot{V}_{max, hw} = \dot{Q}_{coil, des} / (c_{p, w} \cdot \rho_w \cdot \Delta T_{loop, des})$$

where

$c_{p, air}$ is evaluated at the average of the inlet & outlet temperatures and the coil outlet humidity ratio.

Indirect Evaporative Cooler Sizing

The model for the object called EvaporativeCooler:Indirect:ResearchSpecial has a field for the secondary fan flow rate that can be autosized.

Secondary Fan Flow Rate

The secondary fan is not part of an airstream that is directly modeled in EnergyPlus. Because the primary side air flows can be autosized as part of the air system, it is convenient to also scale the size of the secondary flow. If the cooler is part of the main loop of a central air system, then the secondary fan flow rate is sized to equal to the main design flow rate.

$$\dot{V}_{fan,max} = DesMainVolFlow_{sys}$$

If the cooler is part of the outdoor air path of a central air system, then the secondary fan flow rate is sized to be the maximum of either the design minimum outdoor air flow rate or one-half of the main design flow rate.

$$\dot{V}_{fan,max} = MAX(DesOutAirVolFlow, 0.5*DesMainVolFlow)$$

References

- ASHRAE Fundamentals 2001. 2001 ASHRAE Fundamentals Handbook. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- Pedersen, C.O., D.E. Fisher, and R.J. Liesen. 1997. A heat balance based cooling load calculation procedure. ASHRAE Transactions, Vol. 103(2), pp. 459-468.
- Pedersen, C.O. 2001. Toolkit for Building Load Calculations. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Demand Limiting

Demand limiting, or demand management, is a strategy for reducing a building's demand for utilities, e.g., electricity. Utility companies typically charge a monthly fee for "demand charges" that is based on the peak demand during a certain time period. Often the peak demand charge is set by one exceptional day that is much higher than the peak demand for an average day during the month. Therefore, to save utility costs, it is in the interest of building owners to find ways to manage the demand on peak days. While demand management is generally concerned with the demand for electricity, the future will likely see demand management of other utilities, such as natural gas or water.

Demand limiting controls shut off or reduce the power to non-essential loads in order to reduce the overall building demand. Some typical controls:

- shut off or dim electric lights, equipment, or HVAC systems
- reset the thermostatic setpoints on HVAC systems
- reduce the load of a set of similar components by rotating one or more components "off" for a short time interval
- turn on generators to meet some or all of the building's demand.

The demand limiting controls implemented in EnergyPlus are intended to allow some of the more common demand limiting strategies. The DemandManagerAssignmentList object is a high level control that makes demand limiting decisions based on a list of possible demand limiting strategies. Each demand limiting strategy is described in a separate DemandManager object. Each DemandManager object controls a group of similar load objects of the same type, such as DemandManager:Lights, DemandManager:ElectricEquipment, or DemandManager:Thermostats objects.

Algorithm

In EnergyPlus the DemandManagerAssignmentList and DemandManager objects are simulated by the Demand Manager module. The Demand Manager is built into the overall solution method for the program. For each zone time step, the program executes three major segments of code:

- exterior energy use
- zone heat balance (surface heat balances, internal gains, and air flows)
- HVAC system simulation (air and plant loops)

The exterior energy use segment is completely independent of the zone heat balance and HVAC system simulation. Exterior energy use handles energy use accounting for exterior lights and exterior equipment that are outside of the building and are not part of the zone heat balance. The zone heat balance segment includes all of the surface heat balances, internal heat gains, and air flows. The HVAC system simulation includes air and plant loops with their associated HVAC components. The behaviour of the HVAC system depends on the results of the zone heat balance. The HVAC system simulation operates on a variable "system" time step which is automatically shortened if necessary for stability.

The Demand Manager is called after the first pass through the HVAC system simulation, before the system time step is shortened. After evaluating the DemandManagerAssignmentList object, the Demand Manager decides if demand limiting is required. If demand limiting is required, the individual DemandManager objects are surveyed to determine which loads can be limited. Based on the *Demand Manager Priority* selected, the Demand Manager then decides which DemandManager objects should be activated. In turn, the activated DemandManager objects set the demand limiting hooks on their respective load objects. Finally, depending on the type of DemandManager objects that were activated, one or more of the major segments of code must be called to be resimulated because the load conditions have changed. The code segments depend on the type of DemandManager

and the relationship of its load objects to the overall solution method. The table below shows the different DemandManager types and the related code segments that must be resimulated.

Table 36. Demand Manager Types and Resimulation.

<i>Demand Manager Type</i>	<i>Resimulate Exterior Energy</i>	<i>Resimulate Zone Heat Balance</i>	<i>Resimulate HVAC System</i>
DemandManager:ExteriorLights	✓		
DemandManager:Lights		✓	✓
DemandManager:ElectricEquipment		✓	✓
DemandManager:Thermostats			✓

All demand limiting controls are disabled during warmup days and sizing runs.

Sequential Priority

If the *Demand Manager Priority* field of the DemandManagerAssignmentList object is set to SEQUENTIAL, each DemandManager in the list is activated in sequence from first to last until demand is reduced below the limit or until all managers are activated. A DemandManager is skipped if it cannot reduce the demand. Possible reasons that a manager cannot reduce demand include:

- not enough load to limit
- not available because of its *Availability Schedule*
- already activated; load limited during a previous time step.

For each DemandManager in the list that is successfully activated, one or more of the major code segments must be called to be resimulated (see above). The DemandManagerAssignmentList object is then reevaluated to determine if further demand limiting is required before the next DemandManager is activated.

The implication of sequential priority is that a DemandManagerAssignmentList with many DemandManagers may make many passes through the Demand Manager before all demand limiting decisions are finalized. Each pass requires an additional resimulation of the current time step. In some cases this could significantly slow down the overall simulation runtime. The use of the *Minimum Limit Duration* field can help to ensure that not every DemandManager is being activated at every time step.

All Priority

If the *Demand Manager Priority* field of the DemandManagerAssignmentList object is set to ALL, all DemandManagers in the list are activated simultaneously to achieve the maximum demand reduction. This priority option requires only one final resimulation after all DemandManagers are activated.

Alternative Modeling Processes

RoomAir Models

The group of models described in this section is used to account for non-uniform room air temperatures that may occur within the interior air volume of a zone. These models are accessed using the RoomAirModelType input object. RoomAir modeling was added to EnergyPlus starting with Version 1.2. Although there are many types of analyses (comfort, indoor air quality, etc) that might benefit from localized modeling of how room air varies across space, only the *temperature* distribution of room air within the zone is currently addressed in EnergyPlus. This allows surface heat transfer and air system heat balance calculations to be made taking into account natural thermal stratification of air and different types of intentional air distribution designs such as under-floor and side-wall displacement ventilation that purport to extract room air at higher-than-mean temperatures. Note that EnergyPlus does **not** have completely general methods of modeling room air that are applicable to every conceivable type of airflow that might occur in a zone. Such models (e.g. RANS-CFD) are too computationally expensive to use with EnergyPlus for the foreseeable future. The models that are available in EnergyPlus offer only limited modeling capabilities for select room airflow configurations. Also note that because the complete mixing model for room air has long been the standard in building energy simulation, there is not currently a consensus on how to best model non-uniform air temperatures in buildings. Therefore, it is up to the user to have a good understanding of when, where, and how to apply the room air models available in EnergyPlus. The rest of this section provides some guidance in the way of examples and further discussion of the models available in EnergyPlus.

EnergyPlus offers the different types of air models listed in the table below along with the input objects associated with the use of that model.

Table 37. Summary of room air models available in EnergyPlus

Air model name	Applicability	Input Objects Required
Well-Mixed	All zones	None, default
User Defined	Any zone where the user has prior knowledge of the temperature pattern	'RoomAirModelType', 'RoomAir:TemperaturePattern:UserDefined', 'RoomAir:TemperaturePattern: xx'
One-Node Displacement Ventilation (Mundt)	displacement ventilation in typical office-type zones	'RoomAirModelType', 'RoomAirSettings:OneNodeDisplacementVentilation', 'RoomAir:Node'
Three-Node Displacement Ventilation (UCSD)	displacement ventilation	'RoomAirModelType', 'RoomAirSettings:ThreeNodeDisplacementVentilation'
Under-Floor Air Distribution Interior Model (UCSD)	Interior zones served by a UFAD system	'RoomAirModelType', 'RoomAirSettings:UnderFloorAirDistributionInterior'
Under-Floor Air Distribution Exterior Model (UCSD)	Exterior zones served by a UFAD system	'RoomAirModelType', 'RoomAirSettings:UnderFloorAirDistributionExterior'
UCSD Cross Ventilation	cross ventilation	'RoomAirModelType', 'RoomAirSettings:CrossVentilation'

The room air models are coupled to the heat balance routines using the framework described by Griffith and Chen (2004). Their framework was modified to include features needed for a comprehensive program for annual energy modeling rather than one for hourly load calculations. The formulation is largely shifted from being based on the setpoint temperature to one based on the current mean air temperature. This is necessary to allow for floating temperatures and dual setpoint control where there may be times that the mean zone temperatures are inside the dead band. The coupling framework was also extended to allow for exhaust air flows (e.g. bathroom exhaust fans) in addition to air system return flows.

The inside face temperature calculation is modified by rewriting the zone air temperature, T_a , with an additional subscript, i , for the surface index ($T_{a_j} \rightarrow T_{a_{i,j}}$ or $T_a \rightarrow T_{a_i}$). The inside face heat balance is solved for its surface temperature using,

$$T_{s_{i,j}} = \frac{T_{s_{o_{i,j}}} Y_{i,o} + \sum_{k=1}^{nz} T_{s_{o_{i,j-k}}} Y_{i,k} - \sum_{k=1}^{nz} T_{s_{i,j-k}} Z_{i,k} + \sum_{k=1}^{nq} \Phi_{i,k} q''_{ki,j-k} + T_{a_{i,j}} h_{c_{i,j}} + q''_{LWS} + q''_{LWX} + q''_{SW} + q''_{sol}}{Z_{i,o} + h_{c_{i,j}}}$$

- where,
- T_s is the inside face temperature
 - i subscript indicates individual surfaces
 - j subscript indicates current time step
 - k subscript indicates time history steps
 - T_{so} is the outside face temperature
 - Y_i are the cross CTF coefficients
 - Z_i are the inside CTF coefficients
 - Φ_i are the flux CTF coefficients
 - q''_{ki} is the conduction heat flux through the surface
 - h_{c_i} is the surface convection heat transfer coefficient
 - T_a is the near-surface air temperature
 - q''_{LWS} is the long wave radiation heat flux from equipment in zone
 - q''_{LWX} is the net long wavelength radiation flux exchange between zone surfaces
 - q''_{SW} is the net short wavelength radiation flux to surface from lights
 - q''_{sol} is the absorbed direct and diffuse solar (short wavelength) radiation

References:

Griffith, B. and Q. Chen. 2004. Framework for coupling room air models to heat balance load and energy calculations (RP-1222). International Journal of Heating, Ventilating, Air-conditioning and Refrigerating Research. ASHRAE, Atlanta GA. Vol 10. No 2. April 2004.

User Defined RoomAir Temperatures

The input object RoomAir:TemperaturePattern:UserDefined provides a capability for users to define the sort of air temperature pattern he or she expects in the zone. With these models, the pattern is generally set beforehand and does not respond to conditions that evolve during the simulation. (Exception: the pattern available through the RoomAir:TemperaturePattern:TwoGradient object will switch between two different pre-defined vertical gradients depending on the current value of certain temperatures or thermal loads.)

The user-defined patterns obtain the mean air temperature, T_{MAT} , from the heat balance domain and then produce modified values for:

T_{ai} the adjacent air temperature which is then used in the calculation of inside face surface temperature during the heat balance calculations,

$T_{leaving}$ the temperature of air leaving the zone and entering the air system returns

$T_{exhaust}$ the temperature of air leaving the zone and entering the exhaust.

T_{stat} the temperature of air “sensed” at the thermostat (not currently used in air system control because air system flows use load-based control).

The user defined room air models used indirect coupling so that the patterns provide values for, or ways to calculate, how specific temperatures differ from T_{MAT} . The various ΔT values determined from the model are applied to T_{MAT} as follows:

$$T_{ai} = \Delta T_{ai} + T_{MAT}$$

(where “i’s” represents each surface in the zone that is affected by the model)

$$T_{leaving} = \Delta T_{return} + T_{MAT}$$

$$T_{exhaust} = \Delta T_{exhaust} + T_{MAT}$$

$$T_{stat} = \Delta T_{stat} + T_{MAT}$$

The patterns defined by the object ‘RoomAir:TemperaturePattern:SurfaceMapping’ are fairly straightforward. The user directly inputs values for ΔT_{ai} for each surface. The pattern “maps” specific surfaces, identified by name, to ΔT_{ai} values. This provides completely general control (but in practice may be cumbersome to use). The other patterns focus on temperature changes in the vertical direction. Surfaces do not need to be identified, but all the surfaces with the same height will be assigned the same ΔT_{ai} values.

The patterns defined by the object ‘RoomAir:TemperaturePattern:NondimensionalHeight’ apply a temperature profile based on a non-dimensionalized height, Z . The height of each surface is defined to be the z-coordinate of the surface’s centroid relative to the average z-coordinate of the floor surfaces. The zone ceiling height is used as the length scale to non-dimensionalize each surface’s height so that,

$$Z_i = (z_{i,centroid} - z_{floorAvg}) / \text{Ceiling Height}$$

(where “i’s” represents each surface in the zone that is affected by the model)

The values for Z_i are constrained to be between 0.01 and 0.99 because the value is meant to describe the air layer near the surface (say approximate 0.1 m from the surface) rather than the surface itself.

The user-defined profile is treated as a look up table or piecewise linear model. The values for ΔT_{ai} are determined by searching the Z values in the user-defined profile and performing linear interpolation on the associated ΔT_a values.

The patterns defined by the object 'RoomAir:TemperaturePattern:ConstantGradient' apply a constant temperature gradient in the vertical direction. The model assumes that T_{MAT} occurs at the mid-plane so that $Z_{T_{MAT}} = 0.5$ (by definition). The surface Z_i values are compared to $Z_{T_{MAT}}$ and then scaled with zone ceiling height to obtain values for the change in height (in units of meters), Δz . The user defined gradient, $grad$, (units of °C/m) is then used to determine ΔT_{ai} values using

$$\Delta T_{ai} = \Delta z * grad$$

The patterns defined by the object 'RoomAir:TemperaturePattern:TwoGradient' are very similar to the constant gradient pattern above but the value of $grad$ used at any given time is selected by interpolating between two user-defined values for $grad$. Five options are available, three based on temperatures and two based on thermal loads – see the Input Output Reference. The user provides upper and lower bounding values. If the current value of the “sensing” variable lies between the upper and lower bounds, then $grad$ is determined using linear interpolation. If the designated value is above the upper bound then the upper value for $grad$ is used (no extrapolation). Similarly, if the designated value is below the lower bound, then the lower value for $grad$ is used. Note that “upper” and “lower” indicate the temperature and heat rate bounds and that the values for $grad$ do not have to follow in the same way; the $grad$ value for the lower bound could be higher than the $grad$ value for the upper bound (providing a something of a reverse control scheme). Rather than directly using ΔT values from the user, the temperatures for the return air, exhaust and thermostat are determined based on user-entered heights (in units of meters from the floor) and applying the current value for $grad$.

One-Node Displacement Ventilation RoomAir Model

The input object RoomAirSettings:OneNodeDisplacementVentilation provides a simple model for displacement ventilation. Mundt (1996) points out that a floor air heat balance provides a simple and reasonably accurate method of modeling the temperature near the floor surface. The slope of a linear temperature gradient can then be obtained by adding a second upper air temperature value that comes from the usual overall air system cooling load heat balance. The figure below diagrams the temperature distribution versus height being calculated by the model. Mundt's floor air heat balance is extended to include convection heat gain from equipment and by ventilation or infiltration that may be introduced near the floor in order to maintain all the terms in the air heat balance of the Heat Balance Model. This yields the following heat balance for a floor air node,

$$\rho c_p \dot{V} (T_{AirFloor} - T_{Supply}) = h_{cFloor} A_{Floor} (T_{Floor} - T_{AirFloor}) + Q_{ConvSourceFloor} + Q_{InfilFloor}$$

where

ρ is the air density

c_p is the air specific heat at constant pressure

\dot{V} is the air system flow rate

T_{supply} is the air system's supply air drybulb temperature

h_{cFloor} is the convection heat transfer coefficient for the floor

A_{floor} is the surface area of the floor

T_{floor} is the surface temperature of the floor

$\dot{Q}_{convSourceFloor}$ is the convection from internal sources near the floor (< 0.2 m)

$\dot{Q}_{InfilFloor}$ is the heat gain (or loss) from infiltration or ventilation near the floor

“Floor splits” are the fraction of total convective or infiltration loads that are dispersed so as to add heat to the air located near the floor. The user prescribes values for floor splits as input. No guidance is known to be available to use in recommending floor splits, but the user could for example account for equipment known to be near the floor, such as tower computer cases, or supplementary ventilation designed to enter along the floor. The equation above can be solved directly for $T_{AirFloor}$ and is used in the form of the equation below,

$$T_{AirFloor} = \frac{\rho c_p \dot{V} T_{Supply} + \sum h_{cFloor} A_{Floor} T_{Floor} + \dot{Q}_{ConvSourceFloor} + \dot{Q}_{InfilFloor}}{\rho c_p \dot{V} + \sum h_{cFloor} A_{Floor}}$$

The upper air node temperature is obtained by solving the overall air heat balance for the entire thermal zone for the temperature of the air leaving the zone and going into the air system return, $T_{leaving}$.

$$T_{Leaving} = \frac{-\dot{Q}_{sys}}{\rho c_p \dot{V}} + T_{Supply} \quad (210)$$

where \dot{Q}_{sys} is the air system heat load with negative values indicating a positive cooling load.

Values for \dot{Q}_{sys} are computed by the load calculation routines and passed to the air model.

The vertical temperature gradient or slope, dT/dz , is obtained from,

$$\frac{dT}{dz} = \frac{T_{Leaving} - T_{AirFloor}}{H_{return}}$$

where H_{return} is the distance between the air system return and the floor air node assumed to be 0.1 m from the floor and z is the vertical distance.

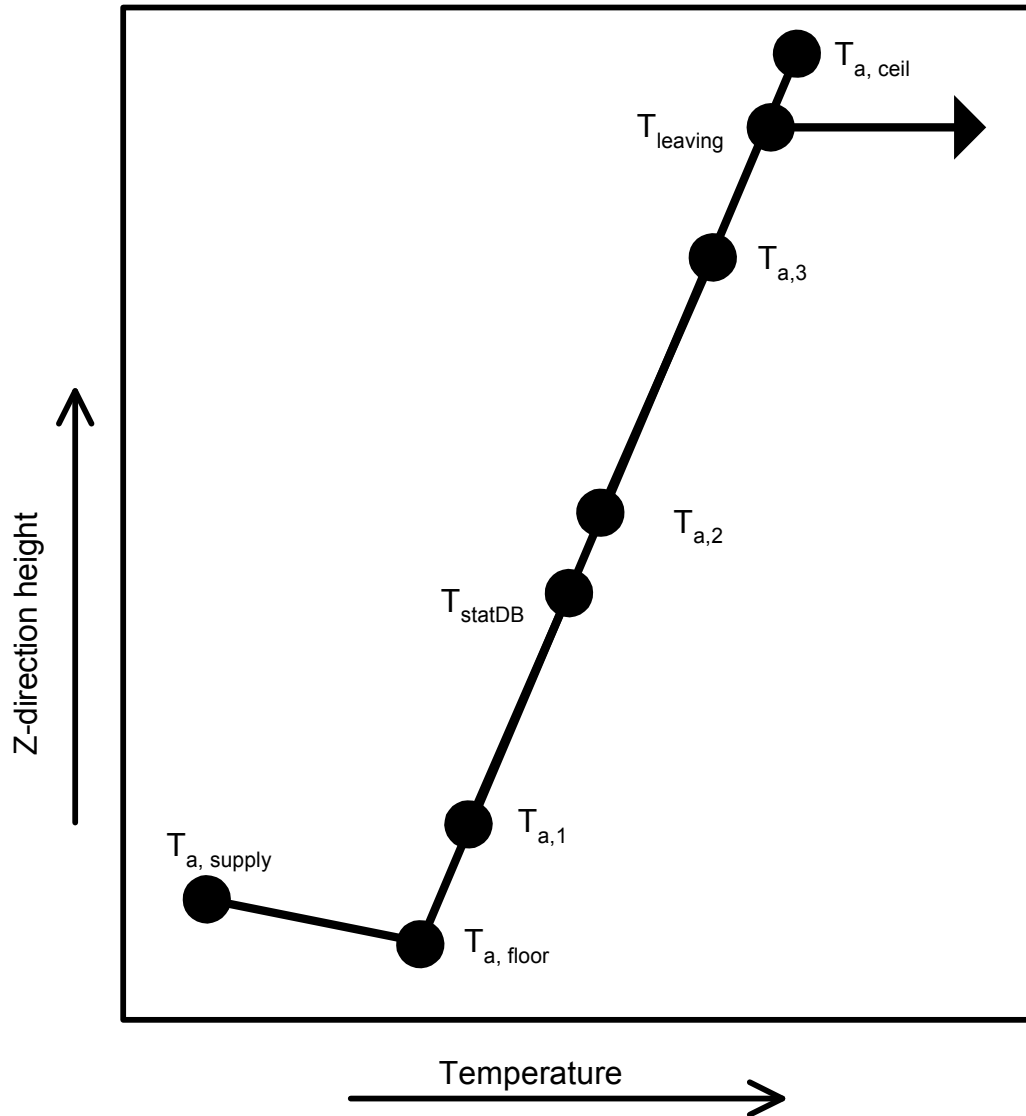


Figure 106. Height versus temperature schematic for Mundt model

The constant slope allows obtaining temperatures at any vertical location using,

$$T_{a_i} = T_{leaving} - \frac{dT}{dz}(z_{leaving} - z_i)$$

So for example the temperatures near the ceiling can easily be determined. Accounting for the location of the thermostat inside the zone (e.g. 1.1 m) is accomplished by returning the temperature for the appropriate height to the appropriate air node used for control. If the walls are subdivided in the vertical direction as shown in the figure above, then the air model can provide individual values for each surface based on the height and slope. However, no additional heat balances are necessarily made (in the air domain) at these points as all the surface convection is passed to the model in the totaled value for \dot{Q}_{sys} .

References

Mundt, E. 1996. The performance of displacement ventilation systems-experimental and theoretical studies, Ph. D. Thesis, Royal Institute of Technology, Stockholm.

Three-Node Displacement Ventilation RoomAir Model

Overview

The input object RoomAirSettings:ThreeNodeDisplacementVentilation provides a simple model for heat transfer and vertical temperature profile prediction in displacement ventilation. The fully-mixed room air approximation that is currently used in most whole building analysis tools is extended to a three node approach, with the purpose of obtaining a first order precision model for vertical temperature profiles in displacement ventilation systems. The use of three nodes allows for greatly improved prediction of thermal comfort and overall building energy performance in low energy cooling strategies that make use of unmixed stratified ventilation flows.

The UCSD Displacement Ventilation Model is one of the non-uniform zone models provided through the Room Air Manager in EnergyPlus. The intent is to provide a selection of useful non-uniform zone air models to enable the evaluation of air-conditioning techniques that use stratified or partially stratified room air. Such techniques include displacement ventilation (DV) and underfloor air distribution (UFAD) systems. The methodology can also include, in principle, natural displacement ventilation and also wind-driven cross-ventilation (CV).

Displacement Ventilation

A DV system is a complete contrast to a conventional forced air system. In a conventional system conditioned air is delivered at ceiling level and the intent is to create a fully mixed space with uniform conditions. In a DV system conditioned air is delivered at floor level and low velocity in order to minimize mixing and to establish a vertical temperature gradient. The incoming air “displaces” the air above it which, in turn, is exhausted through ceiling level vents. In DV a noticeable interface occurs between the occupied zone of the room and a mixed hot layer near the ceiling of the room (Dominique & Guitton, 1997). Maintaining the lower boundary of this warm layer above the occupied zone is one of the many unique challenges of displacement ventilation design. Often DV systems use 100% outside air. The vertical displacement air movement means that convective heat gains introduced near the ceiling will be removed without affecting the occupied region of the room. Also a fraction of the heat gains that occur in the occupied zones rise as plumes into the upper part of the space, thereby reducing the cooling load. Similarly the fresh air will be used more effectively than with a fully mixed system: the fresh air won’t be “wasted” in the upper, unoccupied region of the room. Finally, the vertical temperature gradient means that the average room temperature can be higher for a DV conditioned room than with a conventionally conditioned room: the occupants feel the lower temperature in the lower region of the room and are unaffected by the higher temperature near the ceiling. However, whenever the outside air temperature is above $\approx 19^{\circ}\text{C}$ this advantage is mostly lost: the internal loads must be removed from the space independently of the airflow pattern (during the warmer hours buildings tend to be almost closed to the outside, operating in closed loop). The inflow temperature advantage is then only useful for the minimum outside air that must always be provided (in most cases this remaining advantage is negligible).

DV systems have limitations. In order to avoid chilling the occupants the supply air temperature used for DV is considerably higher than that used in conventional forced-air systems. This can lead to problems in removing both sensible and latent loads. Exterior spaces may have conditions that are not conducive to establishing a vertical temperature gradient. DV systems seem to be best suited to interior spaces with only moderate loads.

Non-uniform zone models

Several types of models have been proposed as suitable for inclusion in building energy simulation (BES) programs. These models must be simple enough not to impose an undue

computational burden on a BES program, yet provide enough predictive capability to produce useful comparisons between conventional and stratified zone operation strategies. ASHRAE RP-1222 (Chen & Griffith 2002) divides the candidate models into two categories: *nodal* and *zonal*. Nodal models describe the zone air as a network of nodes connected by flow paths; each node couples convectively to one or more surfaces. Zonal models are coarse-grained finite volume models. ASHRAE RP-1222 provides a short history (and examples) of each type of model. In terms of nodal models for displacement ventilation we mention the Mundt model (Mundt 1996), since it is implemented in EnergyPlus, and the Rees-Haves model (Rees & Haves 2001) since it is a well developed nodal-type model and is implemented in the RP-1222 toolkit. The Rees-Haves model, while successful in predicting the flow and temperature field for geometries similar to those used in its development, can suffer from lack of flexibility and clarity in the modeling approximations. When dealing with diverse geometries it is not clear that the flow coefficients used in the model are applicable or why they can be used since plumes, the fundamental driving mechanisms of the displacement flow, are not explicitly modeled. This is the main difference between the DV models implemented in the RP-1222 toolkit and the model that is described here.

The UCSD DV model is closer to a nodal model than to a zonal model. However, it is best to classify it in a separate category: plume equation based multi-layer models (Linden *et al.* 1990, Morton *et al.* 1956). These models assume that the dominant mechanism is plume-driven flow from discrete internal sources and that other effects (such as buoyancy driven flow at walls or windows) may be neglected. Alternatively, these heat sources also produce plumes that can be included in the model. The result is a zone divided vertically into two or more well separated regions – each region characterized by a single temperature or temperature profile. This characterization allows the physics of the heat gains and the ventilation flow to be represented in a realistic manner, without the introduction of *ad hoc* assumptions.

Model Description

Single Plume Two Layer Model

The simplest form of the plume equation based models is the case of a single plume in an adiabatic box with constant supply air flow. For this configuration two layers form in the room: a lower layer with similar density and temperature as the inflow air and a mixed upper layer with the same density / temperature as the outflow air. The main assumption of this model, successfully validated against scaled model experiments (Linden *et al.* 1990), is that the interface between the two layers occurs at the height (h) where the vertical buoyancy driven plume flow rate is the same as the inflow rate. For a point source of buoyancy in a non-stratified environment (a plume) the airflow rate increases with vertical distance from the source according to:

$$\dot{V} = C \cdot B^{1/3} z^{5/3} \quad (211)$$

where

\dot{V} = plume volume flux [m³/s]

B = buoyancy flux [m⁴/s³]

z = vertical distance above source [m]

$$C = \frac{6}{5} \alpha \left(\frac{9}{10} \alpha \right)^{1/3} \pi^{2/3}$$

α = plume entrainment constant; a value of 0.127 is used, suitable for top-hat profiles for density and velocity across the plumes.

For an ideal gas

$$\Delta\rho/\rho = \Delta T/T \quad (212)$$

resulting in the following relation between heat input rate and buoyancy flux:

$$B = \frac{g\dot{Q}}{\rho C_p T} \quad (213)$$

where

ρ = density of air [kg/m³]

T = air temperature [K]

g = acceleration of gravity [m/s²]

\dot{Q} = heat input rate [W]

C_p = specific heat capacity of air [J/kgK]

Since the plume volume flow rate increases with height with exponent 5/3, for any room inflow rate (F , (m³/s)) there will always be a height (h , (m)) where the plume driven flow rate matches the inflow rate. This height is obtained by setting (1.1) equal to F and solving for $z=h$:

$$h = F^{3/5} / (C^{3/5} B^{1/5}) \quad (214)$$

Substituting (213) in (214) and introducing air properties at 20 C gives:

$$h = 24.55 \cdot F^{3/5} / \dot{Q}^{1/5} \quad (215)$$

Multiple Plumes and Wall Heat Transfer

Of course, it would be rare for a real world case to consist of a single point-source plume originating on the floor, unaffected by heat gains from walls and windows. For multiple plumes of equal strength a straight-forward extension of the single is possible. N plumes of unequal strength result in the formation of n vertical layers. This case is much more complex but if we are satisfied with a first order precision model the equal strength model can be used by averaging the plume strengths (Carrilho da Graça, 2003). Even in a case where all plumes are of equal strength, nearby plumes may coalesce. Plumes that are less than 0.5 meters apart at their source will coalesce within 2 meters (Kaye & Linden, 2004).

As the complexity of the physical systems modeled increases some limitations must be imposed. In particular, the biggest challenge remains the interaction between wall driven boundary layers (positively and negatively buoyant) and displacement flows. For this reason, the model that is developed below is not applicable when:

1. Downward moving buoyancy driven airflow rate is of the same order of magnitude as plume driven flow (these airflow currents are typically generated on lateral surfaces or in the ceiling whenever these surfaces are much cooler than the room air).
2. Upward moving wall or floor generated buoyancy flux in the lower layer is of the same order of magnitude as plume driven flow.

Although these limitations are significant it is important to note that even in the presence of dominant convection from the floor surface, a buoyancy, two layer flow can be established whenever the plume buoyancy flux is more than 1/7 of the horizontal flux (Hunt *et al.* 2002). A two layer structure can also originate when the only heat source is a heated portion of the room floor, as long as the heated area does not exceed 15% of the room floor (Holford *et al.* 2002).

For the case of multiple non-coalescing plumes (n), with equal strength, the total vertical airflow for a given height is:

$$\dot{V} = CnB^{1/3}z^{5/3} \quad (216)$$

resulting in a mixed layer height of:

$$h = \frac{24.55F^{3/5}}{(n^3\dot{Q})^{1/5}} \quad (217)$$

Implementation

The model predicts three temperatures that characterize the three main levels in the stratification of the room:

1. a floor level temperature T_{floor} to account for the heat transfer from the floor into the supply air
2. an occupied subzone temperature T_{oc} representing the temperature of the occupied region;
3. an upper level temperature T_{mx} representing the temperature of the upper, mixed region and the outflow temperature.

We assume that the model for multiple, equal strength plumes (equations (216) and (217)) will be adequate for our calculations. The supply air flow rate \dot{V} is obtained by summing all the air flows entering the zone: supply air, infiltration, ventilation, and inter-zone flow. The heat gain \dot{Q} is estimated by summing all the convective internal gains located in the occupied subzone – task lights, people, equipment – and dividing this power equally among the n plumes. With these assumptions we can describe the implementation.

The UCSD DV model is controlled by the subroutine *ManageUCSDDVModel* which is called from the *RoomAirModelManager*. The *RoomAirModelManager* selects which zone model will be used for each zone.

The calculation is done in subroutine *CalcUCSDDV*. First we calculate the convective heat gain going into the upper and lower regions.

$$\begin{aligned} \dot{Q}_{\text{ocz}} = & \dot{Q}_{\text{oc,conv}} + \dot{Q}_{\text{tl,conv}} + \dot{Q}_{\text{eleq,conv}} + \dot{Q}_{\text{gaseq,conv}} + \dot{Q}_{\text{otheq,conv}} + \dot{Q}_{\text{hw,conv}} \\ & + \dot{Q}_{\text{stmeq,conv}} + \dot{Q}_{\text{bb,conv}} \end{aligned}$$

$$\dot{Q}_{\text{mxz}} = \dot{Q}_{\text{gl,conv}} + \dot{Q}_{\text{ltpl}} + \dot{Q}_{\text{htrad,conv}}$$

$$\dot{Q}_{\text{tot,conv}} = \dot{Q}_{\text{ocz}} + \dot{Q}_{\text{mxz}}$$

Next we sum up the inlet air flows in the form of MCP (mass flow rate times the air specific heat capacity) and MCPT (mass flow rate times C_p times air temperature).

$$MCP_{\text{zone}} = MCP_i + MCP_{\text{vent}} + MCP_{\text{mix}}$$

$$MCP_{\text{sys}} = \sum_{\text{inlets}} \dot{m}_i C_{p,i}$$

$$MCP_{tot} = MCP_{zone} + MCP_{sys}$$

$$MCPT_{zone} = MCPT_i + MCPT_{vent} + MCPT_{mix}$$

$$MCPT_{sys} = \sum_{inlets} \dot{m}_i C_{p,i} T_i$$

$$MCPT_{tot} = MCPT_{zone} + MCPT_{sys}$$

The number of plumes per occupant $N_{plumesperspers}$ is a user input. The total number of plumes in the zone is:

$$N_{plumes} = N_{occ} \cdot N_{plumespersperson}$$

The gains fraction Fr_{gains} is a user input via a schedule. It is the fraction of the convective gains in the occupied subzone that remain in that subzone. Using this we calculate the total power in the plumes and the power per plume.

$$\dot{Q}_{plumes} = (1 - Fr_{gains}) \cdot \dot{Q}_{tot,conv}$$

$$\dot{Q}_{perplume} = \dot{Q}_{plumes} / N_{plumes}$$

We now make an initial estimate of the height fraction Fr_{hb} (height of the boundary layer divided by the total zone height).

$$Fr_{hb} = (24.55/H_{ceiling}) \left(\frac{0.000833 \cdot MCP_{tot}}{N_{plumes} \cdot \dot{Q}_{perplume}^{1/3}} \right)^{3/5} \quad (218)$$

where $0.000833 = 1/(\rho_{air} \cdot c_{p,air})$ converts MCP_{tot} to a volumetric flow rate. Next we iterate over the following 3 steps.

Iterative procedure

1. Call subroutine *HcUCSDDV* to calculate a convective heat transfer coefficient for each surface in the zone, an effective air temperature for each surface, and HA_{mx} , HAT_{mx} , HA_{oc} , HAT_{oc} , HA_{fl} , and HAT_{fl} . Here HA is $\sum_{surfaces} h_{c,i} \cdot A_i$ for a region and HAT is

$$\sum_{surfaces} h_{c,i} \cdot A_i \cdot T_i \text{ for a region. The sum is over all the surfaces bounding the region; } h_{c,i}$$

is the convective heat transfer coefficient for surface i , A_i is the area of surface i , and T_i is the surface temperature of surface i .

2. Recalculate Fr_{hb} using the equation (218).
3. Calculate the three subzone temperatures: T_{floor} , T_{oc} and T_{mx} .

The h_c 's calculated in step 1 depend on the subzone temperatures and the boundary layer height. In turn the subzone temperatures depend on the HA and HAT's calculated in step 1. Hence the need for iteration

Next we describe each steps 1 and 3 in more detail.

Step 1

Subroutine *HcUCSDDV* is quite straightforward. It loops through all the surfaces in each zone and decides whether the surface is located in the upper, mixed subzone or the lower, occupied subzone, or if the surface is in both subzones. If entirely in one subzone the subzone temperature is stored in the surface effective temperature variable *TempEffBulkAir(SurfNum)* and h_c for the surface is calculated by a call to subroutine *CalcDetailedHcInForDVModel*. This routine uses the "detailed" natural convection coefficient calculation that depends on surface tilt and $\Delta T^{1/3}$. This calculation is appropriate for situations with low air velocity.

For surfaces that bound 2 subzones, the subroutine calculates h_c for each subzone and then averages them, weighting by the amount of surface in each subzone.

During the surface loop, once the h_c for a surface is calculated, the appropriate subzone HA and HAT sums are incremented. If a surface is in 2 subzones the HA and HAT for each subzone are incremented based on the area of the surface in each subzone.

Step 3

The calculation of subzone temperatures follows the method used in the **ZoneTempPredictorCorrector** module and described in the section **Basis for the System and Zone Integration**. Namely a third order finite difference expansion of the temperature time derivative is used in updating the subzone temperatures. Otherwise the subzone temperatures are obtained straightforwardly by solving an energy balance equation for each subzone.

$$T_{fl} = (C_{air,fl} \cdot (3 \cdot T_{-1,fl} - (3/2) \cdot T_{-2,fl} + (1/3) \cdot T_{-3,fl}) + HAT_{fl} + MCPT_{tot}) / ((11/6) \cdot C_{air,fl} + HA_{fl} + MCP_{tot})$$

$$T_{oc} = (C_{air,oc} \cdot (3 \cdot T_{-1,oc} - (3/2) \cdot T_{-2,oc} + (1/3) \cdot T_{-3,oc}) + \dot{Q}_{ocz} \cdot Fr_{gains} + HAT_{oc} + T_{fl} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,oc} + HA_{oc} + MCP_{tot})$$

$$T_{mx} = (C_{air,mx} \cdot (3 \cdot T_{-1,mx} - (3/2) \cdot T_{-2,mx} + (1/3) \cdot T_{-3,mx}) + \dot{Q}_{ocz} \cdot (1 - Fr_{gains}) + \dot{Q}_{mxz} + HAT_{mx} + T_{oc} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,mx} + HA_{mx} + MCP_{tot})$$

Here $C_{air,fl}$, $C_{air,oc}$, and $C_{air,mx}$ are the heat capacities of the air volume in each subzone.

$C_{air,mx}$ is calculated by

$$R_{air,mx} = V_{mx} \cdot (\Delta z_{mx} / z_{ceil}) \cdot \rho_{air,mx} \cdot c_{p,air,mx} \cdot Mul_{cap} / (\Delta t_z \cdot 3600)$$

$$C_{air,mx} = R_{air,mx} \cdot \Delta t_z / \Delta t_{sys}$$

The other subzone air heat capacities are calculated in the same manner.

Mixed calculation

The above iterative procedure assumed that displacement ventilation was taking place: i.e., conditions were favorable temperature stratification in the zone. Now that this calculation is complete and the subzone temperatures and depths calculated, we check to see if this assumption was justified. If not, zone conditions must be recalculated assuming a well-mixed zone.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < H_{fl,top} + \Delta z_{occ,min}$ then the following mixed calculation will replace the displacement ventilation calculation.

Note: $\Delta z_{occ,min}$ is the minimum thickness of occupied subzone. It is set to 0.2 meters. $H_{fl,top}$ is the height of the top of the floor subzone. It is defined to be 0.2 meters; that is, the floor subzone is always 0.2 meters thick and T_{fl} is the temperature at 0.1 meter above the floor surface.

The mixed calculation iteratively calculates surface convection coefficients and room temperature just like the displacement ventilation calculation described above. In the mixed case however, only one zone temperature T_{avg} is calculated. The 3 subzone temperatures are then set equal to T_{avg} .

First, Fr_{hb} is set equal to zero.

Then the code iterates over these steps.

1. Calculate T_{avg} using

$$T_{avg} = (C_{air,z} \cdot (3 \cdot T_{-1,z} - (3/2) \cdot T_{-2,z} + (1/3) \cdot T_{-3,z}) + \dot{Q}_{tot,conv} + HAT_{oc} + HAT_{mx} + HAT_{fl} + MCPT_{tot}) / ((11/6) \cdot C_{air,z} + HA_{oc} + HA_{mx} + HA_{fl} + MCP_{tot})$$

$$T_{mx} = T_{avg}$$

$$T_{oc} = T_{avg}$$

$$T_{fl} = T_{avg}$$

2. Call $HcUCSDDV$ to calculate the h_c 's.
3. Repeat step 1

Final calculations

The displacement ventilation calculation finishes by calculating some report variables. Using equation (218), setting the boundary height to 1.5 meters and solving for the flow, we calculate a minimum flow fraction:

$$\dot{V}_{min} = (1.5 / 24.55)^{5/3} \cdot N_{plumes} \cdot \dot{Q}_{perplume}^{1/3}$$

$$Fr_{min,flow} = .000833 \cdot MCP_{tot} / \dot{V}_{min}$$

We define heights:

$$H_{trans} = Fr_{hb} \cdot H_{ceil}$$

$$H_{mxavg} = (H_{ceil} + H_{trans}) / 2$$

$$H_{ocavg} = (H_{fltop} + H_{trans}) / 2$$

$$H_{flavg} = H_{fltop} / 2$$

Using the user defined comfort height we calculate the comfort temperature.

If mixing:

$$T_{comf} = T_{avg}$$

If displacement ventilation:

If $H_{comf} < H_{flavg}$

$$T_{comf} = T_{fl}$$

Else if $H_{comf} \geq H_{flavg}$ and $H_{comf} < H_{ocavg}$

$$T_{comf} = (T_{fl}(H_{ocavg} - H_{comf}) + T_{mx}(H_{comf} - H_{flavg})) / (H_{ocavg} - H_{flavg})$$

Else if $H_{comf} \geq H_{ocavg}$ and $H_{comf} < H_{mxavg}$

$$T_{comf} = (T_{oc}(H_{mxavg} - H_{comf}) + T_{mx}(H_{comf} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

Else if $H_{comf} \geq H_{mxavg}$ and $H_{comf} < H_{ceil}$

$$T_{comf} = T_{mx}$$

Using the user defined thermostat height we calculate the temperature at the thermostat.

If mixing:

$$T_{stat} = T_{avg}$$

If displacement ventilation:

If $H_{stat} < H_{flavg}$

$$T_{stat} = T_{fl}$$

Else if $H_{stat} \geq H_{flavg}$ and $H_{stat} < H_{ocavg}$

$$T_{stat} = (T_{fl}(H_{ocavg} - H_{stat}) + T_{mx}(H_{stat} - H_{flavg})) / (H_{ocavg} - H_{flavg})$$

Else if $H_{stat} \geq H_{ocavg}$ and $H_{stat} < H_{mxavg}$

$$T_{stat} = (T_{oc}(H_{mxavg} - H_{stat}) + T_{mx}(H_{stat} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

Else if $H_{stat} \geq H_{mxavg}$ and $H_{stat} < H_{ceil}$

$$T_{stat} = T_{mx}$$

The average temperature gradient is:

If $H_{mxavg} - H_{flavg} > 0.1$

$$GradT_{avg} = (T_{mx} - T_{fl}) / (H_{mxavg} - H_{flavg})$$

else $GradT_{avg} = -9.999$

The maximum temperature gradient is:

If $H_{ocavg} - H_{flavg} > 0.1$

$$GradT_{max,1} = (T_{oc} - T_{fl}) / (H_{ocavg} - H_{flavg})$$

else $GradT_{max,1} = -9.999$

If $H_{mxavg} - H_{ocavg} > 0.1$

$$GradT_{max,2} = (T_{mx} - T_{oc}) / (H_{mxavg} - H_{ocavg})$$

else $GradT_{max,2} = -9.999$

and

$$GradT_{max} = \max(GradT_{max,1}, GradT_{max,2})$$

For reporting purposes, if the zone is deemed to be mixed, the displacement ventilation report variables are set to flag values.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < H_{fl,top} + \Delta z_{occ,min}$ or $T_{mx} - T_{oc} < \Delta T_{CritRep}$

$$GradT_{avg} = -9.999$$

$$GradT_{max} = -9.999$$

$$FR_{min,flow} = -1.0$$

$$H_{trans} = -9.999$$

Finally, the zone node temperature is set to T_{mx} .

References

- Carrilho da Graca, G. 2003. Simplified models for heat transfer in rooms. Ph. D. Thesis, University of California, San Diego.
- Chen, Q., and B. Griffith. 2002. Incorporating Nodal Room Air Models into Building Energy Calculation Procedures. ASHRAE RP-1222 Final Report.
- Cooper, P. and P.F. Linden. 1996. Natural ventilation of an enclosure containing two buoyancy sources. Journal of Fluid Mechanics, Vol. 311, pp. 153-176.
- Dominique, M. and P. Guitton. 1997. Validation of displacement ventilation simplified models. Proc. of Building Simulation.
- Holford, J.M., G.R. Hunt and P.F. Linden. 2002. Competition between heat sources in a ventilated space. Proceedings of RoomVent 2002, pp. 577-580.
- Hunt, G.R., J.M. Holford and P.F. Linden. 2002. Characterization of the flow driven by a finite area heat source in a ventilated enclosure. Proceedings of RoomVent 2002, pp. 581-584.
- Hunt, G.R. and P.F. Linden. 2001. Steady-state flows in an enclosure ventilated by buoyancy forces assisted by wind. Journal of Fluid Mechanics, Vol. 426, pp. 355-386.
- Kaye, K.N. and P.F. Linden. 2004. Coalescing axisymmetric turbulent plumes. Journal of Fluid Mechanics, Vol. 502, pp. 41-63.
- Linden, P.F., G.F. Lane-Serff and D.A. Smeed. 1990. Emptying filling boxes: the fluid mechanics of natural ventilation. Journal of Fluid Mechanics, Vol. 212, pp. 309-335.
- Linden, P.F. and P. Cooper. 1996. Multiple sources of buoyancy in a naturally ventilated enclosure. Journal of Fluid Mechanics, Vol. 311, pp. 177-192.
- Morton, B.R., G.I. Taylor and J.S. Turner. 1956. Turbulent gravitational convection from maintained and instantaneous sources. Proceedings of the Royal Society of London, Vol A234, pp. 1-23.
- Mundt, E. 1996. The Performance of Displacement Ventilation Systems – Experimental and Theoretical Studies, Ph. D. Thesis, Bulletin N38, Building Services Engineering KTH, Stockholm.
- Rees, S.J., and P. Haves. 2001. A nodal model for displacement ventilation and chilled ceiling systems in office spaces. Building and Environment, Vol. 26, pp. 753-762.

Under-Floor Air Distribution Interior Zone Model

Overview

The input object RoomAirSettings:UnderFloorAirDistributionInterior provides a simple model for heat transfer and nonuniform vertical temperature profile for interior zones of a UFAD system. These zones are expected to be dominated by internal loads, a portion of which (such as occupants and workstations) will act to generate plumes. The plumes act to potentially create room air stratification, depending on the type & number of diffusers, the amount and type of load, and the system flowrate. In order to better model this situation the fully-mixed room air approximation that is currently used in most whole building analysis tools is extended to a two node approach, with the purpose of obtaining a first order precision model for vertical temperature profiles for the interior zones of UFAD systems. The use of 2 nodes allows for greatly improved prediction of thermal comfort and overall building energy performance for the increasingly popular UFAD systems.

The UCSD UFAD Interior Zone Model is one of the non-uniform zone models provided through the Room Air Manager in EnergyPlus. The intent is to provide a selection of useful non-uniform zone air models to enable the evaluation of air-conditioning techniques that use stratified or partially stratified room air. Such techniques include displacement ventilation (DV) and underfloor air distribution (UFAD) systems. The methodology can also include natural displacement ventilation and also wind-driven cross-ventilation (CV).

Underfloor air distribution systems

UFAD systems represent, in terms of room air stratification, an intermediate condition between a well-mixed zone and displacement ventilation. Air is supplied through an underfloor plenum at low pressure through diffusers in the raised floor. The diffusers can be of various types: e.g., swirl, variable-area, displacement, and produce more or less mixing in the zone. UFAD systems are promoted as saving energy due to: higher supply air temperature; low static pressure; cooler conditions in the occupied subzone than in the upper subzone; and sweeping of some portion of the convective load (from ceiling lights, for instance) into the return air without interaction with the occupied region of the zone.

Modeling a UFAD system is quite complex and involves considerably more than just a non-uniform zone model. The zones' coupling to the supply and return plenums must be modeled accurately (particularly radiative transfer from a warm ceiling to a cool floor and on into the supply plenum by conduction). The supply plenum must be accurately modeled, giving a good estimate of the supply air temperature and conduction heat transfer between supply & return plenums through the slab. The HVAC system must be modeled including return air bypass and various types of fan powered terminal units.

The UCSD UFAD interior zone model is similar to the UCSD DV model. The most obvious difference is that the UFAD model has no separate near-floor subzone. Like the UCSD DV model it is a plume equation based multi-layer model (2 layers in this case). The zone is modeled as being divided into 2 well separated subzones which we denote as "occupied" and "upper". Each subzone is treated as having a single temperature. The boundary between the 2 subzones moves up & down each time step as a function of zone loads and supply air flow rate. Thus at each HVAC time step, the height of the boundary above the floor must be calculated, portions of surfaces assigned to each subzone, and a separate convective heat balance performed on each subzone.

Model Description

The UFAD interior zone model is based upon non-dimensional analysis of the system and using the non-dimensional description to make a comparison between full-scale UCB test chamber data & small-scale UCSD salt tank measurements.

In order to do the non-dimensional comparisons, we need to define two dimensionless parameters. One is Γ , and the other is ϕ . Lin & Linden (Lin & Linden, 2005) showed that in a UFAD system, the buoyancy flux of the heat source (B) and the momentum flux of the cooling jets (M) are the controlling parameters on the stratification. Since $[B] = L^4 T^{-3}$ and $[M] = L^4 T^{-2}$, we can have a length scale as $M^{3/4} / B^{1/2}$.

Definition of Γ for the single-plume single-diffuser basic model

We observed, in our small-scale experiments, that the total room height does not affect the interface position, or the height of the occupied zone. In other words, H might not be the critical length scale for the stratification. Therefore, we started to use \sqrt{A} as the reference length. Then Γ is defined as

$$\Gamma = \frac{M^{3/4}}{B^{1/2} A^{1/2}} = \frac{(Q^2 / A)^{3/4}}{B^{1/2} A^{1/2}} = \frac{Q^{3/2}}{A^{5/4} B^{1/2}} \quad (219)$$

Definition for multi-diffuser and multi-source cases

We only considered single-diffuser, single-source cases in above analysis. Suppose there are n equal diffusers and m equal heat sources in a UFAD room. We shall divide the number of diffusers up into a number of separate heat sources so that each subsection with $n'=n/m$ diffusers per heat source will have the same stratification as other subsections. Further, the air flow and the heat load into the subsection Q' and B' will

be $Q' = Q / m$ $B' = B / m$ respectively, where Q' and B' are the total air flow and the total heat load for the entire UFAD space. Then the momentum flux each diffuser per heat source carries is $M_d = (\frac{1}{n'} Q')^2 / A$. (219) will be modified as

$$\Gamma = \frac{(n' M_d)^{3/4}}{B'^{1/2} \sqrt{n' A}} = \frac{Q'^{3/2}}{(n' A)^{5/4} B'^{1/2}} = \frac{Q'^{3/2}}{m(n' A)^{5/4} B'^{1/2}} \quad (220)$$

Full-scale cases

Because B is the buoyancy flux of the heat sources and M is the momentum flux of the cooling jets, in a real full-scale room, we shall consider the total room net heat load (plume heat input, minus the room losses) and the total net flow rate coming from the diffusers (input room air flow, minus the room leakage). Further, if the diffuser is swirl type, the vertical momentum flux should be used.

$$\Gamma = \frac{(Q \cos \theta)^{3/2}}{m(n' A)^{5/4} (0.0281W)^{1/2}} \quad (221)$$

where, Q is the net flow rate coming out from all diffusers (m^3/s); W is the total net heat load (kW); A is the effective area of each diffuser (m^2); n' is the number of diffusers per heat source; θ is the angle between the diffuser slots and the vertical direction and m is the number of heat sources

Definition of Φ

In our theoretical model, two-layer stratification forms at steady state, provided that each diffuser carries the same momentum flux, and each heat source has the same heat load. We could define a dimensionless parameter Φ , which indicates the strength of stratification.

Small-scale cases

In our salt-water tank experiments, fluid density ρ is measured. Define that

$$\phi = \frac{\rho_u - \rho_l}{\rho_u - \rho_o} \quad (222)$$

where, ρ_u and ρ_l are the fluid density of the upper layer and lower layer, separately; and ρ_o is the source density at the diffusers.

Therefore, $\rho_l = \rho_o$ gives $\phi = 1$, which means the largest stratification (displacement ventilation case); $\rho_l = \rho_u$ leads to $\phi = 0$, in which case there is no stratification (mixed ventilation case).

Full-scale cases

Similarly, we can define ϕ for full-scale cases by using temperature.

$$\phi = \frac{T_r - T_{oz}}{T_r - T_s} \quad (223)$$

where T_r , T_{oz} , and T_s are the return air temperature, the occupied zone temperature and the supply temperature, respectively (K). Again $\phi = 1$ occurs in displacement ventilation; while $\phi = 0$ happens in mixed ventilation.

Comparisons between full-scale UCB data and small-scale UCSD data

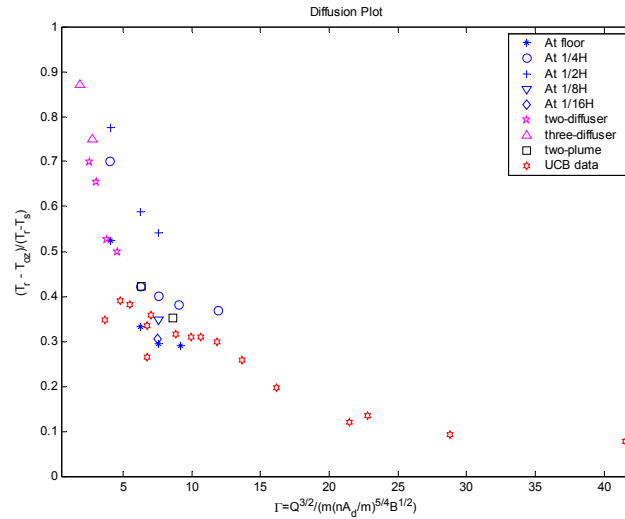


Figure 107. Data comparisons in the non-dimensional (a) regular $\Gamma-\phi$ plot

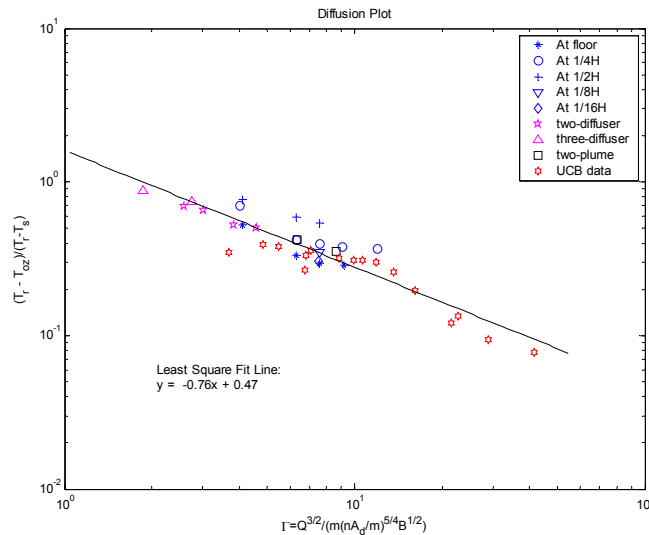


Figure 108. (b) log-log $\Gamma-\phi$ plot.

The figures (Figure 107. Data comparisons in the non-dimensional (a) regular $\Gamma-\phi$ plot and Figure 108. (b) log-log $\Gamma-\phi$ plot.) show the comparisons between UCB's data and the UCSD salt tank data in the $\Gamma-\phi$ plot. As seen in the figures, the full-scale and small-scale data are on the same trend curve. This provides the evidence that the salt-tank experiments have included most characteristics of a UFAD system. Note that big $\Gamma (>20)$ of UCB's experiments all have large DDR (from 1.19 to 2.18). The largest DDR (2.18) even gives a negative $(T_r - T_{oz}) / (T_r - T_s)$, which is NOT shown in the figures.)

We could work out the occupied zone temperature by using the least-square fitting line suggested in figure 1(b). Hence the interface height is needed to determine a entire two-layer stratification. Figure 109 shows the dimensionless interface height $(h / \sqrt{n' A})$ of the UCSD small-scale experiments plotted against Γ . Note that for the experiments with elevated heat

source, the interface heights have been modified by $h' = h - \frac{1}{2}h_s$, where h_s is the vertical position of the elevated heat source. All data then are located along a line in Figure 109. Since the salt-tank experiments are concluded to represent important characteristics of a full-scale UFAD room, this figure provides some guidelines for estimate the interface position in a real UFAD room.

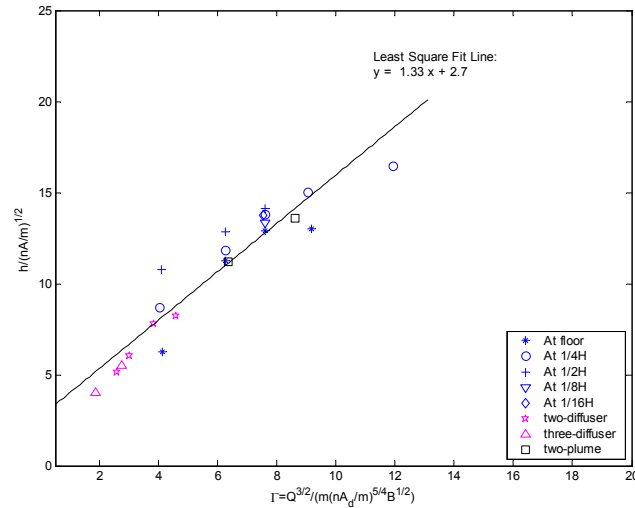


Figure 109. Non-dimensional interface height of small-scale experiments.

Formulas for EnergyPlus based on the dimensionless parameter Γ

If we have input including the supply temperature T_s (K); the number of diffusers n ; the number of heat sources m ; the vertical position of the heat sources h_s (m); the heat load W (kW); the effective area of a diffuser A (m²); and the total supply air flow rate Q (m³/s) then the output will be

$$T_r = \frac{0.0281W}{Qg} T_s + T_s \quad (224)$$

$$T_{oz} = T_r - 1.6\Gamma^{-0.76} (T_r - T_s) \quad (225)$$

$$h = \sqrt{\frac{n}{m}} A (7.43 \ln(\Gamma) - 1.35) + \frac{1}{2} h_s \quad (226)$$

where T_r is the return temperature (K); T_{oz} is the occupied subzone temperature (K); h is the interface height (m); and Γ is defined above.

Implementation

The implementation closely follows the procedure described in the displacement ventilation zone model. The model predicts two temperatures that characterize the two main levels in the stratification of the room:

1. an occupied subzone temperature T_{oc} representing the temperature of the occupied region;
2. an upper level temperature T_{mx} representing the temperature of the upper, mixed region and the outflow temperature.

We will use (226) to calculate the interface height and do a heat balance calculation on each subzone. Γ is given by (221). The supply air flow rate \dot{V} is obtained by summing all the air flows entering the zone: supply air, infiltration, ventilation, and inter-zone flow. The heat gain \dot{Q} is estimated by summing all the convective internal gains located in the occupied subzone – task lights, people, equipment – and dividing this power equally among the n plumes. With these assumptions we can describe the implementation.

The UCSD UFI model is controlled by the subroutine *ManageUCSDUFModels* which is called from the *RoomAirModelManager*. The *RoomAirModelManager* selects which zone model will be used for each zone.

The calculation is done in subroutine *CalcUCSDUI*. First we calculate the convective heat gain going into the upper and lower regions.

$$\begin{aligned}\dot{Q}_{ocz} &= \dot{Q}_{oc,conv} + \dot{Q}_{tl,conv} + \dot{Q}_{eleg,conv} + \dot{Q}_{gaseq,conv} + \dot{Q}_{otheq,conv} + \dot{Q}_{hw,conv} \\ &\quad + \dot{Q}_{stmeq,conv} + \dot{Q}_{bb,conv} + \dot{Q}_{waterheater,conv} + \dot{Q}_{fuelcell,conv} \\ \dot{Q}_{mxz} &= \dot{Q}_{gl,conv} + \dot{Q}_{ltp} + \dot{Q}_{htrrad,conv} \\ \dot{Q}_{tot,conv} &= \dot{Q}_{ocz} + \dot{Q}_{mxz}\end{aligned}$$

Next we sum up the inlet air flows in the form of MCP (mass flow rate times the air specific heat capacity) and MCPT (mass flow rate times C_p times air temperature).

$$\begin{aligned}MCP_{zone} &= MCP_i + MCP_{vent} + MCP_{mix} \\ MCP_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} \\ \dot{m}_{systot} &= \sum_{inlets} \dot{m}_i \\ MCP_{tot} &= MCP_{zone} + MCP_{sys} \\ MCPT_{zone} &= MCPT_i + MCPT_{vent} + MCPT_{mix} \\ MCPT_{sys} &= \sum_{inlets} \dot{m}_i C_{p,i} T_i \\ MCPT_{tot} &= MCPT_{zone} + MCPT_{sys}\end{aligned}$$

The number of plumes per occupant $N_{plumesperpers}$ is a user input. The total number of plumes in the zone is:

$$N_{plumes} = N_{occ} \cdot N_{plumesperperson}$$

Using this we calculate the total power in the plumes and the power per plume.

$$\dot{Q}_{plumes} = \dot{Q}_{ocz}$$

$$\dot{Q}_{perplume} = \dot{Q}_{plumes} / N_{plumes}$$

The number of diffusers per plumes is also a user input. To obtain the number of diffusers in the zone:

$$N_{diffusers} = N_{plumes} \cdot N_{diffusersperplume}$$

The area A_{diff} is also a user input. For swirl diffusers and for displacement diffusers this area is used as input. For the variable area diffusers, though, we calculate the area. We assume 400 ft/min velocity at the diffuser and a design flow rate per diffuser is 150 cfm (.0708 m³/s). The design area of the diffuser is 150 ft³/min / 400 ft/min = .575 ft² = .035 m². Then the variable area each time step is

$$A_{diff} = \dot{m}_{systot} / (.0708 \cdot N_{diffusers})$$

We now calculate the height fraction Fr_{hb} (height of boundary layer divided by the total zone height).

$$\Gamma = \frac{(\dot{m}_{systot} \cdot \cos(\theta_{throw}))^{3/2}}{N_{plumes} (N_{diffusersperplume} \cdot A_{diff})^{5/4} (0.0281 \cdot 0.001 \cdot \dot{Q}_{plumes})^{1/2}}$$

$$Fr_{hb} = \frac{(N_{diffusersperplume} \cdot A_{diff})^{1/2} (7.43 \ln(\Gamma) - 1.35 + 0.5H_s)}{H_{ceil}}$$

where θ_{throw} is a user input: the angle between the diffuser slots and vertical; and H_s is the source height above the floor (m).

Next we iterate over the following 2 steps.

Iterative procedure

1. Call subroutine *HcUCSDUF* to calculate a convective heat transfer coefficient for each surface in the zone, an effective air temperature for each surface, and HA_{mx} , HAT_{mx} , HA_{oc} , HAT_{oc} . Here HA is $\sum_{surfaces} h_{c,i} \cdot A_i$ for a region and HAT is $\sum_{surfaces} h_{c,i} \cdot A_i \cdot T_i$ for a

region. The sum is over all the surfaces bounding the region; $h_{c,i}$ is the convective heat transfer coefficient for surface i , A_i is the area of surface i , and T_i is the surface temperature of surface i .

2. Calculate the two subzone temperatures: T_{oc} and T_{mx} .

The h_c 's calculated in step 1 depend on the subzone temperatures. In turn the subzone temperatures depend on the HA and HAT 's calculated in step 1. Hence the need for iteration

Next we describe each steps 1 and 2 in more detail.

Step 1

Subroutine *HcUCSDUF* is quite straightforward. It loops through all the surfaces in each zone and decides whether the surface is located in the upper, mixed subzone or the lower, occupied subzone, or if the surface is in both subzones. If entirely in one subzone the subzone temperature is stored in the surface effective temperature variable

$TempEffBulkAir(SurfNum)$ and h_c for the surface is calculated by a call to subroutine $CalcDetailedHcInForDVModel$. This routine uses the “detailed” natural convection coefficient calculation that depends on surface tilt and $\Delta T^{1/3}$. This calculation is appropriate for situations with low air velocity.

For surfaces that bound 2 subzones, the subroutine calculates h_c for each subzone and then averages them, weighting by the amount of surface in each subzone.

During the surface loop, once the h_c for a surface is calculated, the appropriate subzone HA and HAT sums are incremented. If a surface is in 2 subzones the HA and HAT for each subzone are incremented based on the area of the surface in each subzone.

Step 2

The calculation of subzone temperatures follows the method used in the **ZoneTempPredictorCorrector** module and described in the section **Basis for the System and Zone Integration**. Namely a third order finite difference expansion of the temperature time derivative is used in updating the subzone temperatures. Otherwise the subzone temperatures are obtained straightforwardly by solving an energy balance equation for each subzone.

$$T_{oc} = (C_{air,oc} \cdot (3 \cdot T_{-1,oc} - (3/2) \cdot T_{-2,oc} + (1/3) \cdot T_{-3,oc}) + \dot{Q}_{ocz} \cdot Fr_{gains} + HAT_{oc} + MCP_{tot}) / ((11/6) \cdot C_{air,oc} + HA_{oc} + MCP_{tot})$$

$$T_{mx} = (C_{air,mx} \cdot (3 \cdot T_{-1,mx} - (3/2) \cdot T_{-2,mx} + (1/3) \cdot T_{-3,mx}) + \dot{Q}_{ocz} \cdot (1 - Fr_{gains}) + \dot{Q}_{mxz} + HAT_{mx} + T_{oc} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,mx} + HA_{mx} + MCP_{tot})$$

Here $C_{air,oc}$ and $C_{air,mx}$ are the heat capacities of the air volume in each subzone. $C_{air,mx}$ is calculated by

$$R_{air,mx} = V_{mx} \cdot (\Delta z_{mx} / z_{ceil}) \cdot \rho_{air,mx} \cdot c_{p,air,mx} \cdot Mul_{cap} / (\Delta t_z \cdot 3600)$$

$$C_{air,mx} = R_{air,mx} \cdot \Delta t_z / \Delta t_{sys}$$

The gains fraction Fr_{gains} is a user input via a schedule. It is the fraction of the convective gains in the occupied subzone that remain in that subzone.

The other subzone air heat capacities are calculated in the same manner.

Mixed calculation

The above iterative procedure assumed that the UFAD nonuniform zone model was appropriate: i.e., conditions were favorable temperature stratification in the zone. Now that this calculation is complete and the subzone temperatures and depths calculated, we check to see if this assumption was justified. If not, zone conditions must be recalculated assuming a well-mixed zone.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < \Delta z_{occ,min}$ then the following mixed calculation will replace the UFAD interior zone calculation.

Note: $\Delta z_{occ,min}$ is the minimum thickness of occupied subzone. It is set to 0.2 meters.

The mixed calculation iteratively calculates surface convection coefficients and room temperature just like the displacement ventilation calculation described above. In the mixed case however, only one zone temperature T_{avg} is calculated. The 3 subzone temperatures are then set equal to T_{avg} .

First, Fr_{hb} is set equal to zero.

Then the code iterates over these steps.

Calculate T_{avg} using

$$T_{avg} = (C_{air,z} \cdot (3 \cdot T_{-1,z} - (3/2) \cdot T_{-2,z} + (1/3) \cdot T_{-3,z}) + \dot{Q}_{tot,conv} + HAT_{oc} + HAT_{mx} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,z} + HA_{oc} + HA_{mx} + MCP_{tot})$$

$$T_{mx} = T_{avg}$$

$$T_{oc} = T_{avg}$$

Call $HcUCSDUF$ to calculate the h_c 's.

Repeat step 1

Final calculations

The UFAD interior zone calculation finishes by calculating some report variables.

We define heights:

$$H_{trans} = Fr_{hb} \cdot H_{ceil}$$

$$H_{mxavg} = (H_{ceil} + H_{trans}) / 2$$

$$H_{ocavg} = H_{trans} / 2$$

Using the user defined comfort height we calculate the comfort temperature.

If mixing:

$$T_{comf} = T_{avg}$$

If UFAD:

If $H_{comf} < H_{ocavg}$

$$T_{comf} = T_{oc}$$

Else if $H_{comf} \geq H_{ocavg}$ and $H_{comf} < H_{mxavg}$

$$T_{comf} = (T_{oc} (H_{mxavg} - H_{comf}) + T_{mx} (H_{comf} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

Else if $H_{comf} \geq H_{mxavg}$ and $H_{comf} < H_{ceil}$

$$T_{comf} = T_{mx}$$

Using the user defined thermostat height we calculate the temperature at the thermostat.

If mixing:

$$T_{stat} = T_{avg}$$

If UFAD:

$$\text{If } H_{stat} < H_{ocavg}$$

$$T_{stat} = T_{occ}$$

$$\text{Else if } H_{stat} \geq H_{ocavg} \text{ and } H_{stat} < H_{mxavg}$$

$$T_{stat} = (T_{oc} (H_{mxavg} - H_{stat}) + T_{mx} (H_{stat} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

$$\text{Else if } H_{stat} \geq H_{mxavg} \text{ and } H_{stat} < H_{ceil}$$

$$T_{stat} = T_{mx}$$

The average temperature gradient is:

$$\text{If } H_{mxavg} - H_{ocavg} > 0.1$$

$$GradT_{avg} = (T_{mx} - T_{occ}) / (H_{mxavg} - H_{ocavg})$$

$$\text{else } GradT_{avg} = 0.0$$

Finally, the zone node temperature is set to T_{mx} .

Other variables that are reported out are Γ and ϕ .

$$\phi = (T_{mx} - T_{occ}) / (T_{mx} - T_{sup})$$

where T_{sup} is the zone supply air temperature.

References

Lin, Y.J. and P.F. Linden. 2005. A model for an under floor air distribution system. Energy&Building, Vol. 37, pp. 399-409.

Under-Floor Air Distribtuion Exterior Zone Model

Overview

The input object RoomAirSettings:UnderFloorAirDistributionExterior provides a simple model for heat transfer and a nonuniform vertical temperature profile for exterior zones of a UFAD system. These zones are expected to be dominated by internal loads, a portion of which (such as occupants and workstations) will act to generate plumes, and by window solar and conduction heat gains. The solar radiation penetrating the room is not expected to generate

plumes. However, a window plume is likely to be generated in sunny conditions, particularly if an interior blind is deployed. Thus the exterior UFAD zone will have potentially have plumes from people and equipment and plumes arising from the windows. The plumes act to potentially create room air stratification, depending on the type & number of diffusers, the amount and type of load, and the system flowrate. In order to better model this situation the fully-mixed room air approximation that is currently used in most whole building analysis tools is extended to a two node approach, with the purpose of obtaining a first order precision model for vertical temperature profiles for the exterior zones of UFAD systems. The use of 2 nodes allows for greatly improved prediction of thermal comfort and overall building energy performance for the increasingly popular UFAD systems.

The UCSD UFAD Exterior Zone Model is one of the non-uniform zone models provided through the Room Air Manager in EnergyPlus. The intent is to provide a selection of useful non-uniform zone air models to enable the evaluation of air-conditioning techniques that use stratified or partially stratified room air. Such techniques include displacement ventilation (DV) and underfloor air distribution (UFAD) systems. The methodology can also include natural displacement ventilation and also wind-driven cross-ventilation (CV).

Underfloor air distribution systems

UFAD systems represent, in terms of room air stratification, an intermediate condition between a well-mixed zone and displacement ventilation. Air is supplied through an underfloor plenum at low pressure through diffusers in the raised floor. The diffusers can be of various types: e.g., swirl, variable-area, displacement, and produce more or less mixing in the zone. UFAD systems are promoted as saving energy due to: higher supply air temperature; low static pressure; cooler conditions in the occupied subzone than in the upper subzone; and sweeping of some portion of the convective load (from ceiling lights, for instance) into the return air without interaction with the occupied region of the zone.

Modeling a UFAD system is quite complex and involves considerably more than just a non-uniform zone model. The zones' coupling to the supply and return plenums must be modeled accurately (particularly radiative transfer from a warm ceiling to a cool floor and on into the supply plenum by conduction). The supply plenum must be accurately modeled, giving a good estimate of the supply air temperature and conduction heat transfer between supply & return plenums through the slab. The HVAC system must be modeled including return air bypass and various types of fan powered terminal units.

The UCSD UFAD exterior zone model is similar to the UCSD interior zone model. The most obvious difference is that the exterior UFAD has 2 different types of plume sources: people & equipment and windows. Like the UCSD UFAD interior model it is a plume equation based multi-layer model (2 layers in this case). The zone is modeled as being divided into 2 well separated subzones which we denote as "occupied" and "upper". Each subzone is treated as having a single temperature. The boundary between the 2 subzones moves up & down each time step as a function of zone loads and supply air flow rate. Thus at each HVAC time step, the height of the boundary above the floor must be calculated, portions of surfaces assigned to each subzone, and a separate convective heat balance performed on each subzone.

Model Description

As in the interior zone case, we define 2 dimensionless parameters: Γ and ϕ . The definitions of the 2 parameters are the same as in the previous section (equations (219), (220), (221), (222), and (223)). As in the previous case, the experimental data can be plotted versus Γ and lines fitted to the data give the following formulas for occupied subzone temperature and interface height.

$$T_{oz} = T_r - 1.4\Gamma^{-0.6}(T_r - T_s) \quad (227)$$

$$h = \sqrt{\frac{n}{m}} A(11.03 \ln(\Gamma) - 10.73) + \frac{1}{2} h_s \quad (228)$$

where T_{oz} is the occupied subzone temperature (K); T_r is the return temperature (K); Γ is the dimensionless height parameter defined above; T_s is the supply temperature (K); h is the interface height (m); n is the number of diffusers; m is the number of heat sources; A is the effective area of a diffuser (m^2); and h_s is the vertical position of the heat sources (m). The formula for T_r is the same as in the previous section.

Implementation

The implementation closely follows the procedure described in the UFAD interior zone model. The model predicts two temperatures that characterize the two main levels in the stratification of the room:

1. an occupied subzone temperature T_{oc} representing the temperature of the occupied region;
2. an upper level temperature T_{mx} representing the temperature of the upper, mixed region and the outflow temperature.

We will use (228) to calculate the interface height and do a heat balance calculation on each subzone. Γ is given by (221). The supply air flow rate \dot{V} is obtained by summing all the air flows entering the zone: supply air, infiltration, ventilation, and inter-zone flow. The heat gain \dot{Q} is estimated by summing all the convective internal gains located in the occupied subzone – task lights, people, equipment – and adding to this the convective gain coming from the window surface. With these assumptions we can describe the implementation.

The UCSD UFE model is controlled by the subroutine *ManageUCSDUFModels* which is called from the *RoomAirModelManager*. The *RoomAirModelManager* selects which zone model will be used for each zone.

The calculation is done in subroutine *CalcUCSDUEI*. First we calculate the convective heat gain going into the upper and lower regions.

$$\begin{aligned} \dot{Q}_{ocz} = & \dot{Q}_{oc,conv} + \dot{Q}_{il,conv} + \dot{Q}_{eleq,conv} + \dot{Q}_{gaseq,conv} + \dot{Q}_{otheq,conv} + \dot{Q}_{hw,conv} \\ & + \dot{Q}_{stmeq,conv} + \dot{Q}_{bb,conv} + \dot{Q}_{waterheater,conv} + \dot{Q}_{fuelcell,conv} \end{aligned}$$

$$\dot{Q}_{mxz} = \dot{Q}_{gl,conv} + \dot{Q}_{ltp} + \dot{Q}_{htrrad,conv}$$

$$\dot{Q}_{tot,conv} = \dot{Q}_{ocz} + \dot{Q}_{mxz}$$

Next we sum up the inlet air flows in the form of MCP (mass flow rate times the air specific heat capacity) and MCPT (mass flow rate times C_p times air temperature).

$$MCP_{zone} = MCP_i + MCP_{vent} + MCP_{mix}$$

$$MCP_{sys} = \sum_{inlets} \dot{m}_i C_{p,i}$$

$$\dot{m}_{systot} = \sum_{inlets} \dot{m}_i$$

$$MCP_{tot} = MCP_{zone} + MCP_{sys}$$

$$MCPT_{zone} = MCPT_i + MCPT_{vent} + MCPT_{mix}$$

$$MCPT_{sys} = \sum_{inlets} \dot{m}_i C_{p,i} T_i$$

$$MCPT_{tot} = MCPT_{zone} + MCPT_{sys}$$

For exterior zone model, we assume one plume: $N_{plumes} = 1$. The number of diffusers in the zone $N_{diffusers}$ is a user input.

The area A_{diff} is also a user input. For swirl diffusers, linear bar grilles, and displacement diffusers this area is used as input. For the variable area diffusers, though, we calculate the area. We assume 400 ft/min velocity at the diffuser and a design flow rate per diffuser is 150 cfm (.0708 m³/s). The design area of the diffuser is 150 ft³/min / 400 ft/min = .575 ft² = .035 m². Then the variable area each time step is

$$A_{diff} = \dot{m}_{systot} / (.0708 \cdot N_{diffusers})$$

We now make an initial estimate of the convective gain from the windows.

$$\dot{Q}_{win,conv} = \sum_{i=1, n_{win}} h_{c,i} A_{win,i} T_{winsurf,i} - h_{c,i} A_{win,i} T_{mat}$$

Then

$$\dot{Q}_{plumes} = \dot{Q}_{ocz} + \dot{Q}_{win,conv}$$

We now calculate the height fraction Fr_{hb} (height of boundary layer divided by the total zone height).

$$\Gamma = \frac{(\dot{m}_{systot} \cdot \cos(\theta_{throw}))^{3/2}}{(N_{diffusers} \cdot A_{diff})^{5/4} (0.0281 \cdot 0.001 \cdot \dot{Q}_{plumes})^{1/2}}$$

$$Fr_{hb} = \frac{(N_{diffusers} \cdot A_{diff})^{1/2} (11.03 \ln(\Gamma) - 10.73 + 0.5 H_s)}{H_{ceil}}$$

where θ_{throw} is a user input: the angle between the diffuser slots and vertical; and H_s is the source height above the floor (m).

Next we iterate over the following 2 steps.

Iterative procedure

1. Call subroutine *HcUCSDUF* to calculate a convective heat transfer coefficient for each surface in the zone, an effective air temperature for each surface, and HA_{mx} , HAT_{mx} , HA_{ocz} , HAT_{ocz} , $HA_{mx,win}$, $HAT_{mx,win}$, $HA_{ocz,win}$, and $HAT_{ocz,win}$. Here HA is $\sum_{surfaces} h_{c,i} \cdot A_i$ for a

region and HAT is $\sum_{surfaces} h_{c,i} \cdot A_i \cdot T_i$ for a region. The sum is over all the surfaces bounding the region; $h_{c,i}$ is the convective heat transfer coefficient for surface i , A_i is the area of surface i , and T_i is the surface temperature of surface i . Variables with the *win* subscript are summed over window surfaces only. Then the convective gain from the window is recalculated:

$$\dot{Q}_{win,conv} = HAT_{mx,win} + HAT_{oc,win} - HA_{mx,win} T_{mx} - HA_{oc,win} T_{oc}$$

and the power in the plume is recalculated: $\dot{Q}_{plumes} = \dot{Q}_{ocz} + \dot{Q}_{win,conv}$.

2. Calculate the two subzone temperatures: T_{oc} and T_{mx} .

The h_c 's calculated in step 1 depend on the subzone temperatures. In turn the subzone temperatures depend on the HA and HAT's calculated in step 1. Hence the need for iteration

Next we describe each steps 1 and 3 in more detail.

Step 1

Subroutine *HcUCSDUF* is quite straightforward. It loops through all the surfaces in each zone and decides whether the surface is located in the upper, mixed subzone or the lower, occupied subzone, or if the surface is in both subzones. If entirely in one subzone the subzone temperature is stored in the surface effective temperature variable *TempEffBulkAir(SurfNum)* and h_c for the surface is calculated by a call to subroutine *CalcDetailedHcInForDVModel*. This routine uses the "detailed" natural convection coefficient calculation that depends on surface tilt and $\Delta T^{1/3}$. This calculation is appropriate for situations with low air velocity.

For surfaces that bound 2 subzones, the subroutine calculates h_c for each subzone and then averages them, weighting by the amount of surface in each subzone.

During the surface loop, once the h_c for a surface is calculated, the appropriate subzone HA and HAT sums are incremented. If a surface is in 2 subzones the HA and HAT for each subzone are incremented based on the area of the surface in each subzone. The subroutine calculates a separate HA and HAT for the windows for use in calculating the window convective gain.

Step 2

The calculation of subzone temperatures follows the method used in the **ZoneTempPredictorCorrector** module and described in the section **Basis for the System and Zone Integration**. Namely a third order finite difference expansion of the temperature time derivative is used in updating the subzone temperatures. Otherwise the subzone temperatures are obtained straightforwardly by solving an energy balance equation for each subzone.

$$T_{oc} = (C_{air,oc} \cdot (3 \cdot T_{-1,oc} - (3/2) \cdot T_{-2,oc} + (1/3) \cdot T_{-3,oc}) + \dot{Q}_{ocz} \cdot Fr_{gains} + HAT_{oc} + MCPT_{tot}) / ((11/6) \cdot C_{air,oc} + HA_{oc} + MCP_{tot})$$

$$T_{mx} = (C_{air,mx} \cdot (3 \cdot T_{-1,mx} - (3/2) \cdot T_{-2,mx} + (1/3) \cdot T_{-3,mx}) + \dot{Q}_{ocz} \cdot (1 - Fr_{gains}) + \dot{Q}_{mxz} + HAT_{mx} + T_{oc} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,mx} + HA_{mx} + MCP_{tot})$$

Here $C_{air,oc}$ and $C_{air,mx}$ are the heat capacities of the air volume in each subzone. $C_{air,mx}$ is calculated by

$$R_{air,mx} = V_{mx} \cdot (\Delta z_{mx} / z_{ceil}) \cdot \rho_{air,mx} \cdot c_{p,air,mx} \cdot Mul_{cap} / (\Delta t_z \cdot 3600)$$

$$C_{air,mx} = R_{air,mx} \cdot \Delta t_z / \Delta t_{sys}$$

The gains fraction Fr_{gains} is a user input via a schedule. It is the fraction of the convective gains in the occupied subzone that remain in that subzone.

The other subzone air heat capacities are calculated in the same manner.

Mixed calculation

The above iterative procedure assumed that the UFAD zone model was applicable: i.e., conditions were favorable temperature stratification in the zone. Now that this calculation is complete and the subzone temperatures and depths calculated, we check to see if this assumption was justified. If not, zone conditions must be recalculated assuming a well-mixed zone.

If $T_{mx} < T_{oc}$ or $MCP_{tot} \leq 0$ or $H_{fr} \cdot H_{ceil} < \Delta z_{occ,min}$ then the following mixed calculation will replace the UFAD exterior zone calculation.

Note: $\Delta z_{occ,min}$ is the minimum thickness of occupied subzone. It is set to 0.2 meters.

The mixed calculation iteratively calculates surface convection coefficients and room temperature just like the displacement ventilation calculation described above. In the mixed case however, only one zone temperature T_{avg} is calculated. The 2 subzone temperatures are then set equal to T_{avg} .

First, Fr_{hb} is set equal to zero.

Then the code iterates over these steps.

Calculate T_{avg} using

$$T_{avg} = (C_{air,z} \cdot (3 \cdot T_{-1,z} - (3/2) \cdot T_{-2,z} + (1/3) \cdot T_{-3,z}) + \dot{Q}_{tot,conv} + HAT_{oc} + HAT_{mx} \cdot MCP_{tot}) / ((11/6) \cdot C_{air,z} + HA_{oc} + HA_{mx} + MCP_{tot})$$

$$T_{mx} = T_{avg}$$

$$T_{oc} = T_{avg}$$

Call $HcUCSDUF$ to calculate the h_c 's.

Repeat step 1

Final calculations

The UFAD exterior zone calculation finishes by calculating some report variables.

We define heights:

$$H_{trans} = Fr_{hb} \cdot H_{ceil}$$

$$H_{mxavg} = (H_{ceil} + H_{trans}) / 2$$

$$H_{ocavg} = H_{trans} / 2$$

Using the user defined comfort height we calculate the comfort temperature.

If mixing:

$$T_{comf} = T_{avg}$$

If UFAD:

If $H_{comf} < H_{ocavg}$

$$T_{comf} = T_{occ}$$

Else if $H_{comf} \geq H_{ocavg}$ and $H_{comf} < H_{mxavg}$

$$T_{comf} = (T_{oc} (H_{mxavg} - H_{comf}) + T_{mx} (H_{comf} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

Else if $H_{comf} \geq H_{mxavg}$ and $H_{comf} < H_{ceil}$

$$T_{comf} = T_{mx}$$

Using the user defined thermostat height we calculate the temperature at the thermostat.

If mixing:

$$T_{stat} = T_{avg}$$

If UFAD:

If $H_{stat} < H_{ocavg}$

$$T_{stat} = T_{occ}$$

Else if $H_{stat} \geq H_{ocavg}$ and $H_{stat} < H_{mxavg}$

$$T_{stat} = (T_{oc} (H_{mxavg} - H_{stat}) + T_{mx} (H_{stat} - H_{ocavg})) / (H_{mxavg} - H_{ocavg})$$

Else if $H_{stat} \geq H_{mxavg}$ and $H_{stat} < H_{ceil}$

$$T_{stat} = T_{mx}$$

The average temperature gradient is:

If $H_{mxavg} - H_{occavg} > 0.1$

$$GradT_{avg} = (T_{mx} - T_{occ}) / (H_{mxavg} - H_{occavg})$$

else $GradT_{avg} = 0.0$

Finally, the zone node temperature is set to T_{mx} .

Other variables that are reported out are Γ and ϕ .

$$\phi = (T_{mx} - T_{occ}) / (T_{mx} - T_{sup})$$

where T_{sup} is the zone supply air temperature.

References

Qing Liu. 2006. The Fluid Dynamics of an Underfloor Air Distribution System. A PhD dissertation in Engineering Sciences (Systems Science) at UC San Diego.

Cross Ventilation Room Air Model

Overview

The input object RoomAirSettings:CrossVentilation provides a simple model, developed by UCSD, for heat transfer and vertical temperature profile prediction in cross ventilated rooms. This model was developed using scaling analysis, experimental correlations, computational fluid dynamics, and approximate solutions of the Navier-Stokes equations. The model distinguishes two regions in the room, the main jet region and the recirculations, and predicts characteristic airflow velocities and average air temperatures. The model consists of a set of analytical expressions that clearly display the first order effects of room geometry and inflow characteristics on Cross Ventilation airflow and heat transfer.

Cross-Ventilation

Cross Ventilation (CV) is common in many naturally ventilated buildings, with air flowing through windows, open doorways and large internal apertures across rooms and corridors in the building. In these cases significant conservation of inflow momentum occurs with the inflow traveling freely across the room. Because of the high momentum conservation, CV strategies are often used when there is need for high ventilation airflow rates. CV flows may be caused by wind or a mechanical ventilation system. CV flows often occur in wind-driven ventilation systems with inflow through operable windows.

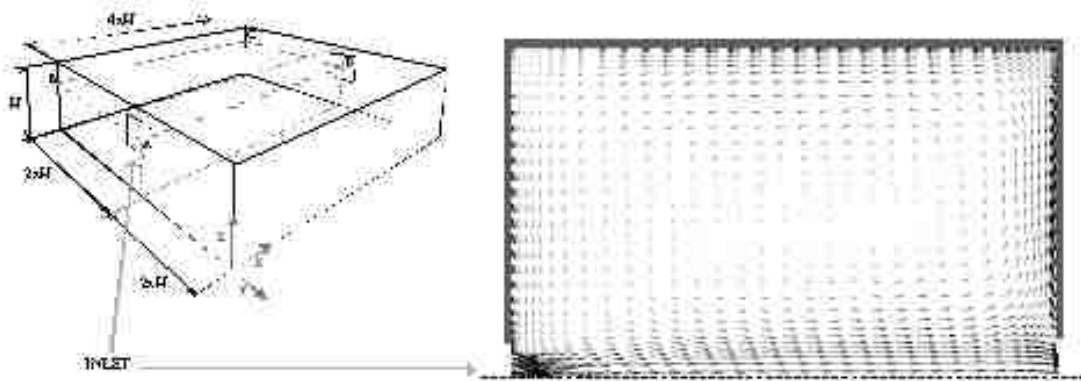


Figure 110. Left: basic compartment geometry considered in the model. Right: top view of one half of the velocity field (at height $H/2$), result of a CFD simulation using the geometry on the left (taking advantage of the system symmetry to simulate only one half of the room volume).

The left hand side of Figure 110 shows a simple room geometry that can lead to cross-ventilation. As ventilation air flows across the room, heat transfer between airflow, room surfaces and internal heat sources occurs and the airflow temperature changes between inlet and outlet, reflecting energy conservation. Heat transfer between air and room surfaces is important in both of these passive-cooling strategies. Its magnitude has considerable influence in the effective indoor temperature as well as in determining the success of the night cooling system. Several studies have shown that surface heat transfer has a large impact on energy consumption predictions in mechanical systems (Carrilho da Graça, 2003, Kalema & Haapala, 1995). However, before the introduction of the present model, the required level of modeling detail was not available in a whole building thermal simulation tool.

Recirculating flow

The figure below shows a schematic representation of the two basic airflow patterns that can occur in CV. Any cross-ventilated room will have an airflow pattern that is either similar to one of the two base cases shown below (with or without recirculations), or a combination of the two with both recirculation and inlet flow attaching to a lateral surface or the ceiling.

The simplest flow configuration, with no recirculations, commonly occurs in corridors and long spaces whose inlet aperture area is similar to the room cross-sectional area. In this case, the flow occupies the full cross section of the room and the transport of pollutants and momentum is unidirectional, similar to turbulent flow in a channel. The flow velocity profile across the channel is approximately flat as a result of the high degree of mixing that is characteristic of turbulent flows. The average airflow velocity in the cross section can be obtained approximately by dividing the flow rate by the cross sectional area of the space.

A more complex airflow pattern occurs when the inlet aperture area is an order of magnitude smaller than the cross sectional area of the room $A_R = W.H$ (for the case shown in Figure 110, $A_R = 4.H^2$). In these cases, the main CV region in the core of the room entrains air from the adjacent regions and forms recirculations that ensure mass conservation, with air moving in the opposite direction to the core jet flow. These recirculating flow regions have been observed in many experiments. The most relevant to the present problem are (Aynsley *et al.* 1977, Baturin & Billington, 1972, Neiswanger *et al.* 1987, Ohba *et al.* 2001). In these room geometries, when the inlet is located close to the center of the inlet surface, most of the contact between ventilation flow and the internal surfaces occurs in the recirculation regions that occupy the majority of the room volume.

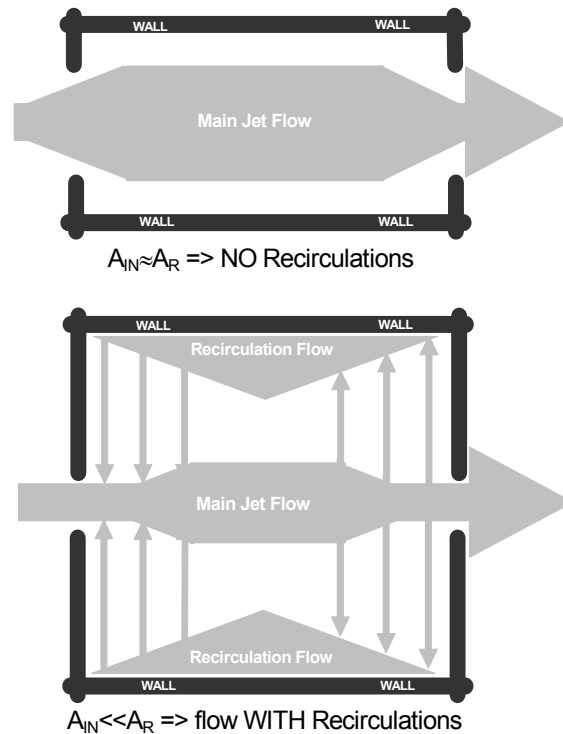


Figure 111. Top View of the possible airflow patterns in cross-ventilation.

Effects of recirculating flow in heat transfer

Ventilation air changes temperature as it is exposed to different parts of the internal room surfaces. Often, these surfaces are flat (walls, floors, furniture etc), but, nevertheless, simple use of flat plate correlations is not possible as the forced air flow velocity or free stream air temperature are not known since the confined flow occurring in most rooms does not resemble a free stream. Figure 112 shows, schematically, the differences between flat plate heat transfer (a)), and two types of cross ventilation flows (no recirculations in b) and with recirculations in c)).

In the figure a), the boundary layer that develops exchanges heat with the environment, a heat sink, with infinite thermal capacity.

In the figure b) and c), the effects of flow confinement are clear, while flat plate heat transfer occurs at constant free stream temperature ($T_{IN}=T_{OUT}$ in a)), all room ventilation situations will display variable temperature gradients as the air temperature changes along the flow (see T_{R1} and T_{R2} in the figure b), c), d)), as a consequence of heat transfer into a flow stream with limited heat capacity (that depends on the local ventilation rate in different parts of the room volume).

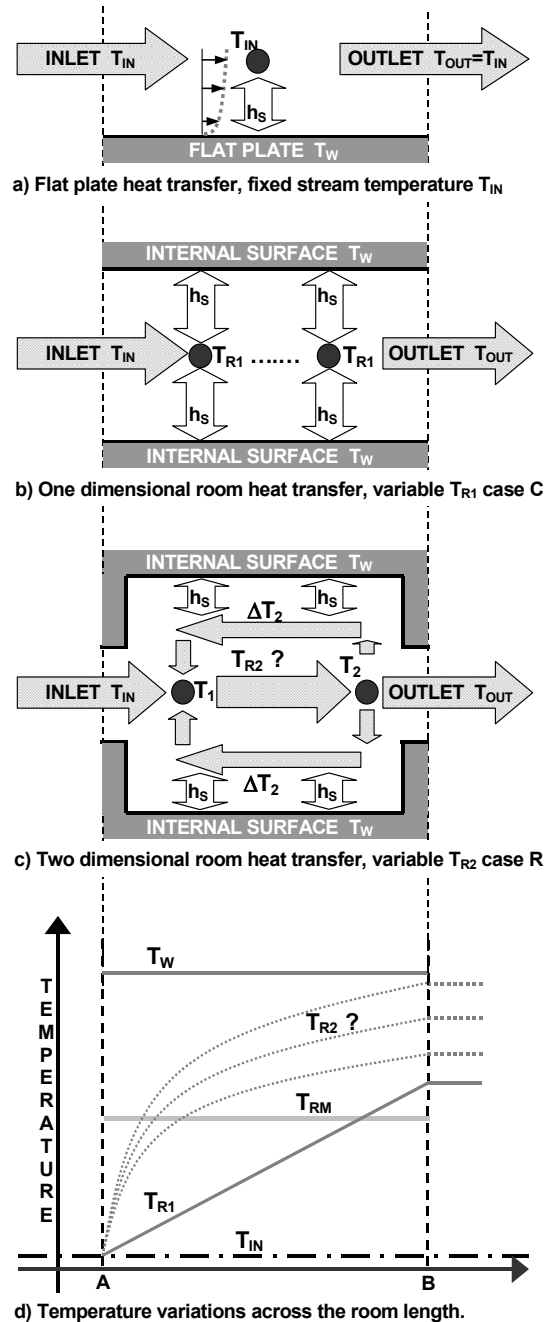


Figure 112. Differences between flat plate heat transfer and cross ventilation flow

In a one-node, fully-mixed model the room temperature is constant (T_{RM} in d)). Whenever mixing is not perfect, the temperature gradients that determine the heat transfer with the room surfaces will be incorrectly modeled. The differences between CFD simulations of recirculating flows with a fixed heat source in the recirculation region and mixed model predictions can reach 1.5°C (Carrilho da Graça 2003). Confinement effects due to recirculation flow are also very influential in internal surface heat transfer predictions where using a perfectly mixed flow, one node model leads to errors of up to 100%.

Existing approaches

In order to predict heat transfer in CV airflows there are currently four available options: computational fluid dynamics (CFD, using Reynolds averaged turbulence models), zonal models, fully mixed room air heat transfer models and experimental correlations.

In many situations, designers have to analyze multi-room ventilation geometries, and, due to the nonlinear interactions that exist in building heat transfer processes, have the need to simulate using local measured weather spanning several days or months. In these cases, the use of CFD is impractical and simpler heat transfer models are needed.

Zonal models simulate room heat transfer by numerically solving for mass and energy conservation in a set of fully mixed zones (often less than twenty). These models generally require user identification of the dominant room airflow components (jets, boundary layers, plumes, etc.) that are “contained” in particular zones (Allard 1992). Because the momentum equation is not solved in the iteration procedure, an artificial flow resistance is imposed between room zones. These features make these models complex to use and often imprecise.

Experimental correlations can be a good solution to model complex physical systems. For the present problem, correlations are only available for particular cases (Chandra & Kerestecioglu 1984, Altmayer et al. 1988, Spitler et al. 1991, Awbi & Hatton 2000). Application of these correlations to most geometries found in real design is difficult as the adequate correlation must be identified (by matching the design geometry to the geometries used to develop the correlations) and is often not available.

Fully mixed room air heat transfer models use a single modeling point to characterize indoor air temperature in the room (one example is the mixed flow model, also available in EnergyPlus). These models, while simple to integrate in energy analysis software, are, in their present form, precise only when the flow is mixed. In all other cases these models fail to predict the magnitude of the heat transfer and also most room geometry induced effects. This inaccuracy is a consequence of the use of two major approximations:

- Room air is considered fully mixed. Temperature variations inside the room are not modeled. This approximation neglects the effects of the temperature variations that occur as air moves across a room and comes into contact with internal surfaces and heat sources resulting in inaccurate calculation of the thermal capacity of the airflow. With this approximation, even if the correct local heat transfer coefficient is used the result can still be imprecise.
- Difficulties in determining the local forced convection heat transfer coefficient lead modelers to consider only natural convection, ignoring airflow pattern effects.

Model Description

In order to model heat transfer and evaluate thermal comfort, two interrelating components of the CV system must be considered: the airflow pattern and the magnitude of the local heat transfer between room air and internal surfaces (partition elements, occupants, etc.).

Each of the two parts of the problem poses considerable challenges. By definition, any CV airflow pattern has an element of direct air movement between inlet and outlet, but, in some regions of the room air can move in other directions. Transfers between airflow and the internal sources depend on the local concentration gradient and transfer coefficient. In particular, when modeling the effects of internal thermal mass, it is relevant to determine how much energy from the internal gains is transferred to the internal surfaces, and not exhausted by the ventilation air.

Airflow pattern

The main challenge when modeling the airflow pattern is to predict the flow characteristics in the recirculation regions. In the present model this characterization is done using scaling analysis principles.

The CV flows to which the model applies are bounded by a stationary geometry and are dominated by horizontal momentum flux, as opposed to buoyancy. For ventilation systems with these characteristics, if the flow regime is stable (either laminar, or, as in the present case, dominated by turbulence), the flow pattern will also be stable, approximately self similar and suitable for the application of scaling analysis principles. In these flows, all the velocities in the room are expected to scale linearly with the characteristic velocity of the inlet flow (Carrilho da Graça 2003):

$$U_R = C_n \cdot f(L, W, H, A_{IN}, \dots) \cdot U_{IN} \quad (229)$$

The function f is expected to depend on L , W , H , A_{IN} , and inlet location. Using dimensional analysis principles it is possible to compose two non-dimensional parameters with the three independent length scales in this problem:

1. Square root of the inlet area.
2. Square root of the room cross section (perpendicular to the CV flow $A_R=W.H$).
3. Room length in the CV direction.

We propose that the two independent parameters (formed using the length scales) that the function f will depend on are:

$$L^* = \frac{L}{\sqrt{A_{IN}}}, \quad A^* = \frac{A_R}{A_{IN}} \quad (230)$$

The velocity scale (U_{IN}) of the inlet jet, is defined using inflow momentum flux:

$$M = \int_{A_{IN}} \rho \cdot (U_{MAX} \mathbf{g}(\vec{r}))^2 dA, \quad U_{IN} = \sqrt{M / \rho A_{IN}} \quad (231)$$

In order to correlate the velocities in different regions of the room the corresponding correlation constants C_n and scaling laws f (see right hand side of Figure 110) must be obtained. By multiplying the velocities predicted using expression (229) by the area over which the recirculation flow occurs correlations for the magnitude of the recirculation flow rates can be obtained.

Sets of CFD simulations (Carrilho da Graça 2003), based on geometries similar to Figure 110 were used to obtain the correlation constants and to validate the simplifying assumptions used when developing the correlation expressions (the “ f ” functions). The simulations spanned rooms with variable dimensions (height: 2.25-3.4m, Length: 2.25-13.5m, Width: 4.5-9.0m) and inflow aperture areas (1-4m²). Two clear trends were visible in these flows: recirculations are present whenever A^* is bigger than two and the behavior of the recirculations depends on the length to width ratio of the room. The criterion that most successfully distinguishes between different “groups” of rooms is are:

$$A^* > 2 \Leftrightarrow \frac{A_{IN}}{A_R} < \frac{1}{2}, \quad C_L = \frac{2L}{W - W_{IN}} \quad (232)$$

The scaling principle behind the correlations is: the momentum flux through the inlet aperture is the dominant feature in the CV flow. This momentum flows into the room in the form of a jet whose characteristic dimension is typically not more than an order of magnitude smaller than the room length (or the characteristic room dimension $((L.W.H)^{1/3})$), resulting in a jet flow that is never fully developed and strongly confined. This correlation results in the following expression for the velocity in the recirculating portion of the flow:

$$U_R^2 = C_M \frac{\sqrt{A_{IN}} \cdot L}{A_R} U_{IN}^2 \quad (233)$$

This scaling relation can also be obtained from a simplified solution of the Navier-Stokes equations, by evaluating the momentum flux through the shear layer that develops in the perimeter of the inflow jet.

The modeling principle described above is used to predict four parameters that will then be used to model thermal comfort and heat transfer in cross ventilated rooms (described below). These parameters are the characteristic velocities in the jet and recirculation regions (for thermal comfort), the maximum airflow rate in the recirculation regions (defining the heat capacity of the recirculating airflow) and the average airflow velocity near the internal partitions (for forced convection heat transfer). Internal forced convection models are not available in EnergyPlus (for natural convection the *ASHRAE Detailed* model is available). In most cross ventilation airflows heat transfer occurs though mixed convection and the forced convection component is relevant and must be modeled.

Recirculation Region Average Velocity

The average recirculating flow velocity in the room cross section with maximum flow rate is predicted using the following expression ($1/3 < C_L < 11$):

$$U_R = C_R \sqrt{\frac{L}{A_R \cdot A_{IN}^{3/2}}} F_{IN} \quad , \quad C_R = \begin{cases} 0.298, & 1/3 \leq C_L \leq 4 \\ 0.162, & 4 < C_L \leq 11 \end{cases} \quad (234)$$

Figure 113 shows the results of the correlation for the standard (gray line) and long rooms (dashed line). The correlation expression labeled with a (*) refers to the cases with $C_L > 4$ (long rooms). As expected the slopes in these correlations are smaller than for the standard correlations, as a consequence of the higher dissipation that occurs in long rooms.

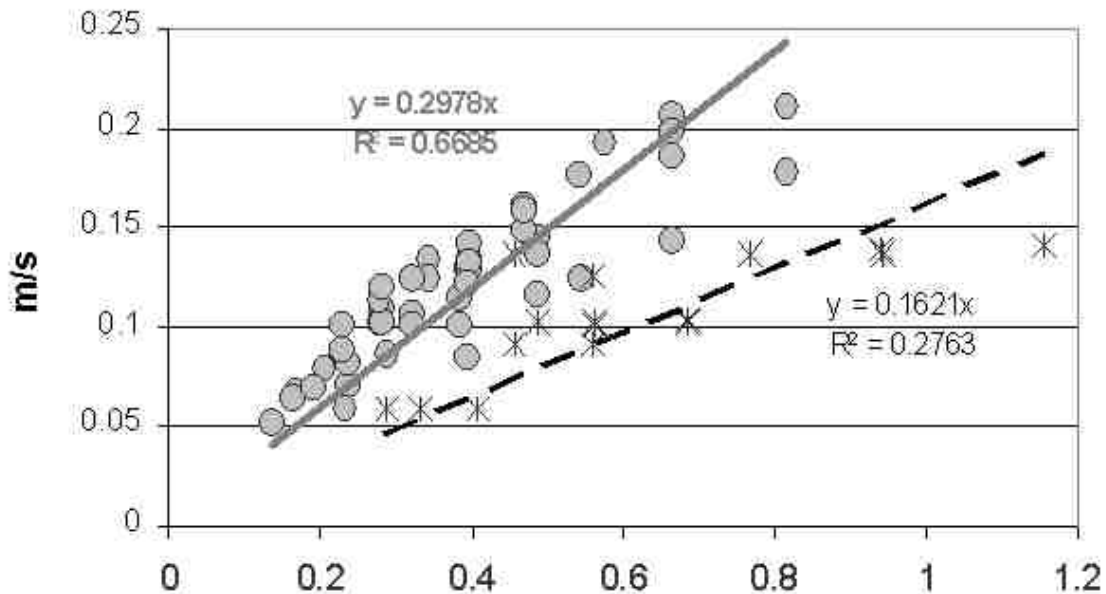


Figure 113. Average velocity in the recirculation region.

Correlation for standard rooms ($C_L < 4$): gray line. Long rooms ($4 < C_L < 11$): dashed line. The horizontal axis is the value of the non-dimensional factor on the right hand side of each correlation expression (see equation (234)).

Jet Region Average Velocity

Average volumetric velocity in the main jet region is given by ($1/3 < C_L < 11$):

$$U_J = 1.56 \frac{F_{IN}}{\sqrt{A_R \cdot A_{IN}}} \quad , \quad \frac{1}{3} \leq C_L \leq 11 \quad (235)$$

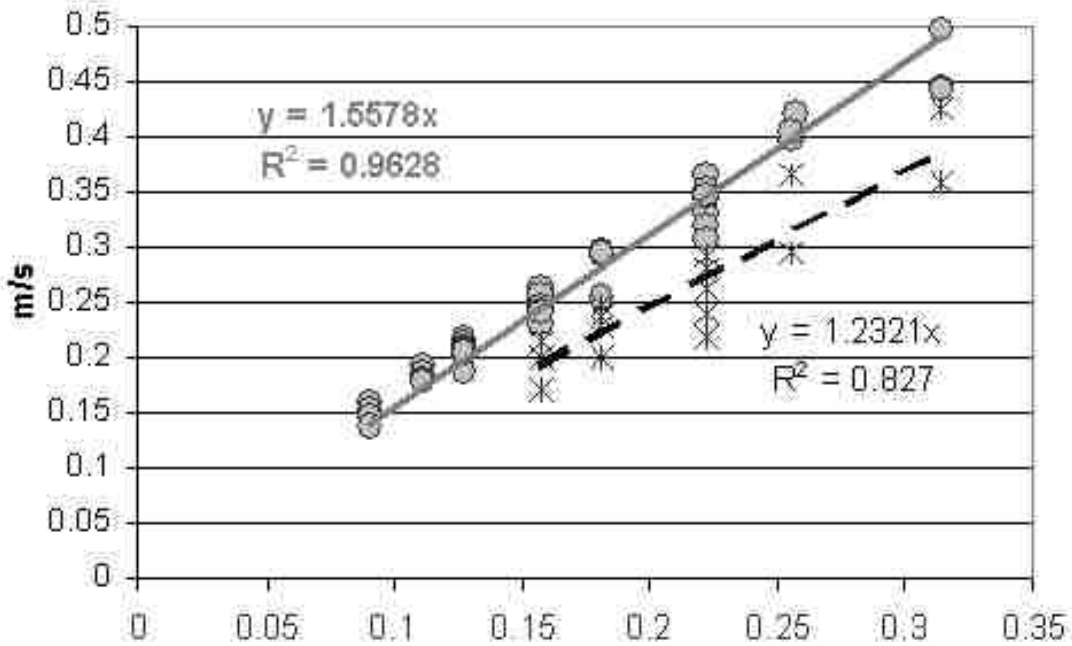


Figure 114. Average velocity in the main jet region.

Correlation for standard rooms ($C_L < 4$): gray line. Long rooms ($4 < C_L < 11$): dashed line. The horizontal axis is the value of the non-dimensional factor on the right hand side of each correlation expression (see right hand side of expression (235)).

Total airflow rate in the recirculation regions

The total volumetric flow rate of the recirculation region of the room airflow is given by ($1/3 < C_L < 11$):

$$F_R = C_F \sqrt{\frac{L A_R}{A_{IN}^{3/2}} \cdot F_{IN}} \quad , \quad C_F = \begin{cases} 0.147, & \frac{1}{3} \leq C_L \leq 4 \\ 0.077, & 4 < C_L \leq 11 \end{cases} \quad (236)$$

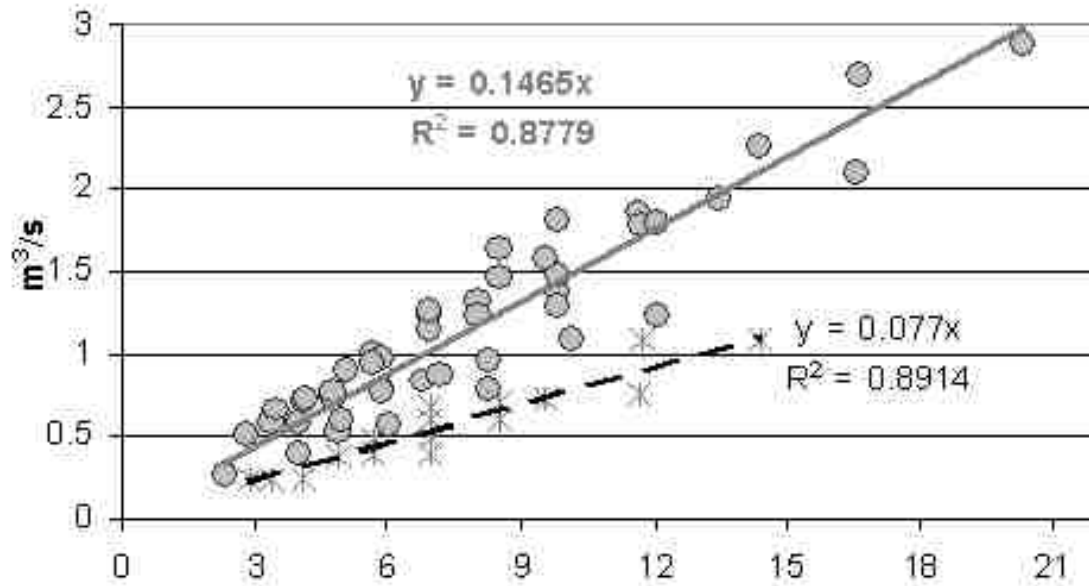


Figure 115. Maximum flow rate in the recirculation region.

Correlation for standard rooms ($C_L < 4$): gray line. Long rooms ($4 < C_L < 11$): dashed line. The horizontal axis is the value of the non-dimensional factor on the right hand side of each correlation expression (see right hand side of expression (236)).

Forced convection

In CV flow patterns that conform to the conditions mentioned in the previous section, airflow is predominantly turbulent and there is forced approximately horizontal air movement close to the room surfaces. Since natural convection always occurs when there is a temperature difference between the air and a surface, room heat transfer always occurs through mixed convection. The boundary layers that form close to the thermally active room surfaces are then driven by buoyancy and momentum transferred from the dominant CV flow. In order to predict the local mixed convection heat transfer coefficient that results from these two mechanisms it is possible to use the following formula (Siebers et al. 1983):

$$h_{MX} = (h_F^\varepsilon + h_N^\varepsilon)^{1/\varepsilon} \quad (237)$$

This formula was verified experimentally (Seibers et al. 1983) and has been subsequently used in two other studies that are relevant to the present work: with vertical surfaces and horizontal forced flow (Neiswanger et al. 1987) and horizontal surfaces with horizontal and vertically impinging forced flow (Awbi&Hatton 2000). In all of these studies, Equation (237) was found to agree better with experimental measurements when ε was set to 3.2.

Air enters the room in a developing jet composed of two shear layers that quickly become turbulent. Part of the air in the shear layers goes into the recirculation regions forming the wall jets, or wall currents, that exist in these regions. Since the air is already turbulent, there is no significant laminar region in these boundary layers. For this reason, it is a reasonable approximation to use a correlation for forced convection turbulent boundary layers in all the internal surfaces. Colburn's analogy between heat and momentum transfers (Bejan 1994) to estimate the forced component of the convection heat transfer, resulting in:

$$h_F = \rho c_p U_\infty P_r^{-\frac{2}{3}} \frac{1}{2} C_{fx} = \rho c_p U_\infty^{\frac{4}{5}} P_r^{-\frac{2}{3}} 0.0296 \left(\frac{x}{\nu} \right)^{-\frac{1}{5}} \quad (238)$$

The heat transfer coefficient has a weak dependence on x through an exponent of -0.2 . This is a consequence of boundary layer growth and the resulting reduction in the temperature gradient. Forced convection boundary layer heat transfer is not influenced by buoyancy driven transport (the turbulent eddy formation process is driven by momentum transfer) and for this reason there is no temperature or orientation dependence in the result.

In a simple room heat transfer model, it is not feasible to integrate the x dependent coefficient in equation (238) for every room surface. Since the x dependence is small, the effects of different lengths, depending on room geometry, are smaller than the model's overall accuracy. Therefore, a further approximation is introduced: an integration is performed only once using a typical room surface length of 6 meters (a characteristic value, half way between short $L=2m$ and long rooms $L=10m$). This result will be used for all room surfaces (floor, ceiling, lateral walls and windows). Performing the integration using standard properties results in:

$$h_F = 4.3 U_S^{0.8} \quad (239)$$

The proposed correlation is:

$$U_S^{0.8} = C_S \left(\frac{121 + L/\sqrt{A_{IN}}}{A_R/A_{IN}} \right)^{0.4} U_{IN}^{0.8}, \quad C_S = \begin{cases} 0.115, & \frac{1}{3} \leq C_L \leq 4 \\ 0.082, & 4 < C_L \leq 11 \end{cases} \quad (240)$$

The correlations presented above are expected to be very imprecise for cases with $C_L < 1/3$ and $C_L > 11$. In the cases with small C_L the model is not applicable because the aspect ratio of the room is too extreme (large cross section to room length ratio) and the dominance of the shear layer momentum transfer in recirculation flow may not occur. The same reasoning applies to the other extreme case, large C_L , in this case due to the large room length.

The upper limit imposed on C_L already indicates that the model should not be applied when the length of expansion for the shear layer is very large. In addition to this length limitation, a limit should be imposed on A^* since it can be expected that for very large values of this parameter the momentum flux in the return flow becomes infinitesimal and therefore the present analysis is not applicable. After analyzing the cases in the CFD library as well as few exploratory runs for very large rooms it was decided to use the following criterion as the upper limit of application of the model:

$$\sqrt{L^* A^*} < 20 \quad (241)$$

This expression places an upper limit on room dimensions, scaled by inflow aperture length scale. One clear reason why this limit is needed is the fact that dissipation in the recirculation region is neglected, clearly as the room becomes larger dissipation effects increase, while the energy source remains constant (the inflow jet), an upper limit is needed. This limit ensures that one of the conditions to apply the model is present, the other being the need to have a flow that is dominated by forced convection.

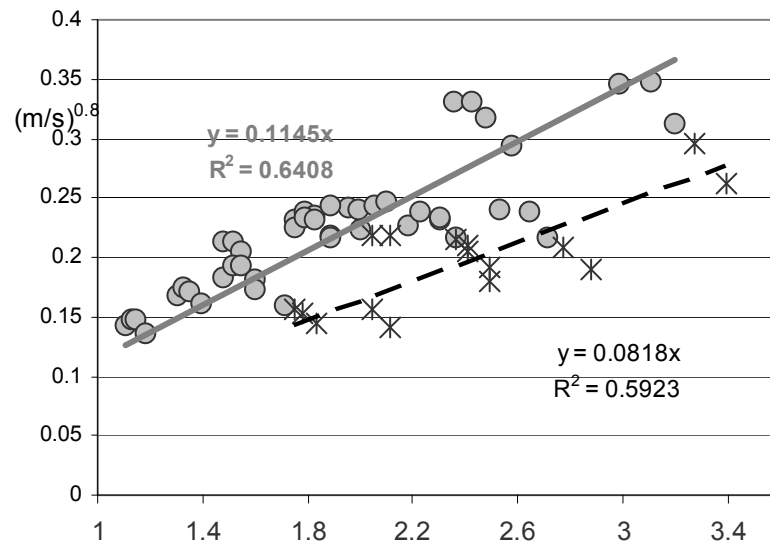


Figure 116. Results of the correlation for CV induced airflow velocity near the room surfaces.

Standard rooms ($C_L < 4$): gray line. Long rooms ($4 < C_L < 11$): dashed line. The units of the vertical axis are: $(\text{m/s})^{0.8}$ (the left hand side of expression (237)). On the horizontal axis the values of the coefficient with exponent 0.4 on the right hand side of expression (237).

Heat transfer in recirculating flows

In most cross ventilation flows with recirculations the mixing between recirculation and inflow jet is only partial. Figure 117 shows a schematic representation of the heat transfer process (showing a top view of one half of a symmetrical room). The flow is divided into three distinct streams with connected temperature variations:

- The main jet (labeled J in the figure).
- The part of the recirculation flow that exchanges heat with the jet (label R).
- The wall boundary layer part of the recirculation flow (label W).

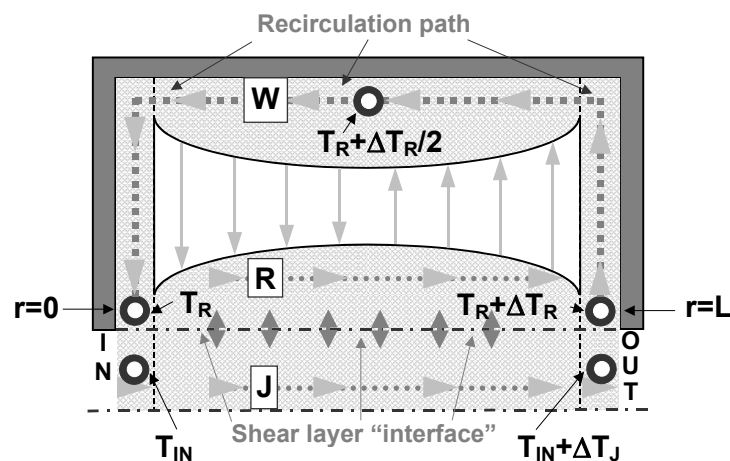


Figure 117. Top view of the flow structure in case R.

The light gray arrows show flow direction. The dark gray arrows show heat transfer in the shear layer. The recirculation region coordinate system is shown in the figure, with coordinate r varying between 0 and L .

Heat transfer in a shear layer

It is possible to use a formulation with a heat transfer area, a convection heat transfer coefficient and an average shear layer temperature difference to model the heat exchange between jet and recirculation flows:

$$Q_{y=0} = A_{SL} h_{SL} \Delta T_{SL}, \quad A_{SL} = L P, \quad h_{SL} = \frac{\rho C_p F}{C_D A_{IN} 4 \sigma \sqrt{\pi}} \quad (242)$$

Locally, the heat transfer process is driven by the variable, local, temperature difference:

$$\rho C_p F \frac{\partial T_J(x)}{\partial x} = -\frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) = \frac{P (T_J(x) - T_R(x))}{C_D A_{IN} 4 \sigma \sqrt{\pi}} \quad (243)$$

Predicting air temperature in the recirculation region of the flow

An accurate representation of system behavior must consider both the local heat transfer in the shear layer (varying along the x direction, driven by the temperature profiles $T_J(x)$ and $T_R(x)$) and the heat transfer in the wall boundary layers (with temperature $T_W(r)$). Both transfers can be modeled using differential equations, resulting in the following system of equations:

$$\left\{ \begin{array}{l} \rho C_p F R \frac{\partial T_W(r)}{\partial r} = \frac{A_S h_S}{L} (T_S - T_W(r)) \\ \rho C_p F R \frac{\partial T_R(x)}{\partial x} = \frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \\ \rho C_p F \frac{\partial T_J(x)}{\partial x} = -\frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \end{array} \right. \quad (244)$$

The relevant boundary conditions are:

$$T_W(L) = T_R(0), \quad T_W(0) = T_R(L), \quad T_J(0) = T_{IN}$$

The first equation in (244) models heat transfer in the wall layer. The second models the temperature variation of the portion of the recirculation flow that is in contact with the jet, it differs from the third equation only in a sign (the temperature variation in T_R is symmetric to the variation in T_J) and the flow rate ($F.R$ for the recirculation and F for the main jet). The temperature in the recirculation region is given by:

$$T_W(r) = T_S - (T_S - T_{IN}) \frac{e^{\frac{h_S A_S}{\rho C_p R F L} r} (e^{-\frac{A_{SL} h_{SL} (1+R)}{\rho C_p R F}} - 1)}{e^{\frac{A_{SL} h_{SL} (1+R)}{\rho C_p R F}} - e^{-\frac{h_S A_S}{\rho C_p R F} (1+R)} + R} \quad (245)$$

Heat transfer in recirculating flows with internal gains

Whereas surface heat transfer is an important component of room heat transfer, internal gains often dominate the room temperature field. Referring to the previously used subdivision of the room into jet and recirculation regions we can see that gains can occur exclusively in the jet region, exclusively in the recirculation region or in both regions.

Heat gains in the jet region

The effect of gains in the jet region on room air temperature distribution is simple to model and, within the first order accuracy goal, can be characterized as follows:

For gains occurring in the jet region (between the inlet and the room outlet) a change in inflow temperature is an adequate, conservative approximation. From energy conservation we conclude that the altered inflow temperature is given by:

$$T_{ING} = T_{IN} + \frac{G_J}{\rho C_p F} \quad (246)$$

Gains occurring in the jet region, close to the outlet can be ignored in a first order accuracy model.

Clearly the two approaches described above are only exact when: the gains occur at the inlet and perfectly mix with the inflow jet (for the first case), or whenever the gains occur very close to the outlet (in the second case). In all other situations, the first approach provides a conservative approximation. In the model implementation in EnergyPlus the user will not define the exact location of the heat gains within each region of the flow therefore, heat gains in the jet region, are inserted in two equal parts in two separate points of the jet flow (in the room entrance and before the exit, see points $r=0$ and $r=L_R$ in Figure 117).

Heat gains in the recirculation region

When heat gains occur in the recirculation region significant heat accumulation occurs due to the limited heat transfer ability of the shear layer. Due to this limited ability, the recirculation is partially isolated from the main jet flow and higher temperatures are generated in this region whenever heat gains are present. The concept of a global heat transfer coefficient is not applicable.

In the case of a flow in a room with adiabatic surfaces and heat gains in the recirculation region the indoor air temperatures can be predicted using the solution to the following system of equations:

$$\left\{ \begin{array}{l} \rho C_p F R (T_R(0) - T_R(L)) = G_R \\ \rho C_p F R \frac{\partial T_R(x)}{\partial x} = \frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \\ \rho C_p F \frac{\partial T_J(x)}{\partial x} = -\frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \end{array} \right. \quad (247)$$

This system of equations differs from system F16 in several ways. Because there is no wall heat transfer there is no need to split the recirculation flow in two parts (R and W). The temperature increase in the recirculation flow is simply defined by the internal gains (first

equation in (246)). The second and third equations in (247) model shear layer heat transfer are similar in the two cases ((244) and (247)).

The recirculation flow temperature at $x=0$ is given by:

$$T_R(0) = T_{IN} + \left(\frac{1}{e^{\frac{A_{SL} h_{SL} (1+R)}{\rho C_p F R}} - 1} + 1 \right) \frac{G_R (1+R)}{\rho C_p F R} \quad (248)$$

The temperature in the recirculation is inversely proportional to the recirculation flow ratio (R) and the shear layer area (measured by the product: $A_{SL} h_{SL}$). As both of these parameters increase this expression becomes similar to (246). The recirculation flow temperature at $x=L$ is given by:

$$T_R(L) = T_R(0) + \Delta T_R \quad (249)$$

The temperature variations in the recirculation flow is given by:

$$\Delta T_R = \frac{G_R}{\rho C_p F R} \quad (250)$$

Combined effects of surface heat transfer and internal gains in the recirculation region

In this case, the following system of equations must be solved:

$$\left\{ \begin{array}{l} \rho C_p F R \frac{\partial T_W(r)}{\partial r} = -\frac{A_S h_S}{L} (T_S - T_W(r)) - \frac{G_R}{L_R} \\ \rho C_p F R \frac{\partial T_R(x)}{\partial x} = \frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \\ \rho C_p F \frac{\partial T_J(x)}{\partial x} = -\frac{A_{SL} h_{SL}}{L} (T_J(x) - T_R(x)) \end{array} \right. \quad (251)$$

With the boundary conditions: $T_W(L) = T_R(0)$, $T_W(0) = T_R(L)$, $T_J(0) = T_{IN}$

For simplicity the heat gains are considered to be evenly distributed along the recirculation path. The temperature variation in the recirculation region is given by:

$$T_W(r) = T_S + \frac{G_R}{A_S h_S} - \frac{e^{\frac{h_S A_S r}{\rho C_p R F L_R}} \left(e^{\frac{A_{SL} h_{SL} (1+R)}{\rho C_p R F}} - 1 \right) (G_R + A_S h_S (T_S + T_{IN}))}{e^{\frac{A_{SL} h_{SL} (1+R)}{\rho C_p R F}} + R - e^{\frac{h_S A_S}{\rho C_p R F}} (1+R)} \quad (252)$$

Limitations of the model

A- The model should be used only in rectangular rooms with flat surfaces. Let us consider the example of a triangular room, in this case the variable cross section means that the flow may

begin with no recirculations and then become attached (or the contrary). In this case the room length perpendicular to the CV direction is not representative of the length of development for the shear layer driving the flow (see expression (233)). In light of the first order precision of the model variations of $\pm 10\%$ in the room cross section are acceptable.

B- Air must enter the room through one “dominant” aperture. Although the model presented above could be extended to more than one inflow aperture a single dominant inflow aperture restriction must be imposed for now. An aperture is considered dominant whenever it introduces in the room more than 90% of the air inflow. In addition the average inflow velocity through that aperture must be higher than 0.2m/s, so as to ensure a clear CV airflow (still additional criteria must be met, see below). As discussed above, the effects of variations in outlet geometry are neglected. Still, the model will be more precise for rooms where the outlet is in the opposing surface of the inlet.

C- The room and inlet dimensions must be such that the value of C_L in the right hand side of Equation (232) must be bigger than 1/3 and smaller than 11 and criterion (241) is met.

D- The temperature variation between inlet and outlet must be smaller than 1.5 °C. Heat gains can affect the CV airflow pattern and, whenever the heat gains are very high the flow structure can change and the modeling approach/assumption of a flow dominated by horizontal movement can become inadequate. Figure 118 shows a schematic depiction of the flow pattern that forms the base of the model. The recirculation flow follows an horizontal path exchanging heat and momentum with the main jet flow. As buoyancy effects become stronger the flow starts to resemble case shown in Figure 119. In this case the recirculation not only exchanges with the main jet flow but is also able to reject heat directly by composing part of the outflow.

Clearly, the amount of buoyancy that can be inserted in the flow with out affecting the CV dominated flow pattern is proportional to the inflow momentum flux (and consequently airflow rate). One straightforward way to scale the influence of buoyancy is to check the airflow temperature increase as the air flows across the room. The upper limit proposed for this temperature increase is 1.5°C; whenever this criterion is exceeded it is more appropriate to model the flow as mixed.

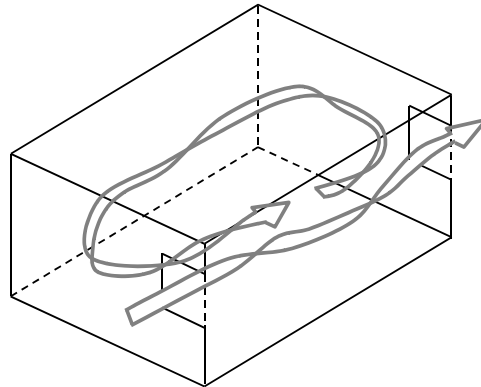


Figure 118. Schematic View - Room airflow dominated by jet

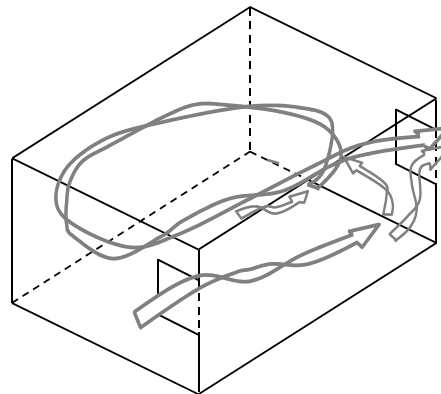


Figure 119. Schematic - Room airflow -- a combination of jet and recirculation flow

In Figure 118, the recirculation flow develops horizontally and the room airflow is dominated by the jet. In Figure 119, when buoyancy effects are important a more complex flow pattern develops. The recirculation flow moves vertically along its path and the outflow is a combination of jet and recirculation flow.

Implementation

The model predicts representative air temperatures and airflow velocities in two regions of the flow: jet and recirculations. In addition a set of flow pattern related parameters are also predicted such as type of flow pattern (with or without recirculations), room length along the cross ventilation direction, inflow aperture area, and, for flow with recirculations the ratio between recirculation and inflow rate.

Structure of the implementation

The calculation routine follows the following steps sequentially in every time step:

- 1) Identify the dominant aperture and obtain its area, width and inflow velocity.
- 2) Calculate room length and cross sectional area, check criteria (241).
- 3) Check whether recirculations are present in the flow, evaluate parameter C_L .
- 4) Predict relevant flow characteristics using expressions (234), (235), (236) and (240).
- 5) If recirculations are present and all CV flow criteria are met, calculate recirculation flow temperature and surface mixed convection heat transfer coefficients.
- 6) Calculate jet region temperatures.
- 7) If the air temperature increase is above 1.5°C or one of the CV flow criteria failed, perform a mixed flow calculation.

Outputs

Representative airflow temperatures for the jet and recirculation regions are obtained by taking the average between predicted temperatures at the beginning and end of each stream:

$$\text{CV Jet Region Temperature}[C] = \frac{T_J(0) + T_J(L)}{2} \quad (253)$$

$$\text{CV Recirculation Region Temperature}[C] = \frac{T_R(0) + T_R(L)}{2} \quad (254)$$

In both cases T_J and T_R are the solutions of the system of differential equations shown in Equation(s) (251).

The outputs *CV Jet Region Average Velocity*[m/s] and *CV Recirculation Region Average Velocity*[m/s] are predicted using expressions (234) and (235).

Nomenclature for Cross-Ventilation RoomAir Model

Table 38. Nomenclature for Cross-Ventilation RoomAir Model

Variable	Description
A^* :	room non dimensional cross-section area ratio.
A_{IN} :	Inlet area (m ²).
A_R :	Room cross section area (m ²)
A_S :	Total room surface area (m ²).
A_{SL} :	Shear layer heat transfer area (m ²)
C_L :	Non-dimensional room aspect ratio (expression 1.18, chapter 1).
C_n :	correlation constant for correlation index "n".
C_p :	Heat capacity of the air at constant pressure (J/(Kg.K)).
f :	Correlation function.
F :	Ventilation airflow rate (m ³ /s).
F_R :	flow rate in the recirculation region (m ³ /s).
H :	Room height (m).
h_F :	Surface averaged forced convection heat transfer coefficient.
h_{FS} :	Free stream heat transfer coefficient (W/(m ² K)).
h_N :	Surface averaged natural convection heat transfer coefficient.
h_S :	Local surface heat transfer coefficient (W/(m ² K)).
h_{SL} :	Shear layer heat transfer coefficient (W/(m ² K)).
L_R :	Room length along the cross ventilation direction (m).
L^* :	room non-dimensional length ratio.
M :	Momentum flux (N or J/m).
$Q_{Y=0}$:	Heat transfer in the shear layer mid plane (W).

R :	Non-dimensional recirculation flow ratio.
$T(x)$:	Room air temperature, averaged in the YZ plane ($^{\circ}\text{C}$).
T_{IN} :	Average inlet airflow temperature ($^{\circ}\text{C}$).
$T_J(x)$:	Temperature in the jet region, averaged in the YZ plane ($^{\circ}\text{C}$).
T_{OUT} :	Average temperature of the air leaving the room (K).
T_R :	Perfectly mixed room air temperature ($^{\circ}\text{C}$).
$T_R(x)$:	Temperature in the section of the recirculation flow that is parallel to the jet region, averaged in the YZ plane ($^{\circ}\text{C}$).
$T_W(r)$:	Temperature in the section of the recirculation that flows along the room walls, averaged in the YZ plane ($^{\circ}\text{C}$).
U_j :	average velocity in the room volume occupied by inlet jet flow (m/s).
U_R :	averaged velocity in a given region of the room that is being modeled (m/s).
U_{∞} :	Characteristic velocity of the forced flow that "drives" the forced convection boundary layer.
W :	Room width (m).
W_{IN} :	width of the inlet aperture (m).
x :	Coordinate along the length of the boundary layer.
x_{Outlet} :	Location of the outlet along the room length (m).
ΔT_j :	Temperature variation in the jet region (K).
ΔT_R :	Temperature variation in the recirculation region (K).
ΔT_{SL} :	Temperature variation in the shear layer region (K).
ρ :	Air density (Kg/m^3).
σ :	experimentally determined constant that scales mixing in the shear layer.

References

- Carrilho da Graca, G. 2003. Simplified models for heat transfer in rooms. Ph. D. Thesis, University of California, San Diego.
- Kalema, T., Haapala, T. 1995. Effect of interior heat transfer coefficients on thermal dynamics and energy consumption, *Energy and Buildings* 22 (2) pp. 101-113.
- Aynsley, R.M., Melbourne, W., Vickery, B.J. 1977. *Architectural Aerodynamics*. Applied Science London.
- Baturin, V. V. , Billington, N. S. 1972. *Fundamentals of Industrial Ventilation*, Franklin Book Company 1972, pp. 174-179.
- Neiswanger, L., Johnson, G.A., Carey, V.P. 1987. An experimental study of high Raleigh number mixed convection in a rectangular enclosure with restricted inlet and outlet openings. *Transactions of ASME*, Vol. 109, 446-453.
- Ohba, M., Irie, K., Kurabuchi, T. 2001. Study on airflow characteristics inside and outside a CVmodel, and ventilation flow rate using wind tunnel experiments. *Journal of Wind Engineering and Industrial Aerodynamics*, in press.

- Chandra, S., Kerestecioglu, A.A. 1984. Heat transfer in naturally ventilated rooms data from full-scale measurements, ASHRAE Transactions, Volume 90, part 1b 211-224.
- Altmayer, E.F., Gadgil, A.J., Bauman, F.S., Kammerud, R.C. Correlations for convective heat transfer from room surfaces. ASHRAE Transactions, No. 2764.
- Spitler, J. D., Pedersen, C.O., Fisher, D.E. 1991. Interior Convective Heat Transfer in Buildings with Large Ventilative Flow Rates. ASHRAE Transactions, Vol. 97, Pt.1, pp. 505-515.
- Awbi, H.B. & Hatton, A. 1999. Natural convection from heated room surfaces, Energy and Buildings, 30, 233-244.
- Siebers, D. L., Schwind, R. G., Moffat, R. J. 1983. Experimental Mixed Convection Heat Transfer From a Large Vertical Surface in a Horizontal Flow. SAND 83-8225, Sandia National Laboratories, Livermore CA.
- Awbi, H.B. & Hatton, A. 2000. Mixed convection from heated room surfaces, Energy and Buildings, 32, 153-166.
- Bejan, A. 1994. Convection Heat Transfer 2nd ed, Wiley, USA.

AirflowNetwork Model

Overview

The AirflowNetwork model provides the ability to simulate the performance of an air distribution system, including supply and return leaks, and calculate multizone airflows driven by outdoor wind and forced air during HVAC system operation. The pressure and airflow model described here was developed based on AIRNET (Walton 1989). This detailed model is used to simulate thermal conduction and air leakage losses for constant volume air distribution systems (e.g., in residential or light commercial buildings). This model replaces the previous EnergyPlus models for COMIS and Air Distribution System (ADS). The main difference is that the AirflowNetwork model adopts the previous COMIS model approach to introduce envelope leakage at a specific surface and modulate window and door openings at specified input and current outdoor conditions in multizone airflow calculations, instead of lumping all envelope leakage together as the previous ADS model did. In addition, the AirflowNetwork model calculates multizone airflows driven by wind when the HVAC system turns off. This capability is equivalent to the previous COMIS model. The multizone airflow calculations are now performed at the HVAC system time step instead of at the zone time step. This enhancement will allow future development of hybrid ventilation system models. In general, the AirflowNetwork model provides the combined capabilities of both the COMIS and ADS models available previously.

Model Description

The input object AirflowNetwork:SimulationControl provides access to the airflow network method, which consists of a set of nodes connected by airflow components through linkages. The objects AirflowNetwork:Multizone:Zone, AirflowNetwork:Multizone:ExternalNode, and AirflowNetwork:Distribution:Node represent airflow nodes. The objects AirflowNetwork:Multizone:Surface and AirflowNetwork:Distribution:Linkage represent airflow linkages. The other objects with a relationship between pressure and airflow represent airflow components.

The AirflowNetwork model consists of three sequential steps:

- Pressure and airflow calculations
- Node temperature and humidity calculations
- Sensible and latent load calculations

The pressure and airflow calculations determine pressure at each node and airflow through each linkage given wind pressures and forced airflows. Based on the airflow calculated for each linkage, the model then calculates node temperatures and humidity ratios given zone air

temperatures and zone humidity ratios. Using these node temperatures and humidity ratios, the sensible and latent loads from duct system conduction and leakage are summed for each zone. The sensible and latent loads obtained in this step are then used in the zone energy balance equations to predict HVAC system loads and to calculate the final zone air temperatures, humidity ratios, and pressures.

The present AirflowNetwork model may only be applied to a single heating and cooling system that uses a single air distribution system (a single AirLoopHVAC object). The model excludes the impact of the air and duct system thermal capacitance. The impact of thermal capacity will be addressed in future upgrades to this model.

Pressure and Airflow Calculations

The EnergyPlus airflow network consists of a set of nodes linked by airflow components. Therefore, it is a simplified airflow model, compared to detailed models such as those used in computational fluid dynamics (CFD) models. The node variable is pressure and the linkage variable is airflow rate. A brief description is presented below. A detailed description of the airflow network model may be found in the work of Walton (1989), Dols and Walton (2002), and Walton and Dols (2003).

Initialization

Newton's method is used to solve for node air pressures and it requires an initial set of values for the node pressures. There are two initialization methods available. The first is linear initialization and equivalent to Initialization flag = 0. These initial values may be obtained by including in each airflow component a linear approximation relating airflow to pressure drop:

$$\dot{m}_i = C_i \rho \left(\frac{\Delta P_i}{\mu} \right)$$

where

\dot{m}_i = Air mass flow rate at i-th linkage [kg/s]

C_i = Air mass flow coefficient [m³]

ΔP_i = Pressure difference across the i-th linkage [Pa]

μ = Air viscosity [Pa.s]

This initialization handles stack effects very well and tends to establish the proper direction for the airflows. The linear approximation is provided by the laminar regime.

The second initialization method assumes the initial pressures are zero and uses Newton's method directly.

Convergence criteria

Conservation of air mass flow rate at each linkage provides the convergence criterion. When the sum of mass flow rates in all the linkages approaches zero within the convergence tolerance, the solution has converged. The solution is assumed to have converged when the sum is less than the convergence value, in order to reduce the number of iterations and obtain sufficient accuracy. There are two convergence criteria used in the AirflowNetwork model: Relative airflow convergence tolerance and Absolute airflow convergence tolerance.

$$\text{Relative airflow tolerance} = \frac{\left| \sum \dot{m}_i \right|}{\sum \left| \dot{m}_i \right|}$$

$$\text{Absolute airflow tolerance} = \left| \sum \dot{m}_i \right|$$

The relative airflow tolerance is equivalent to the ratio of the absolute value of the sum of all network airflows to the sum of the network airflow magnitudes. The absolute airflow tolerance is the summation of the absolute value of all network airflows. The solution has converged when both of these convergence criteria have been met.

Linkage models

A linkage used in the AirflowNetwork model has two nodes, inlet and outlet, and is linked by a component which has a relationship between airflow and pressure. The pressure difference across each component in a linkage is assumed to be governed by Bernoulli's equation:

$$\Delta P = \left(P_n + \frac{\rho V_n^2}{2} \right) - \left(P_m + \frac{\rho V_m^2}{2} \right) + \rho g (z_n - z_m)$$

where

ΔP = Total pressure difference between nodes n and m [Pa]

P_n, P_m = Entry and exit static pressures [Pa]

V_n, V_m = Entry and exit airflow velocities [m/s]

ρ = Air density [kg/m³]

g = Acceleration due to gravity [9.81 m/s²]

z_n, z_m = Entry and exit elevations [m]

By rearranging terms and adding wind pressure impacts, the above equation may be rewritten in the format used by the airflow network model:

$$\Delta P = P_n - P_m + P_S + P_W$$

where

P_n, P_m = Total pressures at nodes n and m [Pa]

P_S = Pressure difference due to density and height differences [Pa]

P_W = Pressure difference due to wind [Pa]

The Input Output Reference provides the relationship between airflow and pressure for the most of the components (Ref. AirflowNetwork Model). The relationship between airflow and pressure for the AirflowNetwork:Multizone:Component:DetailedOpening and AirflowNetwork:Multizone:Component:SimpleOpening objects are provided in detail in this reference. These components are equivalent to the obsolete COMIS AIR FLOW:OPENING object.

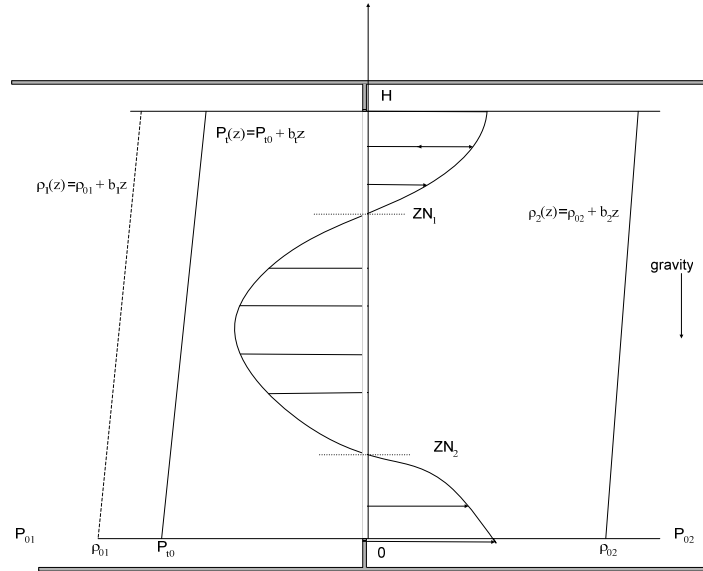


Figure 120. The general problem of gravitational flow through a vertical opening

The schematic drawing of a possible air flow pattern through a detailed vertical opening (AirflowNetwork:Multizone:Component:DetailedOpening) is shown in Figure 120. The equations used below are extracted from the COMIS Fundamentals manual (1990).

The air density is assumed to be a linear function of height:

$$\rho_i(z) = \rho_{0i} + b_i z$$

The pressure difference is assumed to be linear and simulate the effect of turbulence:

$$\Delta P_t = P_{t0} + b_t z$$

The reference pressures on each side are given at the bottom of the opening. By assuming the Bernoulli hypothesis on both sides, the pressure difference can be defined at any level of z as:

$$P_1(z) - P_2(z) = (P_{01} - P_{02}) - g \left[(\rho_{01} z + b_1 z^2 / 2) - (\rho_{02} z + b_2 z^2 / 2) \right] + (P_{t0} + b_t z)$$

The velocity at any level z is given by

$$v(z) = \sqrt{2 \frac{P_1(z) - P_2(z)}{\rho}}$$

The locations of the two possible neutral planes are given by an equilibrium in pressure which leads to a zero velocity point. By assuming the left terms in the equation above to be zero, one may have:

$$g(b_1 - b_2)z^2 / 2 + [g(\rho_{01} - \rho_{02}) - b_t]z + (-P_{01} + P_{02} - P_{t0}) = 0$$

This equation above can have two, one, or zero real solutions. The zero solution represents a one-way flow through the opening and may be expressed in the following equation:

$$\dot{m} = C_d \theta \int_{z=0}^{z=H} \rho v(z) W dz$$

The one real solution represents two-way (bi-directional) flow, which may be written in the following equations.

$$\dot{m}_{0,z1} = C_d \theta \int_{z=0}^{z=z1} \rho v(z) W dz$$

$$\dot{m}_{z1,H} = C_d \theta \int_{z=z1}^{z=H} \rho v(z) W dz$$

The two real solutions represent three-way flow, which may be written in the following equations.

$$\dot{m}_{z2,H} = C_d \theta \int_{z=z2}^{z=H} \rho v(z) W dz$$

$$\dot{m}_{z1,z2} = C_d \theta \int_{z=z1}^{z=z2} \rho v(z) W dz$$

$$\dot{m}_{z2,H} = C_d \theta \int_{z=z2}^{z=H} \rho v(z) W dz$$

where

C_d = discharge coefficient [dimensionless]

θ = Area reduction factor [dimensionless]

W = Opening width [m]

The discharge coefficient, opening width, opening height, and start height factor are modulated based on opening factors. A detailed description of opening factor calculations may be found in the Input Output Reference (Ref. AirflowNetwork:Multizone:Zone, AirflowNetwork:Multizone:Surface, and

AirflowNetwork:Multizone:Component:DetailedOpening).

The above calculation procedure is used for a normal rectangular window. For a horizontally pivoted rectangular window, the calculation procedure is slightly different. A schematic drawing of a horizontally-pivoted window is shown in Figure 121.

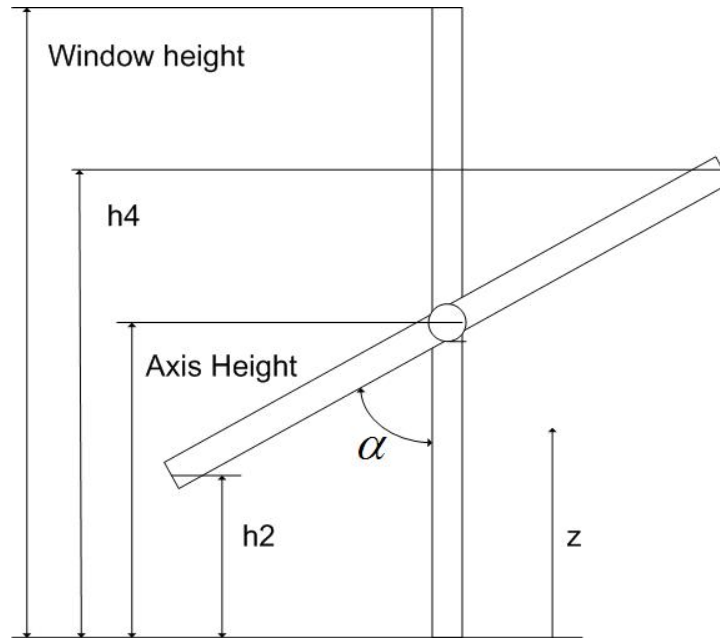


Figure 121. Schematic drawing of a horizontally-pivoted window

The opening angle α ($0-90^\circ$) is linearly proportional to the window opening factor ($0-1.0$). An opening factor of 1.0 is equal to an opening angle of 90° . The heights in the pivoted area are expressed as:

$$h2 = AxisHeight(1 - \cos(\alpha))$$

$$h4 = AxisHeight + (WindowHeight - AxisHeight) \cos(\alpha)$$

When $z < h2$ or $z > h4$, where z is the distance from the bottom of the window, the integration procedure is the same as the procedure for a normal rectangular window. When $h2 < z < h4$, the window width W in the above equations is modified as:

$$W_{pivot} = \sqrt{\frac{1}{\frac{1}{W^2} + \frac{1}{(2(AxisHeight - z) \tan(\alpha))^2}}}$$

The mass flow rate in the pivoted area becomes:

$$\dot{m}_{pivot} = C_d \theta \int_{z=h2}^{z=h4} \rho v(z) W_{pivot} dz$$

It should be pointed out that the discharge coefficient is modulated based on opening factors, while opening width, opening height, and start height factor do not apply for a horizontally-pivoted window. The actual window width and height are used to calculate airflows for a horizontally-pivoted window.

The schematic drawing of air flow patterns through a simple vertical opening (AirflowNetwork:Multizone:Component:SimpleOpening) is shown in Figure 122. The equations used below are available from Walton (1989).

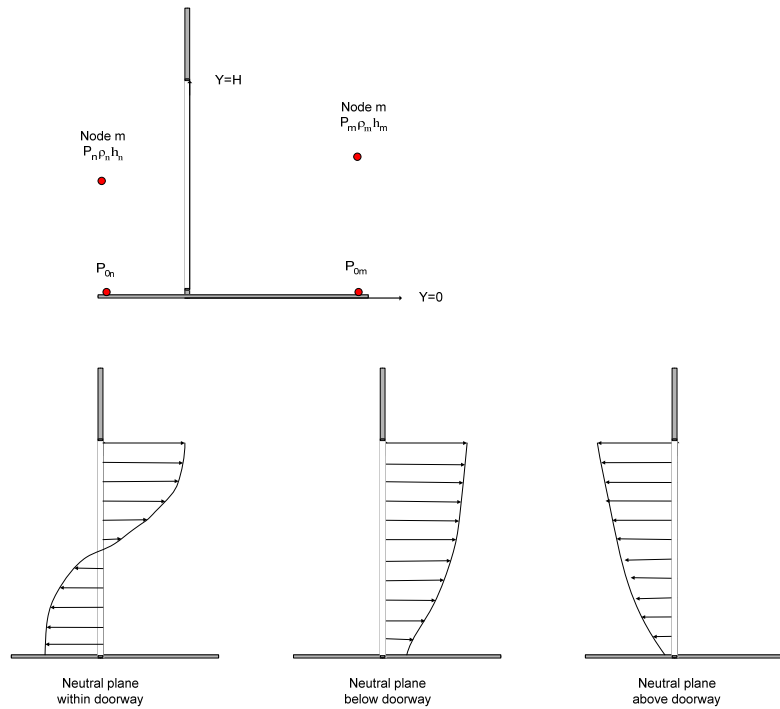


Figure 122. Schematic of large opening and associated three flow patterns

The air density for each node is assumed to be constant. The hydrostatic equation is used to relate pressures at various heights for each node:

$$P_n(y) = P_{0n} - \rho_n g y$$

$$P_m(y) = P_{0m} - \rho_m g y$$

where

P_{0n} , P_{0m} = pressure at nodes (zones) n and m at $y = 0$, the reference elevation of the opening [Pa]

ρ_n , ρ_m = air densities of zones n and m [kg/m^3]

P_n , P_m = reference pressures of zones n and m [Pa]

It is assumed that the velocity of the airflow as a function of height is given by the orifice equation (Brown and Solvason 1962):

$$v(y) = C_d \sqrt{2 \frac{P_n(y) - P_m(y)}{\rho}}$$

where

C_d = discharge coefficient [dimensionless]

ρ = density of the air going through the opening [kg/m^3]

The neutral height, Y , where the velocity of the air is zero, may be calculated in the following equation:

$$Y = \frac{P_{on} - P_{0m}}{g(\rho_n - \rho_m)} \quad \text{or} \quad \frac{P_{om} - P_{0n}}{g(\rho_m - \rho_n)}$$

When the neutral plane is within the opening (first pattern in Figure 122), two-way (bi-directional) flows occur. The total flow through a large opening is the sum of both flows.

$$\dot{m}_{0,Y} = C_d \theta \int_{y=0}^{y=Y} \rho v(y) W dy$$

$$\dot{m}_{Y,H} = C_d \theta \int_{y=Y}^{y=H} \rho v(y) W dy$$

When the neutral plane is below or above the large opening (second and third pattern in Figure 122), one-way flow occurs.

$$\dot{m} = C_d \theta \int_{y=0}^{y=H} \rho v(y) W dy$$

The opening width is modulated based on opening factors. A detailed description of opening factor calculations may be found in the Input Output Reference (Ref. AirflowNetwork:Multizone:Zone, AirflowNetwork:Multizone:Surface, and AirflowNetwork:Multizone:Component:DetailedOpening).

Wind pressure calculations

The wind pressure is determined by Bernoulli's equation, assuming no height change or pressure losses:

$$p_w = C_p \rho \frac{V_{ref}^2}{2}$$

where:

p_w = Wind surface pressure relative to static pressure in undisturbed flow [Pa]

ρ = Air density [kg/m³]

V_{ref} = Reference wind speed at local height [m/s]

C_p = Wind surface pressure coefficient [dimensionless]

V_{ref} may be expressed as (Ref, Local Wind Speed Calculations):

$$V_{ref} = V_{met} \left(\frac{\delta_{met}}{z_{met}} \right)^{\alpha_{met}} \left(\frac{z}{\delta} \right)^{\alpha}$$

C_p is a function of location on the building envelope and wind direction. When Wind Pressure Coefficient Type = "INPUT", the C_p values are explicitly defined in the input for AirflowNetwork:Multizone:Wind Pressure Coefficient Values. When Wind Pressure Coefficient Type = "AVERAGE-SURFACE CALCULATION" and the building shape is rectangular, the program uses the following equations to calculate wind pressure coefficient

(C_p) values for different wind directions. For a low rise building, the normalized surface pressure coefficient may be written as (Swami and Chandra 1988):

$$C_{p,n} = 0.6 * \ln \left[\begin{array}{l} 1.248 - 0.703 \sin(\alpha / 2) - 1.175 \sin^2(\alpha) + 0.131 \sin^3(2\alpha G) \\ + 0.769 \cos(\alpha / 2) + 0.07 G^2 \sin^2(\alpha / 2) + 0.717 \cos^2(\alpha / 2) \end{array} \right]$$

where

$C_{p,n}$ = C_p value at a given angle between wind direction and the outward normal of the surface under consideration [dimensionless]

α = Angle between wind direction and outward normal of wall under consideration [deg]

G = Natural log of the ratio of the width of the wall under consideration to the width of the adjacent wall [dimensionless]

n = Index of incident angle at 30-degree increments

For walls of a high rise building, a two-dimensional array of surface-averaged wind pressure coefficients is generated based on wind incident angle and side ratio. The wind pressure coefficients are provided in 2001 ASHRAE Fundamentals Handbook, p. 16.5, Fig. 7, "Surface Averaged Wall Pressure Coefficients for Tall Buildings". The original work was performed by Atkins et al. (1979). The incident angle has an increment of 30 degrees. The side ratio values are 0.25, 1.0, and 4.0. For a given incident angle and building aspect ratio, the program uses linear interpolation to calculate the corresponding wind pressure coefficient $C_{p,n}$.

For the roof of a high rise building, a two-dimensional array of surface-averaged wind pressure coefficients is also generated based on wind incident angle and side ratio. The wind pressure coefficients are provided in 2001 ASHRAE Fundamentals Handbook, p. 16.6, Fig. 9, "Surface Averaged Roof Pressure Coefficients for Tall Buildings". The original work was performed by Holmes (1986). The incident angle has an increment of 30 degrees. The side ratio values are 0.25, 1.0, and 4.0. At a given wind incident angle and building aspect ratio, the program uses linear interpolation to calculate the corresponding wind pressure coefficient $C_{p,n}$.

The wind surface pressure at the given incident angle can be calculated by combining the above two equations:

$$P_{w,n} = C_{p,n} \rho \frac{V_{ref}^2}{2}$$

Solution method

Based on the relationship between airflow rate and pressure drop for each component, a system of equations for all components can be assembled together in an $n \times n$ square matrix, where n is the number of nodes. Newton's method is used to iteratively solve for the air pressure at each node. A new estimated vector for all node pressures, $\{P\}^*$, is computed from the current estimated vector of node pressures, $\{P\}$, by:

$$\{P\}^* = \{P\} - \{C\}$$

where the correction vector, $\{C\}$, is computed by the matrix relationship:

$$[J]\{C\} = \{B\}$$

{B} is a column vector with each component given by:

$$B_n = \sum_i \dot{m}_i$$

where n is the node number and i indicates all flow paths connecting node n to other nodes, and [J] is the square Jacobian matrix whose elements are given by:

$$J_{n,m} = \sum_i \frac{\partial \dot{m}}{\partial P_m}$$

Convergence acceleration

The convergence tolerance is used to check the sum of the mass flow rates by applying mass conservation. The convergence acceleration equation shown below is used to correct the node pressures to more rapidly obtain a solution. By assuming a constant ratio of correction values from one iteration to the next, the following method is applied:

$$P_n^* = P_n - C_n / (1 - r)$$

where

r = the ratio of C_n for the current iteration to its value for the previous iteration [dimensionless]

C_n = Correction value at the n_{th} node [Pa]

P_n = Estimated pressure at the n_{th} node [Pa]

P_n^* = Corrected pressure at the n_{th} node used in the next iteration [Pa]

This method is similar to a Steffensen iteration (Conte and de Boor 1972) which is used as a fixed-point iteration method for individual nonlinear equations.

The iteration correction method presented in the above equation gives a variable factor. When the solution is close to convergence, the solution method converges quadratically. By limiting cases where the value of r is less than some value, such as -0.5, the solution will not interfere with the rapid convergence. It has not been proven that the convergence acceleration equation will always lead to convergence, but it can be shown that it will not prevent convergence. Newton's method converges when the estimated solution values are within some distance, called the radius of convergence, or the correct solution. Applying the convergence acceleration equation when $-1 < r < 0$, will cause a smaller correction than Newton's method, which therefore, can not force the iterations outside the radius of convergence. When $r < -1$, the solution diverges in an oscillatory fashion. When $r > 1$, the solution also diverges, but in a nonoscillatory manner. For $0 < r < 1$, the solution is approached from one direction. In all three cases, the convergence acceleration equation applies as long as r is truly constant over several iterations. However, for the last case, this involves a true extrapolation of correction factor which is very sensitive to the accuracy of r . This is most extreme for the case of $r=1$, which would cause an infinite correction.

Node Temperature Calculations

A brief description of the air node temperature calculation is given below. A detailed description can be found in the work of Swami et al. (1992). The following equation is used to calculate temperature distribution across a duct element at the given airflow rate and inlet air temperature:

$$\dot{m} C_p \frac{dT}{dx} = UP(T_\infty - T)$$

where

C_p = Specific heat of duct wall [J/kg•K]

\dot{m} = Airflow rate [kg/s]

P = Perimeter of a duct element [m]

T = Temperature as a field variable [°C]

T_∞ = Temperature of air surrounding the duct element [°C]

U = Overall heat transfer coefficient [W/m²•K]

$$U = \frac{1}{\frac{1}{h_i} + \frac{1}{h_o} + \sum \frac{t_j}{k_j}}$$

h_i = Inside heat transfer coefficient [W/m²•K]

h_o = Outside heat transfer coefficient [W/m²•K]

t_j = Thickness at j-th layer [m]

k_j = Thermal conductivity at j-th layer [W/m•K]

The outlet air temperature at the end of the duct (x=L) is:

$$T_o = T_\infty + (T_i - T_\infty) * \exp\left(-\frac{UA}{\dot{m}C_p}\right)$$

where

T_i = Inlet air temperature [°C]

T_o = Outlet air temperature [°C]

T_∞ = Temperature of air surrounding the duct element [°C]

A = Surface area (Perimeter * Length) [m²]

The heat transfer by convection to ambient, Q , is:

$$Q = \dot{m} C_p (T_\infty - T_i) \left[1 - \exp\left(-\frac{UA}{\dot{m}C_p}\right) \right]$$

The outlet air temperature can be calculated using the above equation at the given inlet air temperature. Since the inlet temperature at one linkage is the outlet temperature for the connected linkage, the outlet air temperatures at all nodes are solved simultaneously. A square linear system assembled by the AirflowNetwork model is expressed below:

$$\{M\}[T] = [B]$$

where

- $\{M\}$ = Airflow matrix
 $[T]$ = Temperature vector
 $[B]$ = Given boundary conditions

The zone air temperatures and primary air loop component (fan and coils) outlet conditions are used as prescribed conditions in the AirflowNetwork model. In addition, the temperature difference across zone loop components (terminal units) is held constant during the calculations. For example, thermal zone temperatures calculated during the previous system time step are used as prescribed temperatures when calculating all other node temperatures. The zone air temperature is assumed constant (prescribed) throughout the AirflowNetwork iterative solution. The fan and coil outlet air temperatures, and terminal unit temperature differences are assumed constant within an AirflowNetwork iteration. The sensible heat gains calculated during the AirflowNetwork solution are then used to predict a new zone air temperature.

Node Humidity Ratio Calculations

A brief description of the air node humidity ratio calculation is given below. A detailed description can found in the work of Swami et al. (1992). The following equation is used to calculate humidity ratio distribution across a duct element at the given airflow rate and inlet air humidity ratio:

$$\dot{m} \frac{dW}{dx} = U_m P (W_\infty - W)$$

where

- \dot{m} = Airflow rate [kg/s]
 P = Perimeter of a duct element [m]
 W = Humidity ratio [kg/kg]
 W_∞ = Humidity ratio of air surrounding the duct element [kg/kg]
 U_m = Overall moisture transfer coefficient [kg/m²•s]

$$U_m = \frac{1}{\frac{1}{h_{m,i}} + \frac{1}{h_{m,o}} + \sum \frac{t_j}{D_j}}$$

- $h_{m,i}$ = Inside moisture transfer coefficient [kg/m²•s]
 $h_{m,o}$ = Outside moisture transfer coefficient [kg/m²•s]
 t_j = Thickness at j-th layer [m]
 D_j = Moisture diffusivity at j-th layer [kg/m•s]

The outlet air humidity ratio at the end of the duct ($x=L$) is:

$$W_o = W_\infty + (W_i - W_\infty) * \exp\left(-\frac{U_m A}{\dot{m}}\right)$$

where

W_i = Inlet air humidity ratio [kg/kg]

W_o = Outlet air humidity ratio [kg/kg]

A = Surface area (Perimeter * Length) [m²]

The moisture transfer by convection to ambient, Q_m , is

$$Q_m = \dot{m}(W_\infty - W_i) \left[1 - \exp\left(-\frac{U_m A}{\dot{m}}\right) \right]$$

The outlet air humidity ratio can be calculated using the above equation at the given inlet air humidity ratio. Since the inlet humidity ratio at one linkage is the outlet humidity ratio for the connected linkage, the outlet air humidity ratio at all nodes are solved simultaneously. A square linear system assembled by the AirflowNetwork model is expressed below:

$$\{M_m\}[W] = [B_m]$$

where

$\{M_m\}$ = Airflow matrix

$[W]$ = Humidity ratio vector

$[B_m]$ = Given boundary conditions

The zone air humidity ratios and primary air loop component (fan and coils) outlet conditions are used as prescribed conditions in the AirflowNetwork model. For example, thermal zone humidity ratios calculated during the previous system time step are used as prescribed humidity ratios when calculating all other node humidity ratios. The zone air humidity ratio is assumed constant (prescribed) throughout the AirflowNetwork iterative solution. The coil outlet air humidity ratio is assumed constant within an AirflowNetwork iteration. The latent heat gains calculated during the AirflowNetwork solution are then used to predict a new zone air humidity ratio.

Sensible and Latent Load Calculations

The zone sensible and latent loads calculated in the AirflowNetwork model consist of multizone, duct conduction and leakage. In addition, the impact of infiltration and mixing is accounted for in this calculation. The multizone load only includes incoming airflows from outside (infiltration) and other adjacent zones (mixing) with and without forced-air fan operation. It is divided into two terms: variable and constant. The constant term is the sum of the mass flow rate multiplied by the specific heat for both infiltration and mixing. The variable term includes the impact of zone and outdoor air temperature. Each of these terms is used in the zone energy balance equation. The sensible load items from the multizone load calculations may be written as follows:

$$MCP_{airflow} = \dot{m}_{inf} C_p + \sum \left(\dot{m}_{mix} C_p \right)$$

$$MCPT_{airflow} = \dot{m}_{inf} C_p T_{amb} + \sum \left(\dot{m}_{mix} C_p T_{zone} \right)$$

where

$MCP_{airflow}$ = Sum of air mass flow rate multiplied by specific heat for infiltration and mixing [W/K]

$MCPT_{airflow}$ = Sum of air mass flow rate multiplied by specific heat and temperature for infiltration and mixing [W]

\dot{m}_{inf} = Incoming air mass flow rate from outdoors [kg/s]

\dot{m}_{mix} = Incoming air mass flow rate from adjacent zones [kg/s]

T_{amb} = Outdoor air dry-bulb temperature [°C]

T_{zone} = Adjacent zone air temperature [°C]

The latent load items from multizone load calculations may be written as follows:

$$M_{airflow} = \dot{m}_{inf} + \sum \dot{m}_{mix}$$

$$MW_{airflow} = \dot{m}_{inf} W_{amb} + \sum \dot{m}_{mix} W_{zone}$$

where

$M_{airflow}$ = Sum of air mass flow rates for infiltration and mixing [kg/s]

$MW_{airflow}$ = Sum of air mass flow rate multiplied by humidity ratio for infiltration and mixing [kg/s]

\dot{m}_{inf} = Incoming air mass flow rate from outdoors [kg/s]

\dot{m}_{mix} = Incoming air mass flow rate from adjacent zones [kg/s]

W_{amb} = Outdoor air humidity ratio [kg/kg]

W_{zone} = Adjacent zone air humidity ratio [kg/kg]

The air distribution system (ADS) loads due to duct conduction and leakage depend on the air distribution system component (e.g., duct) location. The air distribution system sensible and latent loads are calculated for each zone as follows:

$$Q_{ADS,i} = \sum_j Q_{cond(i,j)} + \sum_j Q_{leak(i,j)}$$

$$Q_{ADS,m,i} = \sum_j Q_{cond,m(i,j)} + \sum_j Q_{leak,m(i,j)}$$

where

$Q_{ADS,i}$ = Total sensible load in the i-th zone due to ADS losses [W]

$Q_{cond(ij)}$ = Duct wall conduction loss at the j-th duct located in the i-th zone [W]

$Q_{leak(ij)}$ = Sensible supply leak loss at the j-th linkage located in the i-th zone [W]

$Q_{ADS,m,i}$ = Total latent load in the i-th zone due to ADS losses [kg/s]

$Q_{cond,m(ij)}$ = Duct wall vapor diffusion loss at the j-th duct located in the i-th zone [kg/s]

$Q_{leak,m(ij)}$ = Latent supply leak loss at the j-th linkage located in the i-th zone [kg/s]

Impacts of Supply Air Fan Control on Load: Cycling vs. Continuous

The AirflowNetwork model currently allows two types of constant volume fans: Fan:ConstantVolume and Fan:OnOff. The Fan:ConstantVolume object has only one type of supply air fan operation mode: continuous fan, cycling compressor (ContinuousFanWithCyclingCompressor). However, the Fan:OnOff has two types of supply air fan operation modes: cycling fan, cycling compressor (CyclingFanAndCompressor) or continuous fan, cycling compressor (ContinuousFanWithCyclingCompressor). The CyclingFanAndCompressor operation mode is frequently referred to as "AUTO fan", where the compressor(s) and supply air fan operate in unison to meet the zone heating or cooling load, and cycle off together when the heating or cooling load has been met. The ContinuousFanWithCyclingCompressor operation mode is often referred to as "fan ON", where the compressor(s) cycle on and off to meet the zone heating or cooling load but the supply air fan operates continuously regardless of compressor operation. The supply air fan operation mode is specified in an HVAC system object based on a given fan operation mode schedule (e.g., AirLoopHVAC:UnitaryHeatCool object).

The determination of the zone sensible and latent loads caused by multizone airflows and forced air distribution system operation is dependent on the supply air fan operation mode (see Sensible and Latent Load Calculations section above). The zone loads calculated by the AirflowNetwork model are added to zone sensible and latent balances in the ZonePredictorCorrector module to calculate zone air temperatures and humidity ratios (see Integration of the AirflowNetwork Model section below). For the case of continuous fan/cycling compressor, the zone loads during forced air distribution system operation are calculated with the system design air mass flow rate without modification, since the system air node conditions (temperature and humidity) reflect the average values for the time step considering the coil/fan on and off periods during the time step.

For the case of cycling fan/cycling compressor, where the forced air distribution system can operate for a portion of the simulation time step, the airflows are determined based on the air distribution system part-load ratio (ratio of the average air mass flow rate for the time step divided by the design air mass flow rate). The **airflows** for the AirflowNetwork:Distribution:Linkage objects are reported during the air distribution system on cycle, since no airflow is assumed during the system off cycle. The **airflows** for the AirflowNetwork:Multizone:Surface objects are weighted by the air distribution system part-load ratio. The **zone loads** are the sum of energy losses during both the air distribution system on and off cycle at each system time step. The energy losses when the air distribution system is on are calculated using the system "on" air flow rate multiplied by the air distribution system run time fraction. The energy losses when the air distribution system is off are obtained from the multizone airflow calculations (without forced air through the air distribution system) and these losses are multiplied by (1.0 - system run time fraction), assuming no airflows through the air distribution system when the fan is off. The formulas used to calculate airflows and zone loads are given below:

Airflow

$$\text{Airflow} = \text{Airflow during ADS on cycle} * \text{ADS Part-load ratio} + \text{Airflow during ADS off cycle} * (1.0 - \text{ADS Part-load ratio})$$

where ADS = Air Distribution System

Zone load

System run time fraction = Maximum run time fraction of coils and fans in the air distribution system

$$\text{Zone energy losses} = \text{Zone energy loss during ADS on cycle} * \text{System run time fraction} + \text{Zone energy loss during ADS off cycle} * (1.0 - \text{System run time fraction})$$

The calculation of loads due to multizone airflow, without forced air distribution system operation, is performed when the HVAC system is first simulated during a simulation time step (FirstHVACIteration = True). The calculation of loads due to air distribution system operation is performed on subsequent iterations within the same simulation time step when

the mass flow rate at the supply air fan inlet is greater than 0.0 (and FirstHVACIteration = False).

Integration of the AirflowNetwork Model

The loads calculated by the AirflowNetwork model are integrated into the EnergyPlus heat balance equation in a similar manner as described elsewhere in this document in the section "Basis for the Zone and System Integration". The mass flow rate summations and sensible and latent loads described in the previous section are included in the calculation of zone temperature and humidity ratio.

The revised zone temperature update equation becomes:

$$T_z^t = \frac{\sum_{i=1}^{N_{sl}} \dot{Q}_i + \sum_{i=1}^{N_{surfaces}} h_i A_i T_{si} + MCPT_{airflow} + \dot{m}_{sys} C_p T_{supply} + Q_{ADS,z} - \left(\frac{C_z}{\delta t}\right) \left(-3T_z^{t-\delta t} + \frac{3}{2}T_z^{t-2\delta t} - \frac{1}{3}T_z^{t-3\delta t}\right)}{\left(\frac{11}{6}\right)\frac{C_z}{\delta t} + \sum_{i=1}^{N_{surfaces}} h_i A + MCP_{airflow} + \dot{m}_{sys} C}$$

Where $MCPT_{airflow}$ is the sum of mass flow rate multiplied by specific heat and temperature for infiltration and mixing, $Q_{ADS,z}$ is the added total sensible load in the zone due to Air Distribution System losses, and $MCP_{airflow}$ is the sum of mass flow rate multiplied by specific heat for infiltration and mixing as calculated in the AirflowNetwork model described above.

The revised coefficient (B) used in the zone humidity ratio calculation is shown below:

$$B = \sum kg_{massSched\ Loads} + MW_{airflow} + \dot{m}_{sys,in} W_{sys} + \sum_{i=1}^{surfs} A_i h_{mi} \rho_{air,z} W_{surfs_i} + Q_{ADS,m,z}$$

Where $MW_{airflow}$ is the sum of mass flow rate multiplied by humidity ratio for infiltration and mixing and $Q_{ADS,m,z}$ is the added total latent (moisture) load in the zone due to Air Distribution System losses from the AirflowNetwork model described above. This coefficient is used in the prediction of moisture as described in the section "Moisture Predictor-Corrector" found elsewhere in this document.

Model Output

The available outputs from the AirflowNetwork model are described in the EnergyPlus Input Output Reference manual.

References

Atkins, R. E., J. A. Peterka, and J. E. Cermak. 1979. "Averaged pressure coefficients for rectangular buildings," Wind Engineering, Proceedings of the Fifth International Conference 7:369-80, Fort Collins, CO. Pergamon Press, NY.

COMIS Fundamentals. 1990. Edited by Helmut E. Feustel and Alison Rayner-Hooson, LBL-28560, Lawrence Berkeley Laboratory, Berkeley, CA

Conte, S. D. & de Boor, C. 1972. Elementary Numerical Analysis: an Algorithmic Approach, McGraw-Hill.

Dols, W. S. & G. N. Walton. 2002. "CONTAMW 2.0 User Manual," NISTIR 6921, National Institute of Standards and Technology, Gaithersburg, Maryland

Holmes, J. D. 1986. Wind Loads on low-rise buildings: The structural and environmental effects of wind on buildings and structures, Chapter 12, Faculty of Engineering, Monash University, Melbourne, Australia

Swami, M. V. and S. Chandra. 1988. Correlations for pressure distribution on buildings and calculation of natural-ventilation airflow, ASHRAE Transactions 94(1988) (Pt 1), pp. 243-266.

- Swami, M. V., L. Gu, & V. Vasanth. 1992. "Integration of Radon and Energy Models for Building," FSEC-CR-553-92, Florida Solar Energy Center, Cocoa, Florida
- Walton, G. N. 1989. "AIRNET – A Computer Program for Building Airflow Network Modeling," NISTIR 89-4072, National Institute of Standards and Technology, Gaithersburg, Maryland
- Walton, G. N. & W. S. Dols. 2003. "CONTAM 2.1 Supplemental User Guide and Program Documentation," NISTIR 7049, National Institute of Standards and Technology, Gaithersburg, Maryland

Simulation Models – Encyclopedic Reference

The following descriptions are grouped alphabetically (as much as possible) with some also having additional tags of “Heat Balance”, “HVAC”, and “Plant”. With the integrated solution, these designations signify where the effects of these models have their first impacts.

Air System Distribution Terminals

Constant Volume Single Duct Uncontrolled Air Terminal

The input object `AirTerminal:SingleDuct:Uncontrolled` provides a method for directly connecting a central air system to a thermal zone. Central system air is usually supplied to a zone through a terminal unit such as a single duct VAV reheat box. Sometimes, however, it is desirable to supply central system air directly to a zone without any zone level control or tempering. An example would be Furnace or Central DX equipment. `AirTerminal:SingleDuct:Uncontrolled` is the input object for a component used to pass supply air directly into a zone without any thermostatic control. This unit allows the program to know what zone this branch of the air system is attached to and a place to input the maximum air flow rate. It is typically used with an `AirLoopHVAC` running as a constant-volume, variable-temperature system.

The `AirTerminal:SingleDuct:Uncontrolled` object creates the capability of supplying central system air directly to a zone and only contains the zone inlet node. This node is both the zone inlet node and the outlet node of the `AirLoopHVAC:ZoneSplitter`. It can be thought of as a balancing damper in the duct branch going to the zone. This inlet flow can be controlled by an availability schedule. This can be thought of as a seasonal shut off of the balancing damper.

For the `AirTerminal:SingleDuct:Uncontrolled` objects to work correctly, it is important in any systems including them for the sum of the maximum zone air flow rates to be equal to the maximum central system flow rate. The zone maximum flow rates are specified in the direct air inputs. The central air system flow rate is specified in the `AirLoopHVAC` input and also in the air loop branch and central fan inputs.

Constant Volume Single Duct Reheat Air Terminal

The input object `AirTerminal:SingleDuct:ConstantVolume:Reheat` provides a model for single duct constant volume systems with reheat that satisfy the cooling load in a zone by changing the inlet air temperature with a reheat coil. The supply air temperature must be low enough to meet the cooling load in the zone having the greatest load. For zones with a smaller cooling load, a reheat coil is used to raise the temperature of the zone inlet air.

This object can be configured with a water, steam, electric or gas reheat coil. Operation is basically the same with all coil types. The coil is controlled to raise the zone supply air temperature (i.e., the Unit Air Outlet Node temperature) to match the zone load. If the coil is undersized, the zone setpoint temperature will not be maintained.

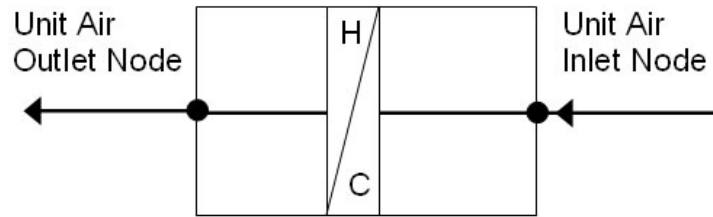


Figure 123. Schematic of AirTerminal:SingleDuct:ConstantVolume:Reheat Unit

Variable Air Volume Single Duct Reheat and No Reheat Air Terminals

The input objects AirTerminal:SingleDuct:VAV:Reheat and AirTerminal:SingleDuct:VAV:NoReheat provide models for single duct variable-air-volume (VAV) systems that control zone temperature primarily by varying the quantity of supply air rather than by varying the supply air temperature. The supply air temperature must be low enough to meet the cooling load in the zone having the greatest load when the zone terminal device is wide open. For zones with a smaller cooling load, the terminal device damper reduces the flow to match the zone setpoint. If the lower flow limit on the terminal device is reached and the load is not matched, the inlet air temperature can be moderated if the terminal device has a reheat coil. In that case both the quantity of air and its temperature entering the zone are varied to meet the load.

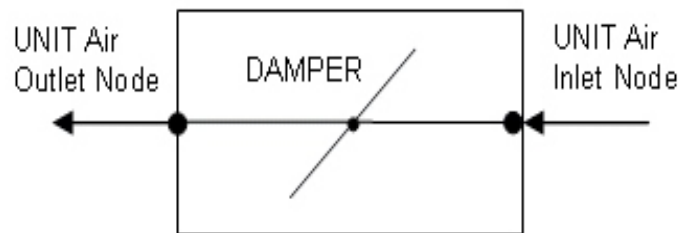


Figure 124. Schematic of AirTerminal:SingleDuct:VAV:NoReheat Unit

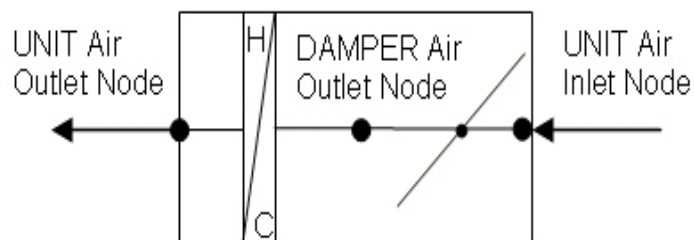


Figure 125. Schematic of AirTerminal:SingleDuct:VAV:Reheat Unit

The operation of the dampers and the control are described in the section AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat and AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat, which follows. The exception is that the section below describes how the air flow rate is varied for both cooling and heating. For the case of AirTerminal:SingleDuct:VAV:NoReheat and AirTerminal:SingleDuct:VAV:Reheat, air flow only varies during cooling operation and the air flow rate is set at the minimum value (minimum air flow fraction) when zone heating is required.

Variable Air Volume Heating and Cooling Single Duct Reheat and NoReheat Air Terminal

Overview

The input objects `AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat` and `AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat` provide models for variable-air-volume (VAV) terminal units are widely used in commercial and industrial applications. The VAV terminal units contain actuated dampers that vary the amount of central system air supplied to a zone. These terminal units may also contain a heating coil to trim the supply air temperature when overcooling is possible. The heating coil may also serve as the primary air heating source when the central system contains cooling-only equipment.

The VAV terminal units described here are used primarily with central air handling equipment with cooling and heating capability. The terminal unit dampers modulate in **both** cooling and heating mode to maintain the zone setpoint temperature(s). The central air handling equipment may be either variable air volume or constant volume where a bypass duct is used to shunt excess system air flow back to the inlet of the central air handler as terminal unit dampers modulate to satisfy the zone thermostat (i.e., `AirLoopHVAC:UnitaryHeatCool:VAVChangeoverBypass`).

Model Description

The no reheat version of the single duct VAV heat and cool terminal unit contains a single virtual damper assembly and requires minimal inputs. The reheat version contains both a virtual damper assembly and an air reheat coil. Multiple reheat coil types are available:

- `Coil:Heating:Water`
- `Coil:Heating:Electric`
- `Coil:Heating:Gas`
- `Coil:Heating:Steam`

Both units are simulated to provide an air flow rate sufficient to satisfy the thermostat request. The air flow rate is a function of the terminal unit's inlet air temperature and the load sensed by the thermostat. The output of the models are simply the damper position required to satisfy the zone's thermal load. Other information regarding terminal unit performance may be viewed using node report variables and heating coil report variables.

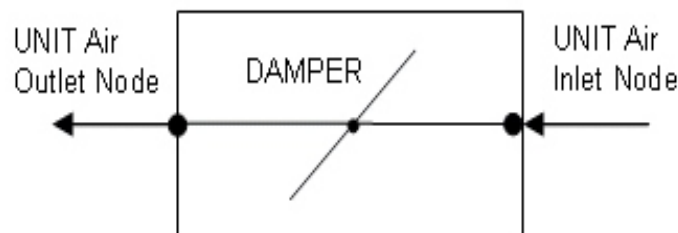


Figure 126. Schematic of `AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat` Unit

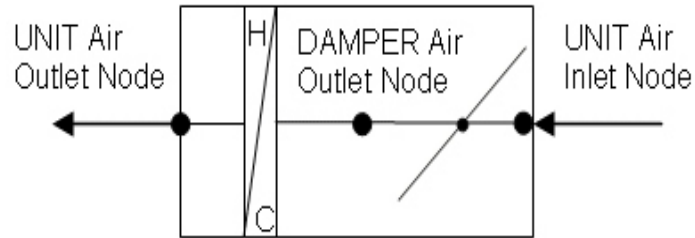


Figure 127. Schematic of AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat Unit

Terminal Unit Inputs

Both terminal unit types share several common input fields. A unique terminal unit name must be entered. A system availability schedule is also defined to allow operational control of the terminal unit. The user must then connect the unit to the air distribution system by defining the unit inlet and outlet node names. Design air flow rates are then specified: maximum total air flow rate (autosizable) and minimum air flow fraction.

The reheat version of this terminal unit requires additional information. The name and type of reheat coil and the damper air outlet node name (same as reheat coil inlet node name). Maximum and minimum water flow rates are entered when a water or steam heating coil is used, as well as a control node name for actuating water-side flow rates and a convergence tolerance for iteration control.

Simulation and Control

The simulation begins by determining the air mass flow rate required to satisfy the heating/cooling demand.

$$Cp_{zone} = PsyCpAirFnWTdb(\omega_{zone}, T_{zone})$$

$$Cp_{inlet} = PsyCpAirFnWTdb(\omega_{inlet}, T_{inlet})$$

$$\Delta CpT = (Cp_{inlet})(T_{inlet}) - (Cp_{zone})(T_{zone})$$

$$\dot{m} = \text{MIN} \left(\dot{m}_{max}, \text{MAX} \left(\dot{m}_{max} * \text{MinAirFlowFrac}, \frac{\dot{Q}_{zone}}{\Delta CpT} \right) \right)$$

where

Cp_{zone} = Specific heat of zone air, J/kg-K

Cp_{inlet} = Specific heat of terminal unit inlet air, J/kg-K

ω_{zone} = Zone air humidity ratio, kg/kg

T_{zone} = Zone air dry-bulb temperature, °C

ω_{inlet} = Terminal unit inlet air humidity ratio, kg/kg

T_{inlet} = Terminal unit inlet air dry-bulb temperature, °C

\dot{Q}_{zone} = Zone load, W (positive values denote heating, negative values denote cooling)

\dot{m} = Terminal unit air mass flow rate, kg/s

$PsyCpAirFnWTdb$ = Psychrometric function calculating air specific heat given air humidity ratio and dry-bulb temperature

$MinAirFlowFrac$ = User-specified zone minimum air flow fraction

\dot{m}_{max} = Terminal unit maximum air mass flow rate, kg/s

The damper position is then calculated as:

$$FRAC_{damper} = \dot{m} / \dot{m}_{max}$$

where

$FRAC_{damper}$ = Output variable 'Damper position', fraction of maximum flow

Simulation of the reheat coil occurs next when applicable. The heating demand required to maintain the thermostat heating setpoint temperature and the heating capacity of air flowing through the terminal unit are used to determine the amount of reheat required.

$$\dot{Q}_{reheat} = \dot{Q}_{heatSP} + \dot{m}(Cp_{zone})(T_{inlet} - T_{zone})$$

where

\dot{Q}_{reheat} = Reheat coil load, W (positive values denote heating)

\dot{Q}_{heatSP} = Load to heating setpoint temperature, W (positive values denote heating)

References

No specific references.

Constant Volume Single Duct Four Pipe Induction Air Terminal

The four pipe induction terminal unit (object name: AirTerminal:SingleDuct:ConstantVolume:FourPipeInduction) is a hybrid air-hydronic unit that supplies both centrally conditioned air and local hydronic heating/cooling to a zone. Centrally conditioned air is supplied to the terminal unit at high pressure and constant flow. The central (primary) air is discharged into the terminal unit through a nozzle, inducing a fixed flow of zone (secondary) through a hydronic heating/cooling coil. The primary and secondary air streams mix and are discharged to the zone. Hot or cold water flow through the coil is varied to meet the zone heating or cooling requirement.

Model

The four pipe induction terminal unit is modeled as a compound component consisting of three sub-components: a hot water coil, a chilled water coil and an air mixer. In terms of EnergyPlus objects these are *Coil:Heating:Water*, *Coil:Cooling:Water*, and *AirLoopHVAC:ZoneSplitter*. The terminal unit is a forward model: its inputs are defined by the state of its inlets: namely its 2 air streams – primary and secondary; and its two water inlets – hot and cold. The outputs of the model are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The terminal unit data and simulation are encapsulated in the module *HVACSingleDuctInduc*.

Inputs and Data

The user describes the terminal unit by inputting the name and type of the heating and cooling coils and the name of the zone mixer. The user must also specify the connectivity of the component by naming the inlet air and water nodes and the air outlet node. Finally maximum and fixed flow rates need to be specified (although these can be autosized): maximum and minimum hot and cold water volumetric flow rates and the total air volumetric flow rate (sum of primary and secondary flow rates). The relative convergence tolerances for the hot and cold water flow rates also need to be input (or allowed to default). Finally the induction ratio needs to be specified: this is defined as the ratio of the secondary air flow rate to the primary air flow rate. The relationship between the flow rates is:

$$\dot{m}_{air,tot} = \dot{m}_{air,pri} + \dot{m}_{air,sec}$$

$$\dot{m}_{air,sec} = R_{induc} \cdot \dot{m}_{air,pri}$$

so

$$\dot{m}_{air,pri} = \dot{m}_{air,tot} / (1 + R_{induc})$$

$$\dot{m}_{air,sec} = \dot{m}_{air,tot} \cdot R_{induc} / (1 + R_{induc})$$

where R_{induc} is the user-input induction ratio.

All input data for the four pipe induction terminal units is stored in the array *IndUnit*.

Calculation

Given the needed inputs, the output is calculated in subroutine *CalcFourPipeIndUnit*. The temperature, humidity ratio and flow rate of the primary and secondary air streams are taken from the inlet air nodes. The inlet hot and chilled water flow rates are passed in as parameters – temperatures are taken from the inlet water nodes. Then

The hot water coil is simulated (Call *SimulateWaterCoilComponents*);

The chilled water coil is simulated (Call *SimulateWaterCoilComponents*);

The two air streams are mixed (Call *SimAirMixer*).

Finally the load met by the terminal unit is calculated and passed back to the calling routine:

$$\dot{Q}_{out} = \dot{m}_{tot} \cdot c_{p,air} \cdot (T_{air,out} - T_{air,zone})$$

Note that data is never explicitly passed between the sub-components. This is all handled automatically by the node connections and the data stored on the nodes.

Simulation and Control

From the result of the zone simulation we have the heating/cooling demand on the terminal unit $\dot{Q}_{z,req}$. For a given hot and cold water flow *CalcFourPipeIndUnit* will give us the terminal unit heating/cooling output. We need to vary the hot or cold water flow to make the unit output match the demand. To do this we need to numerically invert *CalcFourPipeIndUnit*: given the output, we want one of the inputs – the hot or cold water flow. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function (the *residual* function) of a single independent variable. In this case the residual function calculates $(\dot{Q}_{z,req} - \dot{Q}_{out}) / \dot{Q}_{z,req}$. *SolveRegulaFalsi* varies either the hot water or cold water mass flow rate to zero the residual

Decide whether the unit is on or off. The unit is off if: a) it is scheduled off; b) the inlet air mass flow rate is zero; c) the zone thermostat is in the deadband; d) or the zone heating/cooling demand is very small.

If the unit is off, call *CalcFourPipeIndUnit* with the hot and cold water flow rates set to their minimum flows and return.

If the unit is on, check whether active heating or cooling by the hydronic coils is needed. Call *CalcFourPipeIndUnit* with minimum water flows to see what how much cooling (or possibly heating) the unit is doing with primary air only. The output for this case is \dot{Q}_{pri} .

If $\dot{Q}_{z,req} > \dot{Q}_{pri}$ we need active heating. Set the cold water flow rate to the minimum. Check that the terminal unit can meet the load by setting the hot water flow rate to the maximum and calling *CalcFourPipeIndUnit*. If the output is less than the zone demand we are done – all the outputs have been calculated. Otherwise call *SolveRegulaFalsi* to obtain the hot water flow rate that will make the unit output match the zone demand. This ends the unit simulation.

If $\dot{Q}_{z,req} < \dot{Q}_{pri}$ we need active cooling. We set the hot water flow rate to the minimum. We check whether the terminal unit can supply the needed output by setting the cold water flow rate to the maximum and calling *CalcFourPipeIndUnit*. If this maximum cooling output is not able to meet the zone cooling demand we are done. Otherwise call *SolveRegulaFalsi* to obtain the cold water flow rate that will make the unit output match the zone demand. This ends the unit simulation.

Note that the terminal unit output is never explicitly passed to another routine. Instead the output is saved as the outlet conditions on the terminal unit outlet air node. The node data is accessed when the terminal unit output is needed elsewhere in the program (in *SimZoneAirLoopEquipment* for instance).

References

No specific references.

Fan Powered Induction Series and Parallel Single Duct Reheat Air Terminal

Overview

The input objects *AirTerminal:SingleDuct:SeriesPIU:Reheat* and *AirTerminal:SingleDuct:ParallelPIU:Reheat* provide models for fan powered induction terminal units that occur in a variety of configurations. EnergyPlus models 2 types: *series* (sometimes called *constant*) and *parallel* (sometimes called *intermittent*). The series unit provides a constant flow of air to the zone (the fan is always on at a constant flow) with a variable proportion of primary and secondary air. The parallel unit has an intermittent fan: the fan is off at maximum cooling and does not switch on until primary air flow is significantly reduced from the maximum. Once on it provides a constant flow of secondary air. Both units induce air from the zone or plenum (secondary air) and mix it with centrally conditioned supply air (primary air). Both units are variable volume: the supply air flow rate is varied to match zone conditioning requirement.

Model

Both types of PIU are modeled as compound components. The series unit, in sequence from its inlet, consists of an air mixer, a constant volume fan, and a heating coil. In terms of EnergyPlus objects this is:

1. *AirLoopHVAC:ZoneMixer*
2. *Fan:ConstantVolume*
3. *Coil:Heating:Water*, *Coil:Heating:Electric*, *Coil:Heating:Gas* or *Coil:Heating:Steam*

The parallel unit contains a fan (in the secondary air stream, an air mixer, and a heating coil. In terms of EnergyPlus objects this is:

1. Fan:ConstantVolume
2. AirLoopHVAC:ZoneMixer
3. Coil:Heating:Water, Coil:Heating:Electric, Coil:Heating:Gas or Coil:Heating:Steam

Both units are forward models: their inputs are defined by the state of their inlets: namely the air inlet and the hot water inlet. The outputs of the models are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The PIUs data and simulation are encapsulated in the module *PoweredInductionUnits*. The main simulation routine for the units within the module are *CalcSeriesPIU* and *CalcParallelPIU*.

Inputs and Data

The user describes the PIU by inputting the names of the mixer, fan and heating coil sub-components plus the type of the heating coil sub-component. The user must connect the unit to the overall HVAC systems by naming various inlet and outlet nodes: the primary air inlet node, the secondary air inlet node, the unit air outlet node, and the hot water inlet node. One internal node name is needed: the coil air inlet node (same as fan outlet mode for series, mixer outlet node for parallel). Design flow rates need to be specified (although these can be autosized): maximum total air flow rate (series), maximum primary air flow rate, minimum primary air flow fraction, maximum secondary air flow rate (parallel), fan-on flow fraction (parallel), and maximum and minimum hot water flow rates. A convergence tolerance needs to be specified for units with hot water coils. There is an on/off availability schedule that need to be set.

All input data for the powered induction terminal units is stored in the array *PIU*.

Calculation

The calculation is performed by simulating the sub-components in the order given above. Most of the code is involved with figuring out what the various flow rates should be before simulating the subcomponents. These calculations are described below.

Simulation and Control

Series

From the result of the zone simulation we have the heating/cooling demand on the terminal unit \dot{Q}_{tot} . The subroutine *CalcSeriesPIU* needs to determine the flow rates that will allow the unit to meet this load. The first step is to determine the on/off state of the unit and determine the air flow rates.

- If the unit is scheduled off, the primary and secondary flow rates are set to zero.
- If there is no primary air flow (or less than .001 kg/s), the primary air flow is set to zero and the secondary air flow is set to the constant total air flow input by the user.
- If the zone temperature is in the deadband or the zone load is less than 1 watt or the zone needs heating, the primary air flow rate is set to the minimum flow rate specified by the input and the secondary air flow rate is set to the difference between the fixed total air flow rate and the primary air flow rate.
- Otherwise, the zone needs cooling and the unit is active.
 - We determine the fan temperature rise: the secondary air flow is set to the max total air flow, primary air flow to zero, and the mixer and fan are simulated. The fan delta T is the difference between the temperature at the fan's outlet node and inlet node.
 - We calculate the unit air outlet temperature needed to meet the zone cooling load: $T_{out} = T_z + \dot{Q}_{z,req} / (\dot{m}_{air,tot} \cdot c_{p,air,z})$
 - The temperature needed at the outlet of the mixer is then: $T_{mix} = T_{out} - \Delta T_{fan}$
 - We can then set the primary air flow rate.

- If $T_{mix} \leq T_{in,pri}$ then $\dot{m}_{pri} = \dot{m}_{pri,max}$
- else if $T_{mix} \geq T_{in,pri}$ and $T_{mix} \geq T_{in,sec}$ then $\dot{m}_{pri} = \dot{m}_{pri,min}$
- otherwise $\dot{m}_{pri} = \dot{m}_{air,tot} \cdot (T_{in,sec} - T_{mix}) / (T_{in,sec} - T_{in,pri})$, subject to the constraints that the flow rate can't be bigger than the max and min allowed.

The air flow rates are now determined and we can fire the air mixer (Call *SimAir-Mixer*) and fan (Call *SimulateFanComponents*) component simulations. Finally we simulate the heating coil:

- for a hot water coil, if the coil is off (no flow, deadband, no load) just fire the coil simulation once (Call *SimulateWaterCoilComponents*). Otherwise call *ControlCompOutput*, *ControlCompOutput* is a general component control routine. In this case it calls *SimulateWaterCoilComponents* repeatedly while varying the hot water flow rate and minimizing

$$(c_{p,air} \cdot \dot{m}_{air,tot} \cdot (T_{out} - T_z) - \dot{Q}_{z,req}) / \dot{Q}_{z,req}$$

to within the heating convergence tolerance.

- For gas, electric or steam coils, the required coil output is set to $\dot{Q}_{coil,req} = \dot{Q}_{z,req} - c_{p,air} \cdot \dot{m}_{air,coil} (T_{air,coil,in} - T_z)$. Then the coil simulation is fired (Call *SimulateHeatingCoilComponent* or *SimulateSteamCoilComponents*).

Finally the unit sensible output is calculated:

$$\dot{Q}_{sens,out} = \text{PsyHFnTdbW}(T_{air,out}, W_z) - \text{PsyHFnTdbW}(T_z, W_z)$$

where *PsyHFnTdb* is the EnergyPlus function giving enthalpy as a function of temperature and humidity ratio.

Parallel

From the result of the zone simulation we have the heating/cooling demand on the terminal unit \dot{Q}_{tot} . The subroutine *CalcParallelPIU* needs to determine the flow rates that will allow the unit to meet this load. The first step is to determine the on/off state of the unit and determine the air flow rates.

- If the unit is scheduled off, the primary and secondary flow rates are set to zero.
- If there is no primary air flow (or less than .001 kg/s), the primary air flow is set to zero and the secondary air flow is set to the max secondary air flow input by the user.
- If the zone temperature is in the deadband or the zone load is less than 1 watt or the zone needs heating, the primary air flow rate is set to the minimum flow rate specified by the input and the secondary air flow rate is set to max secondary air flow input by the user.
- Otherwise, the zone needs cooling and the unit is active.
 - We determine the fan temperature rise: the secondary air flow is set to the max secondary air flow, primary air flow to zero, and the fan and mixer are simulated. The fan delta T is defined as the difference between the temperature at the mixer outlet node and the fan inlet node.
 - Assuming that the fan is off, we calculate the primary air flow needed to meet the cooling demand.

$$\dot{m}_{pri} = \dot{Q}_{z,req} / (c_{p,air} \cdot (T_{in,pri} - T_z))$$

The flow rate is constrained to be between the min and max primary air flow rates. If this calculated primary flow rate is greater than the fan-on flow rate, the secondary flow rate is set to zero and we are done. Otherwise, the fan is on and we need to recalculate the primary air flow rate.

$$\dot{m}_{pri} = (\dot{Q}_{z,req} - c_{p,air} \cdot \dot{m}_{sec} \cdot (T_{in,sec} + \Delta T_{fan} - T_z)) / (c_{p,air} \cdot (T_{in,pri} - T_z))$$

The secondary flow rate is set to the user input fixed flow rate. The primary air flow rate is constrained to be between the min and max primary flow rated.

The air flow rates are now determined and we can fire the fan (Call *SimulateFanComponents*) and air mixer (Call *SimAirMixer*) component simulations. Finally we simulate the heating coil:

- for a hot water coil, if the coil is off (no flow, deadband, no load) just fire the coil simulation once (Call *SimulateWaterCoilComponents*). Otherwise call *ControlCompOutput*; *ControlCompOutput* is a general component control routine. In this case it calls *SimulateWaterCoilComponents* repeatedly while varying the hot water flow rate and minimizing

$$(c_{p,air} \cdot \dot{m}_{air,tot} \cdot (T_{out} - T_z) - \dot{Q}_{z,req}) / \dot{Q}_{z,req}$$

to within the heating convergence tolerance.

- For gas, electric or steam coils, the required coil output is set to $\dot{Q}_{coil,req} = \dot{Q}_{z,req} - c_{p,air} \cdot \dot{m}_{air,coil} (T_{air,coil,in} - T_z)$. Then the coil simulation is fired (Call *SimulateHeatingCoilComponent* or *SimulateSteamCoilComponents*).

Finally the unit sensible output is calculated:

$$\dot{Q}_{sens,out} = \text{PsyHFnTdbW}(T_{air,out}, W_z) - \text{PsyHFnTdbW}(T_z, W_z)$$

where *PsyHFnTdb* is the EnergyPlus function giving enthalpy as a function of temperature and humidity ratio.

References

No specific references.

Variable Air Volume Fan Powered Single Duct Air Terminal

Overview

The input object *AirTerminal:SingleDuct:VAV:Reheat:VariableSpeedFan* provides a model for variable speed (VS) fan VAV terminal unit that is a typical VAV reheat unit with the addition of a variable-speed blower fan to assist in moving supply air from the plenum to the conditioned zone. It is typically used with under-floor air distribution systems (UFAD) where the supply air is sent at low static pressure through an under-floor supply plenum. The fan has two maximum flow rate settings: one for cooling and one for heating. The cooling maximum is usually the actual fan maximum while the heating maximum is a lesser flow rate. The fan is upstream of the heating coil (this is a blow-through unit). The heating coil can be hot-water, electric or gas. Cooling control is obtained by varying the supply air flow rate from the cooling maximum to the minimum flow rate. Heating control is established by varying both the heating coil output (or hot water flow rate for hot water coils) and the supply air flow rate. Note that for this unit the minimum supply air flow rate is the flow rate when the fan is off.

Model

The VS fan VAV terminal unit is modeled as a compound component consisting of two sub-components: a fan and a heating coil. In terms of EnergyPlus objects the fan is a *Fan:VariableVolume* object and the heating coil is a *Coil:Heating:Water*, *Coil:Heating:Electric* or a *Coil:Heating:Gas*. The terminal unit is a forward model: its inputs are defined by the state of its inlets: namely its air inlet and its hot water inlet, if it has a hot water coil. The outputs of the model are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The terminal unit data and simulation are encapsulated in the module *SingleDuct*. The main simulation routine for the unit within the module is *SimVAVVS*.

Inputs and Data

The user describes the terminal unit by inputting the name and type of the heating coil and the name and type of the fan. The user must also specify the connectivity of the component by naming the inlet air node; the air node connecting the fan and heating coil (fan outlet, coil inlet); the unit air outlet node (same as the zone inlet node); and hot water inlet node (if any). Maximum flow rates need to be specified (although these can be autosized): maximum cooling and heating air flow rates and the maximum hot water flow rate (if there is a hot water coil). Minimum flow rates are specified by giving a minimum flow fraction for the air flow and a volumetric flow rate minimum for the hot water. For the units with hot water coils the relative convergence tolerance for the hot water flow rate also needs to be input (or allowed to default).

All input data for the VS fan VAV terminal units is stored in the array *Sys*.

Calculation

Given the needed inputs, the output is calculated in subroutine *CalcVAVVS*. The temperature and humidity of the supply air stream are taken from the inlet air node. The inlet air flow rate and the hot water flow rate are passed in as parameters. If the coil is electric or gas the coil heating power is passed instead of the hot water flow rate. Then

The fan is simulated (call *SimulateFanComponents*). If the fan is off the fan outlet conditions are set to the inlet conditions.

The heating coil is simulated (call *SimulateWaterCoilComponents* if the coil is a hot water coil; call *SimulateHeatingCoilComponents* if the coil is gas or electric).

Finally the sensible load met by the terminal unit is calculated and passed back to the calling routine:

$$\dot{Q}_{out} = \dot{m}_{air} \cdot c_{p,air} \cdot (T_{air,out} - T_{air,zone})$$

Note that data is never explicitly passed between the sub-components. This is all handled automatically by the node connections and the data stored on the nodes.

Simulation and Control

From the result of the zone simulation we have the heating/cooling demand on the terminal unit \dot{Q}_{tot} . For the given inlet conditions *CalcVAVVS* will give us the terminal unit heating/cooling output. We need to vary the air or hot water flow rate or the heating coil power (for gas or electric coils) to make the unit output match the demand. To do this we need to numerically invert *CalcVAVVS*: given the output, we want one of the inputs – the air or hot water flow rate or the heating coil power. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function (the *residual* function) of a single independent variable. In this case the residual function calculates $(\dot{Q}_{tot} - \dot{Q}_{out}) / \dot{Q}_{tot}$. *SolveRegulaFalsi* varies either the air mass flow rate, the hot water mass flow rate or the heating coil power to zero the residual.

The unit is simulated in the following sequence.

Decide whether the unit is on or off. The unit is off if: a) it is scheduled off; b) the inlet air mass flow rate is zero; or c) the zone thermostat is in the deadband

If the unit is off, call *CalcVAVVS* with flow rates set to their minimum flows and return.

If the unit is on, we need to establish the boundaries of 4 conditioning regions: a) active cooling with fan on; b) active heating with fan on; c) active heating with fan off; d) passive cooling with fan off. The heating/cooling demand will fall into one of these regions. Once the correct region is determined, we will know which model input to vary for control and thus how to invert the calculation.

To establish the boundaries of region a) we call *CalcVAVVS* twice: once with the supply air flow rate set to the cooling maximum, once with the cooling air flow rate set to the minimum. In both cases the heating coil output is at the minimum and the fan is on. Call the 2 cooling outputs $\dot{Q}_{cool,max,fanon}$ and $\dot{Q}_{cool,min,fanon}$. Remembering that EnergyPlus convention is that cooling loads are negative, then if $\dot{Q}_{tot} < \dot{Q}_{cool,max,fanon}$ the terminal unit can not meet the demand. Set the air mass flow rate to the cooling maximum and call *CalcVAVV* again. This concludes the simulation. If $\dot{Q}_{cool,max,fanon} < \dot{Q}_{tot} < \dot{Q}_{cool,min,fanon}$ the cooling demand is in the active cooling region. We hold the heating at the minimum, allow the supply air flow to vary between the cooling maximum and the minimum with the fan on, and call *SolveRegulaFalsi* to obtain the supply air flow rate that will produce the unit sensible cooling output that matches the demand. This concludes the simulation.

To establish the boundaries of region b) call *CalcVAVVS* twice: once with the supply air flow rate set to the heating maximum, once with the supply air flow rate set to the minimum. In both calls, if the heating coil is a hot water coil, the hot water flow rate is at the maximum. For electric and gas coils, the heating power is set to the maximum at maximum supply air flow and to zero at the minimum supply air flow. In both calls the fan is set to be on. Call the 2 heating outputs returned from the two calls to *CalcVAVVS* $\dot{Q}_{heat,max,fanon}$ and $\dot{Q}_{heat,min,fanon}$.

If $\dot{Q}_{heat,max,fanon} < \dot{Q}_{tot}$ the terminal unit can not meet the load. Set the air flow rate to the heating maximum and the hot water flow rate or heating coil power to the maximum and call *CalcVAVVS* again. This concludes the simulation for this case. If $\dot{Q}_{heat,min,fanon} < \dot{Q}_{tot} < \dot{Q}_{heat,max,fanon}$ the heating demand is in the active heating, fan on region.

For a hot water coil we call *SolveRegulaFalsi* with the supply air flow rate as the input that is varied and the hot water flow rate set to the maximum. For electric and gas coils the coil power and the supply air flow rate are both varied together from their minimum to maximum in a call to *SolveRegulaFalsi*. The call to *SolveRegulaFalsi* concludes the simulation for this case.

This region only applies to terminal units with a hot water coil. To establish the boundaries of region c) the fan is set to off, the supply air flow rate is set to minimum flow and *CalcVAVVS* is called twice: once with the hot water flow at maximum and once with the hot water flow at minimum. Call the two heating outputs $\dot{Q}_{heat,max,fanoff}$ and $\dot{Q}_{heat,min,fanoff}$. If \dot{Q}_{tot} is between these values, the supply air flow rate is set to its minimum, the fan is set to off, and in the call to *SolveRegulaFalsi* the hot water flow rate is varied to meet the load. This concludes the simulation for this case.

If the cooling demand does not fall into cases a) – c), the unit is assumed to be in the passive cooling state: heating is off or at the minimum, the fan is off, and the minimum supply air flow is delivered to the zone.

Note that the terminal unit output is never explicitly passed to another routine. Instead the output is saved as the outlet conditions on the terminal unit outlet air node. The node data is accessed when the terminal unit output is needed elsewhere in the program (in *SimZoneAirLoopEquipment* for instance).

References

No specific references.

Cooled Beam Unit (AirTerminal:SingleDuct:ConstantVolume:CooledBeam)

Cooled beam (frequently called chilled beams) systems are usually hybrid water – air systems. Commonly there is a constant flow, fixed temperature central forced air system for meeting ventilation and latent load requirements. Sometimes this forced air system's flow rate is varied according to ventilation demand; and of course its supply air temperature could be reset in various ways. Sensible cooling load is met by the cooled beam units; these are ceiling suspended units with cool water circulating through them. Some types of units are "passive" – they cool by radiation and natural convection. Other types of units are "active" and act as supply air terminal units with the supply air inducing room air over the beam cooling elements. These units cool almost entirely by convection. The DOE-2 model (upon which this model is based) is a convection only model – even the "passive" units are assumed to operate 100% convectively. The cooled beam elements act as an alternative to normal ceiling radiant cooling: they are not coupled to the building mass and they operate more in a convective mode, but they can, like radiant cooling, use fairly warm cooling water.

Heating is accomplished separately from the cooled beam system – usually baseboards are used on the building perimeter to meet heating loads.

Model

The chilled beam system is modeled as an EnergyPlus terminal unit. In terms of configuration within the overall HVAC system it will resemble a 4 pipe induction terminal unit. The user describes the system as a typical single duct constant volume system (with outside air mixer, fan, heating and cooling coils) on the air loop side, and with cooled beam terminal units on the zone equipment side.

The model is an empirical model developed at the equipment manufacturer Halton Oy. It consists of the following relationships.

$P_{beam} = A \bullet K \bullet \Delta T$	beam cooling output per unit length W/m
$K = \alpha \bullet \Delta T^{n1} \bullet v \rho^{n2} \bullet \omega^{n3}$	coil heat transfer coefficient W/(m ² K)
$v \rho = (q_{in}/\alpha_0) \bullet \rho_{air}$	room air mass flow rate across coil kg/(m ² s)
$q_{in} = K_1 \bullet \Delta T^n + K_{in} \bullet q_{pr}$	room air volumetric flow rate across coil per unit length m ³ /(s-m)

ΔT is the room air –water temperature difference (average water temperature is used) in degrees C.

ω is the water velocity in m/s.

q_{pr} is the supply air flow rate per unit length m³/(s-m)

The other symbols are the model parameters input by the user (see the IO Ref for descriptions).

Inputs and Data

The user describes the unit by inputting the name, referencing an availability schedule, and choosing a type (*active* or *passive*). The user must also specify the connectivity of the component by naming the inlet and outlet air and water nodes. The maximum water and fixed air flow rates need to be specified (although these can be autosized). The design inlet and outlet water temperatures are inputs. Generally the inlet water temperature is quite warm (15C is the default) and the temperature rise is small (design outlet water temperature defaults to 17C). Two key inputs are the number of beams (in the zone) and the beam length. It is generally wise to let these inputs autosize.

The remaining inputs are parameters specific to the product model. Good defaults are supplied and they should not be changed without information from the manufacturer.

Sizing

The Cooled Beam sizing calculations generally follow the procedures used for other terminal units (see *Loop Equipment Sizing*). One difference is that the Cooled Beams use the Cooled Beam inputs *Design Inlet Water Temperature* and *Design Outlet Water Temperature* for the chilled water ΔT rather than the ΔT from Plant Sizing. There are also two inputs unique to the Cooled Beam units that are autosized and will be described here.

The input *Number of individual beam units in the zone* is autosized by dividing the beam system zone design chilled water flow rate (either input by the user or autosized) by a nominal chilled water beam flow rate: 0.07 kg/s.

The input *Length of an individual beam unit* is autosized by using the model equations to calculate the length. The inputs to the equations are:

- 1) the design load per beam. The design load is calculated from the design water mass flow rate and the design water inlet and outlet temperatures. The design load is divided by the number of beams to obtain the design load per beam.
- 2) The design air supply air flow per beam – obtained by dividing the design supply air flow by the number of beams.
- 3) The design water flow per beam (m^3/s) – obtained by dividing the design water flow by the number of beams.
- 4) The design water velocity – obtained by dividing the design water flow per beam by the cross sectional inside area of a water tube ($\pi D^2/4$, where D is the input *Pipe inside diameter*).
- 5) Average air to water $\Delta T = T_{z, \text{cool peak}} - 0.5(T_{w, \text{des inlet}} + T_{w, \text{des outlet}})$; where $T_{z, \text{cool peak}}$ is the zone air temperature at the cooling peak and the $T_{w, \text{des}}$'s are the water design inlet and outlet temperatures.

With these inputs the model equations can be solve directly for beam length for passive cooled beams, and iteratively for active cooled beams.

Calculation

The subroutine *CalcCoolBeam* uses the model equations to calculate the cooling power $P_{\text{beams, out}}$ delivered to the room air and the outlet water temperature given the water flow rate (and the room air temperature and water inlet temperature). Since the model equations are nonlinear they must be solved iteratively. The subroutine does this by varying the outlet water temperature $T_{w, \text{out}}$ and calculating the water-side cooling power

$$P_w = q_{w, \text{beam}} \cdot c_{p, w} \cdot (T_{w, \text{out}} - T_{w, \text{in}})$$

and comparing it to the air-side cooling power

$$P_{\text{air}} = K \cdot A \cdot \Delta T \cdot L_{\text{beam}}$$

where $q_{w, \text{beam}}$ is the water mass flow rate (kg/s) per beam and L_{beam} is the length of a beam (m). When P_w and P_{air} match to within 0.1 W the subroutine terminates the iteration.

Simulation and Control

From the result of the zone simulation we have the heating/cooling demand for the zone equipment. For the cooling demand, we use the load to cooling set point P_c . Part of the demand may be satisfied by the zone supply air:

$$P_{\text{sup}} = Q_{\text{air}} \cdot (c_{p, \text{air, sys}} \cdot T_{\text{sys}} - c_{p, \text{air, z}} \cdot T_z)$$

The demand on the actual beams is then

$$P_{\text{beams,dem}} = P_c - P_{\text{sup}}$$

We want to know the chilled water flow rate that will give a beam cooling output of P_{beams} . To obtain this we need to numerically invert *CalcCoolBeam*: given its desired output, we want to know the chilled water flow rate. This numerical inversion is carried out by calling the subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function (the *residual* function) of a single independent variable. In this case the residual function is basically

$$(P_{\text{beams,out}} - P_{\text{beams,dem}}) / P_{\text{beams,out,max}}$$

SolveRegulaFalsi varies the cold water mass flow rate to zero the residual. The water inlet and outlet node flow rates are set to the flow rate found by *SolveRegulaFalsi* and the water outlet node temperature is set to the outlet water temperature from *SolveRegulaFalsi*.

References

Documentation Package Update #2 for DOE-2.1E, Version 107, page 3.152 describes the input and the model for the DOE-2 cooled beam model.

Constant Volume Dual Duct Air Terminal

Overview

The input object AirTerminal:DualDuct:ConstantVolume provides a model for dual duct constant-air-volume (DDCAV) systems that are typically used in special applications where precise temperature and humidity control are required and energy efficiency is not of primary concern. Thermal control for each zone is achieved by mixing air from the hot deck with air from the cold deck to achieve a supply air temperature that will exactly meet the zone load and satisfy the zone thermostat demand. Each zone has its own mixing box which is connected directly to the hot and cold decks. The mixing box dampers change the relative amount of hot and cold air that will be delivered (at a constant volumetric flow rate) to the zone.

Model Description

The DDCAV model will attempt to meet all of the thermostatic loads of a particular zone by explicitly calculating the hot and cold deck mass flow rates. For the energy and mass balance equations shown below, the zone load, temperatures, specific heats and the design mass flow rate are all known. These equations can then be solved directly for the hot deck and cold deck mass flow rates.

$$\dot{Q}_{\text{Zone}} = \dot{m}_c C_{p_c} T_c + \dot{m}_h C_{p_h} T_h - \dot{m}_d C_{p_z} T_z$$

$$\dot{m}_d = \dot{m}_c + \dot{m}_h$$

Where:

\dot{Q}_{Zone} = Zone load, W (positive=heating, negative=cooling)

C_{p_z} = Specific heat of zone air, J/kg-K

C_{p_c} = Specific heat of cold deck air, J/kg-K

C_{p_h} = Specific heat of hot deck air, J/kg-K

T_z = Zone air dry-bulb temperature, °C

T_c = Cold deck air dry-bulb temperature, °C

T_h = Hot deck air dry-bulb temperature, °C

\dot{m}_d = System design air mass flow rate through both heating or cooling duct, kg/s

\dot{m}_c = Cold deck air mass flow rate, kg/s

\dot{m}_h = Hot deck air mass flow rate, kg/s

Simulation and Control

The simulation first calculates the hot deck and cold deck air mass flow rates required to satisfy the heating/cooling demand on the zone. Once the individual flow rates have been calculated based on temperature control, the zone mixed air conditions are calculated assuming adiabatic mixing of the two air streams.

Variable Air Volume Dual Duct Air Terminal

Overview

The input object AirTerminal: DualDuct: VAV provides a model for dual duct variable-air-volume (DDVAV) systems that are typically used in special applications where both temperature and humidity control as well as energy efficiency are of primary concern. This system combines the advantages of the standard dual duct system for better thermal control with the possibility to reduce fan energy using a variable speed fan. The DDVAV terminal units contain actuated dampers that vary the amount of central system air supplied to a zone from both the hot and cold deck.

The DDVAV terminal units described here are used primarily with central air handling equipment with cooling and heating capability. The terminal unit dampers modulate the amount of cold air and hot air as well as the overall flow rate to maintain the zone setpoint temperature(s).

Model Description

The DDVAV model will attempt to meet all of the thermostatic loads of a particular zone by first sending air through either the heating duct or the cooling duct depending on whether there is a heating or cooling load (respectively). Flow rate through the opposite duct is kept at zero and flow through the active duct is varied between the minimum air flow rate (minimum zone air fraction multiplied by the maximum flow rate) and the maximum air flow rate. If the flow rate to meet the load through either the heating or cooling duct results in a flow outside these ranges, then air must be passed through the other duct as well to avoid over- or under-heating or –cooling. This is done using a conservation of energy and mass analysis of the terminal unit as well as the known inlet and necessary outlet condition to meet the thermal needs of the zone.

When there is no load on the zone, the system could either be scheduled off or be in a “no load” condition. If the system is scheduled off, the model keeps the flow rate at zero for both the heating and cooling duct. If in a no load condition, the system attempts to throttle back to the minimum possible flow and then find a balance between flow through the heating and cooling duct that will provide no net conditioning to the space. This means that the enthalpy of air delivered to the space must be equal to the enthalpy of the (average) air in the zone.

DDVAV Terminal Unit Inputs

Like other terminal units, the DDVAV terminal unit requires an availability schedule and inlet and outlet node designations. The DDVAV terminal unit, like the DD terminal unit, has two inlet nodes (one for the heating duct and one for the cooling duct) and one outlet node.

In addition, the DDVAV terminal unit also has a maximum flow rate and a minimum flow fraction like the VAV terminal unit. This allows the flow to be throttled back when it is

possible to provide the proper amount of conditioning with less flow. The maximum flow rate can be auto-sized, if desired.

Simulation and Control

The simulation begins by determining the air mass flow rate required to satisfy the heating/cooling demand using either the heating duct or cooling duct.

$$Cp_{zone} = PsyCpAirFnWTdb(\omega_{zone}, T_{zone})$$

$$Cp_{inlet} = PsyCpAirFnWTdb(\omega_{inlet}, T_{inlet})$$

$$\Delta CpT = (Cp_{inlet})(T_{inlet}) - (Cp_{zone})(T_{zone})$$

$$\dot{m} = \text{MIN} \left(\dot{m}_{max}, \text{MAX} \left(\dot{m}_{max} * \text{MinAirFlowFrac}, \frac{\dot{Q}_{zone}}{\Delta CpT} \right) \right)$$

where

Cp_{zone} = Specific heat of zone air, J/kg-K

Cp_{inlet} = Specific heat of terminal unit inlet air, J/kg-K

ω_{zone} = Zone air humidity ratio, kg/kg

T_{zone} = Zone air dry-bulb temperature, °C

ω_{inlet} = Terminal unit inlet air humidity ratio, kg/kg

T_{inlet} = Terminal unit inlet air dry-bulb temperature, °C

\dot{Q}_{zone} = Zone load, W (positive values denote heating, negative values denote cooling)

\dot{m} = Terminal unit air mass flow rate through either heating or cooling duct, kg/s

$PsyCpAirFnWTdb$ = Psychrometric function calculating air specific heat given air humidity ratio and dry-bulb temperature

MinAirFlowFrac = User-specified zone minimum air flow fraction

\dot{m}_{max} = Terminal unit maximum air mass flow rate, kg/s

The damper position is then calculated as:

$$FRAC_{damper} = \dot{m} / \dot{m}_{max}$$

where

$FRAC_{damper}$ = Output variable 'Damper position', fraction of maximum flow

If the flow rate was between the maximum flow rate and the minimum flow rate for the terminal unit, then no other calculations are needed. However, if the flow was reset to either the maximum or minimum flow rate, then flow through the active duct must be balanced by flow through the other duct to achieve the proper conditioning.

References

No specific references. Refer to the ASHRAE Handbook series for general information on different system types as needed.

Boilers

Simple Hot Water Boiler

The input object Boiler:HotWater provides a simple model for boilers that only requires the user to supply the nominal boiler capacity and thermal efficiency. An efficiency curve can also be used to more accurately represent the performance of non-electric boilers but is not considered a required input. The fuel type is input by the user for energy accounting purposes.

The model is based the following three equations

$$\text{OperatingPartLoadRatio} = \frac{\text{BoilerLoad}}{\text{BoilerNomCapacity}}$$

$$\text{TheoreticalFuelUse} = \frac{\text{BoilerLoad}}{\text{NominalThermalEfficiency}}$$

$$\text{FuelUsed} = \frac{\text{TheoreticalFuelUse}}{\text{BoilerEfficiencyCurveOutput}}$$

-or-

$$\text{FuelUsed} = \frac{\text{BoilerLoad}}{(\text{NominalThermalEfficiency})(\text{BoilerEfficiencyCurveOutput})}$$

The final equation above includes the impact of the optional boiler efficiency performance curve. To highlight the use of the normalized boiler efficiency curve, the fuel use equation is also shown in an expanded format. The normalized boiler efficiency curve represents the changes in the boiler's nominal thermal efficiency due to loading and changes in operating temperature. If the optional boiler efficiency curve is not used, the boiler's nominal thermal efficiency remains constant throughout the simulation (i.e., BoilerEfficiencyCurveOutput = 1).

When a boiler efficiency performance curve is used, any valid curve object with 1 or 2 independent variables may be used. The performance curves are accessed through EnergyPlus' built-in performance curve equation manager (curve objects). The linear, quadratic, and cubic curve types may be used when boiler efficiency is solely a function of boiler loading, or part-load ratio (PLR). These curve types are used when the boiler operates at the specified setpoint temperature throughout the simulation. Other curve types may be used when the boiler efficiency can be represented by both PLR and boiler operating temperature. Examples of valid single and dual independent variable equations are shown below. For all curve types, PLR is always the x independent variable. When using curve types with 2 independent variables, the boiler outlet water temperature (Toutlet) is always the y independent variable.

Single independent variable:

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) \quad \text{linear}$$

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) + C3(PLR)^2 \quad \text{quadratic}$$

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) + C3(PLR)^2 + C4(PLR)^3 \quad \text{cubic}$$

Dual independent variables:

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) + C3(PLR)^2 + (C4 + C5(PLR) + C6(PLR)^2)(T_{\text{outlet}}) \quad \text{quadraticlinear}$$

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) + C3(PLR)^2 + C4(T_{\text{outlet}}) + C5(T_{\text{outlet}})^2 + C6(PLR)(T_{\text{outlet}}) \quad \text{biquadratic}$$

$$\text{BoilerEfficiencyCurveOutput} = C1 + C2(PLR) + C3(PLR)^2 + C4(T_{\text{outlet}}) + C5(T_{\text{outlet}})^2 + C6(PLR)(T_{\text{outlet}}) + C7(PLR)^3 + C8(T_{\text{outlet}})^3 + C9(PLR)^2(T_{\text{outlet}}) + C10(PLR)(T_{\text{outlet}})^2 \quad \text{bicubic}$$

When a boiler efficiency curve is used, a constant efficiency boiler may be specified by setting $C1 = 1$ and all other coefficients to 0. A boiler with an efficiency proportional to part-load ratio or which has a non-linear relationship of efficiency with part-load ratio will typically set the coefficients of a linear, quadratic, or cubic curve to non-zero values. Using other curve types allows a more accurate simulation when boiler efficiency varies as a function of part-load ratio and as the boiler outlet water temperature changes over time due to loading or as changes occur in the water temperature setpoint.

The parasitic electric power is calculated based on the user-defined parasitic electric load and the operating part load ratio calculated above. The model assumes that this parasitic power does not contribute to heating the water.

$$P_{\text{parasitic}} = P_{\text{load}}(PLR)$$

where:

$$P_{\text{parasitic}} = \text{parasitic electric power (W), average for the simulation time step}$$

$$P_{\text{load}} = \text{parasitic electric load specified by the user (W)}$$

Steam Boiler

Description of Model

A steam boiler is the essential part of a building steam heating system and can be described as primary driver of the steam loop. It is the component that maintains the desired loop temperature.

The emphasis in EnergyPlus was laid on developing a building simulation model for steam boiler with ability to model detailed boiler performance without the cost of exhaustive user inputs to the boiler model. The Boiler:Steam input object is used on the plant loop supply side of EnergyPlus with the primary purpose of supplying steam to the heating coils, which constitute the demand side of the loop.

The steam boiler is a variable mass flow rate device. The mass flow rate of steam through the boiler is determined by the heating demand on the loop which in turn is determined by the equipment that is hooked to the demand side of the loop, namely the steam coils and hot

water heater. In short, the steam coil determines the mass flow rate of steam required for heating the zone to its required setpoint, the mixer sums up the total steam demanded by each of the individual coils and reports it to the boiler via the pump.

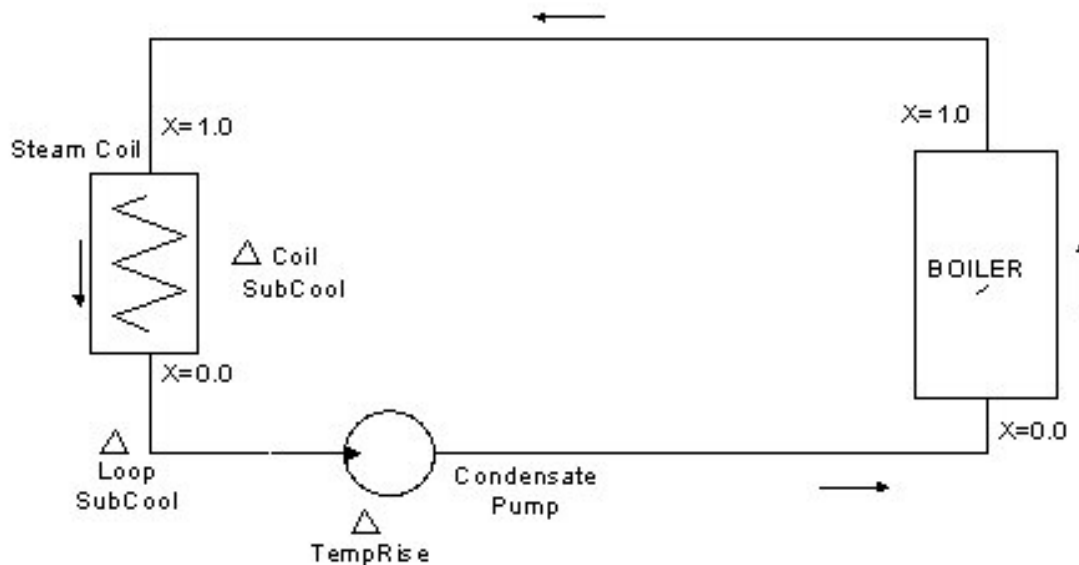


Figure 128. Schematic of Steam Boiler in the Steam loop

Figure 128 describes the rudimentary loop structure with steam flowing from coils to boiler. It is essential to mention that it is the coils that determine the mass of steam required and the boiler simply delivers the required mass flow at desired temperature provided it is adequately sized. The algorithm for determining the mass flow rate is structured on the demand side and the variable flow boiler has no role to play in determining the steam mass flow.

Figure 129 outlines the simple steam boiler model. Sub cooled water enters the variable flow boiler through the pump, the boiler inputs energy to water stream consuming fuel, boiler losses are accounted via boiler efficiency. The boiler delivers steam at a quality equal to 1.0 at saturated condition.

The advantage of steam heating systems over hot water is the high latent heat carrying capacity of steam, which reduces the mass flow rate of the fluid required. The amount of superheated and sub cooled heat transfer in Steam heating systems is negligible, latent heat transfer accounts for almost all of the heat exchange into the zones via steam to air heat exchangers.

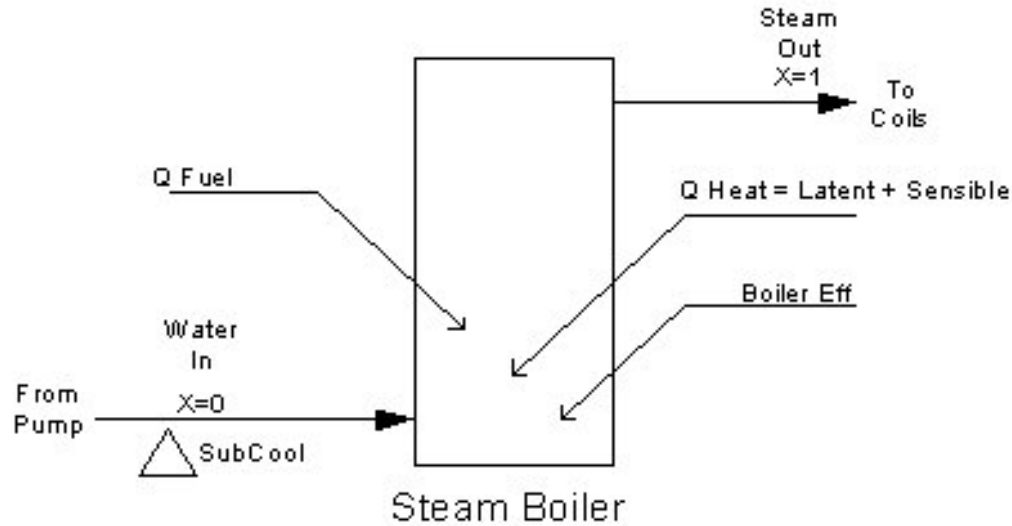


Figure 129. Schematic of Steam Boiler Operation

Boiler Load is a summation of sensible and latent heat addition to the water stream as described with the following equation. The mass flow rate through the boiler is known, while delta temp is the temperature difference between the boiler inlet and boiler outlet. Latent heat of steam is calculated at loop operating temperature.

$$Q_B = \dot{m} \times [(c_{p,w} \times \Delta T) + h_{fg}]$$

Theoretical fuel used is calculated with the following equation. Boiler efficiency is a user input and accounts for all the losses in the steam boiler.

$$F_t = \frac{Q_B}{\eta_B}$$

The operation part load ratio is calculated with the following equation. This is later used to calculate the actual fuel consumption, its ratio of boiler load to boiler nominal capacity.

$$O_{PLR} = \frac{Q_B}{Q_{B,N}}$$

The actual fuel consumption by the boiler is calculated as using the following equation, where C1, C2, and C3 are the Part Load Ratio coefficients.

$$F = \frac{F_t}{C_1 + C_2 \times O_{PLR} + C_3 \times O_{PLR}^2}$$

Essentially the boiler model provides a first order approximation of performance for fuel oil, gas, and electric boilers. Boiler performance is based on theoretical boiler efficiency and a single quadratic fuel use-part load ratio curve represented in the equation above. This single curve accounts for all combustion inefficiencies and stack losses.

The control algorithm for a steam boiler is an important issue. The user may want the boiler to be undersized and in such a case it will not be able to meet the demand side steam flow request. Subsequently the boiler load exceeds the boiler nominal capacity. The boiler operates at its nominal capacity but is unable to meet the plant heating demand. Pseudo code from EnergyPlus has been used to describe the control logic used in the steam boiler simulation.

*****PSEUDO CODE SECTION STARTS*****

At start of simulation an initial value of steam mass flow rate is calculated. This is required to start the flow of steam around the loop.

If (FirstTimeThrough) THEN

Calculate the boiler supply steam mass flow rate at start of simulation.

$$\dot{m}_s = \frac{Q_B}{h_{fg} + c_{p,w} \times \Delta T_{loop}}$$

ELSE ! Not first time through

Steam boiler calculations rely heavily on the variable \dot{m}_b , boiler mass flow rate. This variable \dot{m}_b is the assigned equal to mass flow at boiler inlet node for preliminary calculations.

$$\dot{m}_b = \dot{m}_{Inlet_Node}$$

Calculating the boiler delta temperature difference between the inlet and outlet nodes. This calculation is used to determine various boiler control situation.

$$\Delta T_{in_out} = T_{SP} - T_{in}$$

In case the temperature difference calculated with the previous equation is zero then the boiler just needs to supply latent heat to steam, else the boiler performs its normal load calculations by providing both sensible and latent heat to the inlet stream.

If ($\Delta T_{in_out} < 0$) THEN

$$Q_B = \dot{m}_b \times h_{fg}$$

ELSE

$$Q_B = \dot{m}_b \times (h_{fg} + c_{p,w} \times \Delta T_{in_out})$$

End If

Sometimes the boiler load Q_B is greater than the demand side requested load at the current time step, which may occur because the boiler inlet conditions is from previous time step. There is sudden fall in request of steam mass flow from the demand side. The boiler now recalculates its new mass flow and adjusts to these new conditions.

If ($Q_B > Q_{HeatingDemand}$) THEN

Boiler load is set equal to the new boiler heating demand and steam mass flow rate is recalculated.

$$Q_B = Q_{HeatingDemand}$$

$$\dot{m}_s = \frac{Q_B}{h_{fg} + c_{p,w} \times \Delta T_{loop}}$$

End If

In case the requested load exceeds the boiler nominal capacity, which is its maximum heating capacity. In this case the requested steam mass flow is not met and the zone is not heated adequately. This happens if the boiler is undersized. The steam mass flow rate is recalculated at nominal capacity.

If ($Q_B > Q_{Nominal_Capacity}$) *THEN*

Boiler load is set equal to boiler nominal capacity and steam mass flow rate recalculated.

$$Q_B = Q_{Nominal_Capacity}$$

$$\dot{m}_s = \frac{Q_B}{h_{fg} + c_{p,w} \times \Delta T_{loop}}$$

End If

End If

End If statement for the boiler load control algorithm. This algorithm determines all possible control conditions that might while simulating a system in EnergyPlus.

*****PSEUDO CODE SECTION ENDS*****

If the boiler operating pressure exceeds the maximum allowable boiler pressure, the simulation trips and outputs a warning regarding the same. This notifies the user about potential system pressure sizing problems.

Integration of the steam boiler simulation model in EnergyPlus required developing number of subroutines, which operate in sequence. These subroutines are designed to read inputs from the input file, initialize the variables used in the boiler simulation model, simulate the boiler performance, update the node connections, and report the required variables. In case the user has difficulty with boiler inputs, provisions have been made to auto size the boiler nominal capacity and maximum steam flow rate. These two values play an important role in sizing the boiler.

Model Assumptions

The EnergyPlus boiler model is “simple” in the sense that it requires the user to supply the theoretical boiler efficiency. The combustion process is not considered in the model. The model is independent of the fuel type, which is input by the user for energy accounting purposes only. This is an ideal model for Building Simulation Program such that it utilizes the desired amount of resources in terms of simulation run time, but successfully provides fairly good sizing parameters for an actual boiler.

It is assumed that the steam boiler operates to maintain a desired temperature, the temperature being saturation temperature of steam and corresponding to this saturation temperature there exist a single value of saturation pressure at which the loop operates. Hence the boiler could either be saturation pressure controlled or temperature controlled. Since users would have better idea of steam temperatures rather than pressure the boiler inputs are designed for temperature control.

Nomenclature for Steam Loop

Table 39. Steam Loop Nomenclature

Q_B	Boiler Heat Transfer. W.
$Q_{B,N}$	Boiler Nominal Capacity. W.
O_{PLR}	Boiler Operating Part Load Ratio.
ΔT_{sc}	Degree of subcooling in coil.
ΔT_{in_out}	Temperature difference across the steam boiler. °C.
ρ_w	Density of condensate entering the pump. Kg/m ³ .
Q_{Des}	Design Load on the steam coil. W.
$h_{f,n}$	Enthalpy of fluid at point n on the Ts diagram. J/kg.
P_{Frac}	Fraction of Pump Full Load Power. W.
$F_{m,f}$	Fractional Motor Power Lost to Fluid. W.
$Q_{a,l}$	Heating load on the Air Loop Steam Coil. W.
$Q_{z,c}$	Heating load on the Zone Steam Coil. W.
$h_{fg,T_{Loop}}$	Latent heat of steam at Loop operating Temperature. J/kg.
h_{fg}	Latent Heat of Steam. J/kg.
$Q_{L,H}$	Latent Heat Part of the Heating Coil Load. W.
ΔQ_{loss}	Loop losses in steam coil. W.
ΔT_{loop}	Loop Temperature Difference.
\dot{m}_a	Mass flow rate for steam coil Kg/s.
\dot{m}_{in}	Mass flow rate of steam entering the steam coil Kg/s.
$\dot{m}_{a,l}$	Mass flow rate of steam for Air loop steam coil Kg/s
$\dot{m}_{z,c}$	Mass flow rate of steam for zone steam coil Kg/s.
\dot{m}_s	Mass flow rate of steam. Kg/s.
\dot{m}_{loop}	Mass flow rate of steam for the steam loop. Kg/s.
\dot{m}	Mass of condensate entering the pump. Kg/s.
$\dot{m}_{a,max}$	Maximum allowed mass flow rate of air. Kg/s
$\dot{m}_{s,max}$	Maximum Mass flow rate of steam Kg/s

$\dot{m}_{B,Supply}$	Maximum steam mass flow rate supplied by boiler. Kg/s.
$\dot{V}_{w,max}$	Maximum Volume flow rate of condensate in pump. m ³ /s.
$\dot{V}_{w,loop}$	Maximum Volume flow rate of condensate in steam loop. m ³ /s.
$T_{a,in,min}$	Minimum inlet air temperature possible. °C.
P_n	Nominal Power Capacity for condensate pump. W.
P_{nom}	Nominal power of the pump. W.
H_n	Nominal Pump Head. M.
\dot{V}_{nom}	Nominal volume flow rate through the condensate pump. m ³ /s.
PLR	Part Load Ratio for condensate pump.
η_p	Pump efficiency.
η_m	Pump Motor Efficiency.
P	Pump Power. W.
$Q_{S,H}$	Sensible Heat Part of the Heating Coil Load. W.
T_{sp}	Setpoint Temperature of the zone. °C.
$T_{a,out,SP}$	Setpoint air outlet temperature for the steam coil. °C.
P_S	Shaft power of the pump. W.
$c_{p,a}$	Specific Heat Capacity for Air. J/Kg K.
$c_{p,w}$	Specific Heat Capacity for Water. J/Kg K.
η_B	Steam Boiler Efficiency.
$T_{a,in}$	Temperature of air entering the coil. °C.
T_a	Temperature of air entering the steam coil. °C.
$T_{a,out}$	Temperature of air leaving the coil. °C.
$T_{s,in}$	Temperature of steam entering the coil. °C.
F_t	Theoretical Fuel Consumption by the Steam Boiler. W.
$\dot{m}_{coils,R}$	Total Mass flow rate requested by all the steam coils. Kg/s.
\dot{V}	Volume of condensate entering the pump. m ³ /s.
$T_{w,out}$	Water outlet temperature from pump. °C.

References

- ASHRAE Handbook. 1996. HVAC Systems and Equipment, Air Conditioning and Heating Systems. Chapter 10, Steam Systems. pp. 10.1-10.16. 1996.
- BLAST 3.0 Users Manual*. 1999. Building Systems Laboratory. Urbana-Champaign: Building Systems Laboratory, Department of Mechanical and Industrial Engineering, University of Illinois.
- Chillar, R.J. 2005. "Development and Implementation of a Steam Loop In The Building Energy Simulation Program EnergyPlus," M.S. Thesis, Department of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign.
- TRNSYS 16 User Manual*. 2004. A Transient System Simulation Program. Solar Energy Laboratory, Madison. University of Wisconsin-Madison.
- El-Wakil, M. M. 1984. Power Plant Technology, McGraw Hill, New York, pp. 30-72.
- Babcock & Wilcox. 1978. Steam-Its Generation and Use, The Babcock & Wilcox Company, New York, Section I, II, IV, and VII.
- S.A. Klein. 2004. Engineering Equation Solver EES. University of Wisconsin Madison.

Chillers

Absorption Chiller

The input object Chiller:Absorption provides a model for absorption chillers that is an empirical model of a standard absorption refrigeration cycle. The condenser and evaporator are similar to that of a standard chiller, which are both water-to-water heat exchangers. The assembly of a generator and absorber provides the compression operation. Low-pressure vapor from the evaporator is absorbed by the liquid solution in the absorber. A pump receives low-pressure liquid from the absorber, elevates the pressure of the liquid, and delivers the liquid to the generator. In the generator, heat from a high temperature source (hot water or steam) drives off the vapor that has been absorbed by the solution. The liquid solution returns to the absorber through a throttling valve whose purpose is to provide a pressure drop to maintain the pressure difference between the generator and absorber. The heat supplied to the absorber can be waste heat from a diesel jacket, or the exhaust heat from diesel, gas, and steam turbines. For more information on absorption chillers, see the Input/Output Reference Document (Object: Chiller:Absorption).

The part-load ratio of the absorption chiller's evaporator is simply the actual cooling effect produced by the chiller divided by the maximum cooling effect available.

$$PLR = \frac{\dot{Q}_{evap}}{\dot{Q}_{evap, rated}}$$

where

PLR = part-load ratio of chiller evaporator

\dot{Q}_{evap} = chiller evaporator load [W]

$\dot{Q}_{evap, rated}$ = rated chiller evaporator capacity [W]

This absorption chiller model is based on a polynomial fit of absorber performance data. The Generator Heat Input Part Load Ratio Curve is a quadratic equation that determines the ratio of the generator heat input to the *demand* on the chiller's evaporator (Q_{evap}).

$$GeneratorHeatInputRatio = \frac{C1}{PLR} + C2 + C3(PLR)$$

The Pump Electric Use Part Load Ratio Curve is a quadratic equation that determines the ratio of the actual absorber pumping power to the nominal pumping power.

$$ElectricInputRatio = C1 + C2 * PLR + C3 * PLR^2$$

Thus, the coefficient sets establish the ratio of heat power in-to-cooling effect produced as a function of part load ratio. The ratio of heat-power-in to cooling-effect-produced is the inverse of the coefficient of performance.

If the operating part-load ratio is greater than the minimum part-load ratio, the chiller will run the entire time step and cycling will not occur (i.e. *CyclingFrac* = 1). If the operating part-load ratio is less than the minimum part-load ratio, the chiller will be on for a fraction of the time step equal to *CyclingFrac*. Steam (or hot water) and pump electrical energy use are also calculated using the chiller part-load cycling fraction.

$$CyclingFrac = MIN\left(1, \frac{PLR}{PLR_{min}}\right)$$

$$\dot{Q}_{generator} = GeneratorHeatInputRatio(\dot{Q}_{evap})(CyclingFrac)$$

$$\dot{Q}_{pump} = ElectricInputRatio(P_{pump})(CyclingFrac)$$

where

CyclingFrac = chiller part-load cycling fraction

PLR_{min} = chiller minimum part-load ratio

$\dot{Q}_{generator}$ = generator input power [W]

\dot{Q}_{pump} = absorption chiller pumping power [W]

The evaporator water mass flow rate is calculated based on the Chiller Flow Mode as follows.

Constant Flow Chillers:

$$\dot{m}_{evap} = \dot{m}_{evap,max}$$

Variable Flow Chillers:

$$\Delta T_{evap} = T_{evap,in} - T_{evap,SP}$$

$$\dot{m}_{evap} = \frac{\dot{Q}_{evap}}{C_{p,evap}(\Delta T_{evap})}$$

where

\dot{m}_{evap} = chiller evaporator water mass flow rate (kg/s)

- $\dot{m}_{evap,max}$ = chiller design evaporator water mass flow rate (kg/s)
 ΔT_{evap} = chiller evaporator water temperature difference (°C)
 $T_{evap,in}$ = chiller evaporator inlet water temperature (°C)
 $T_{evap,SP}$ = chiller evaporator outlet water setpoint temperature (°C)
 C_p = specific heat of water entering evaporator (J/kg•°C)

The evaporator outlet water temperature is then calculated based on the cooling effect produced and the evaporator entering water temperature.

$$T_{evap,out} = T_{evap,in} + \frac{\dot{Q}_{evap}}{C_{p,evap} (\dot{m}_{evap})}$$

where

- $T_{evap,out}$ = chiller evaporator outlet water temperature [°C]
 $T_{evap,in}$ = chiller evaporator inlet water temperature [°C]
 $C_{p,evap}$ = specific heat of chiller evaporator inlet water [J/kg/°C]
 \dot{m}_{evap} = chiller evaporator water mass flow rate [kg/s]

The condenser heat transfer and condenser leaving water temperature are also calculated.

$$\dot{Q}_{cond} = \dot{Q}_{evap} + \dot{Q}_{generator} + \dot{Q}_{pump}$$

$$T_{cond,out} = T_{cond,in} + \frac{\dot{Q}_{cond}}{C_{p,cond} (\dot{m}_{cond})}$$

where

- \dot{Q}_{cond} = chiller condenser heat transfer rate [W]
 $T_{cond,out}$ = chiller condenser outlet water temperature [°C]
 $T_{cond,in}$ = chiller condenser inlet water temperature [°C]
 $C_{p,cond}$ = specific heat of chiller condenser inlet water [J/kg/°C]
 \dot{m}_{cond} = chiller condenser water mass flow rate [kg/s]

The absorption chiller can model the impact of steam or hot water entering the generator, although the connection of the steam (hot water) nodes to a plant is not actually required. The calculations specific to the generator depend on the type of fluid used and are described here in further detail.

Steam Loop Calculations

When a steam loop is used and the inlet and outlet node names are specified (i.e. the nodes are connected to a steam loop), the generator outlet node steam mass flow rate and temperature are calculated based on the generator input power, latent heat of steam, the specific heat of water, and the amount of subcooling in the steam generator. The model assumes dry saturated steam enters the absorption chiller's generator and exits the generator as a subcooled liquid. The temperature leaving the generator is calculated based on the user entered amount of liquid subcooling in the generator. The effect of subcooling of the liquid (condensate) in the pipe returning to the boiler is not modeled.

$$\dot{m}_{steam} = \frac{\dot{Q}_{generator}}{h_{fg} + c_{p,water} \times \Delta T_{sc}}$$

$$T_{generator,out} = T_{generator,in} - \Delta T_{sc}$$

where

\dot{m}_{steam} = chiller steam mass flow rate [kg/s]

h_{fg} = latent heat of steam [J/kg]

$c_{p,water}$ = specific heat of saturated water in the generator [J/Kg °K]

ΔT_{sc} = amount of subcooling in steam generator [°C]

$T_{generator,out}$ = generator steam outlet node temperature [°C]

$T_{generator,in}$ = generator steam inlet node temperature [°C]

Hot Water Loop Calculations

When a hot water loop is used and the inlet and outlet node names are specified (i.e. the nodes are connected to a hot water loop), the generator outlet node temperature is calculated based on the generator input power, mass flow rate of water, and the specific heat of water entering the hot water generator. The calculations are based on the Chiller Flow Mode as follows.

Constant Flow Chillers:

$$\dot{m}_{generator} = \dot{m}_{generator,max}$$

Variable Flow Chillers:

$$\dot{m}_{generator} = \frac{\dot{Q}_{generator}}{C_{p,water} (\Delta T_{generator})}$$

$$T_{generator,out} = T_{generator,in} - \frac{\dot{Q}_{generator}}{\dot{m}_{generator} (C_{p,water})}$$

where

$\dot{m}_{generator}$ = generator hot water mass flow rate [kg/s]

$\dot{m}_{generator,max}$ = generator design hot water mass flow rate (kg/s)

$\Delta T_{generator}$ = generator design hot water temperature difference (°C)

Indirect Absorption Chiller

The Chiller:Absorption:Indirect object is an enhanced version of the absorption chiller model found in the Building Loads and System Thermodynamics (BLAST) program. This enhanced model is nearly identical to the existing absorption chiller model (Ref. Chiller:Absorption) with the exceptions that: 1) the enhanced indirect absorption chiller model provides more flexible performance curves and 2) chiller performance now includes the impact of varying evaporator, condenser, and generator temperatures. Since these absorption chiller models are nearly identical (i.e., the performance curves of the enhanced model can be manipulated to produce similar results to the previous model), it is quite probable that the Chiller:Absorption model will be deprecated in a future release of EnergyPlus.

The indirect absorption chiller's condenser and evaporator are similar to that of a standard chiller, which are both water-to-water heat exchangers. The assembly of a generator and absorber provides the compression operation. A schematic of a single-stage absorption chiller is shown in the figure below. Low-pressure vapor from the evaporator is absorbed by the liquid solution in the absorber. A pump receives low-pressure liquid from the absorber, elevates the pressure of the liquid, and delivers the liquid to the generator. In the generator, heat from a high temperature source (hot water or steam) drives off the vapor that has been absorbed by the solution. The liquid solution returns to the absorber through a throttling valve whose purpose is to provide a pressure drop to maintain the pressure difference between the generator and absorber. The heat supplied to the generator can be either hot water or steam, however, connection to an actual plant loop is not required. For more information on indirect absorption chillers, see the Input/Output Reference Document (Object: Chiller:Absorption:Indirect).

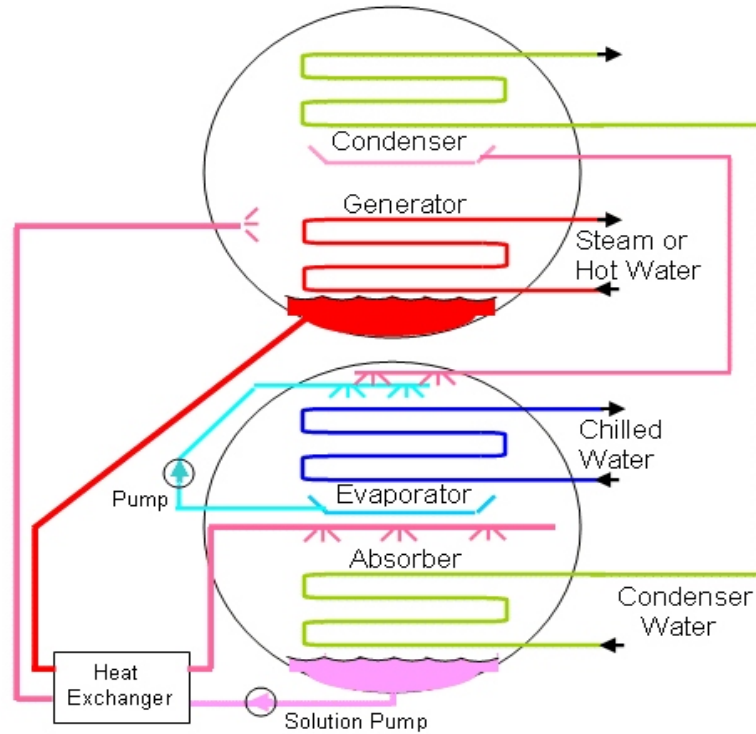


Figure 130. Schematic Diagram of a Single-Stage Absorption Chiller

The chiller cooling effect (capacity) will change with a change in condenser water temperature. Similarly, the chiller cooling effect will change as the temperature of the evaporator water changes. The chiller cooling effect will also change with a change in or generator inlet water temperature and only applies when Hot Water is used as the generator heat source. A quadratic or cubic equation is used to modify the rated chiller capacity as a function of both the condenser and generator inlet water temperatures and the evaporator outlet water temperature. If any or all of the capacity correction factor curves are not used, the correction factors are assumed to be 1.

$$CAPFT_{evaporator} = a + b(T_{evaporator}) + c(T_{evaporator})^2 + d(T_{evaporator})^3$$

$$CAPFT_{condenser} = e + f(T_{condenser}) + g(T_{condenser})^2 + h(T_{condenser})^3$$

$$CAPFT_{generator} = i + j(T_{generator}) + k(T_{generator})^2 + l(T_{generator})^3 \text{ (Hot Water only)}$$

$$\dot{Q}_{evap,max} = \dot{Q}_{evap,rated} (CAPFT_{evaporator})(CAPFT_{condenser})(CAPFT_{generator})$$

where

$CAPFT_{evaporator}$ = Capacity correction (function of evaporator temperature) factor

$CAPFT_{condenser}$ = Capacity correction (function of condenser temperature) factor

$CAPFT_{generator}$ = Capacity correction (function of generator temperature) factor

$T_{evaporator}$ = evaporator outlet water temperature [C]

$T_{condenser}$ = condenser inlet water temperature [C]

$T_{generator}$ = generator inlet water temperature [C]

$\dot{Q}_{evap, max}$ = maximum chiller capacity [W]

$\dot{Q}_{evap, rated}$ = rated chiller capacity [W]

The part-load ratio of the indirect absorption chiller's evaporator is simply the actual cooling effect required (load) divided by the maximum cooling effect available.

$$PLR = \frac{\dot{Q}_{evap}}{\dot{Q}_{evap, max}}$$

where

PLR = part-load ratio of chiller evaporator

\dot{Q}_{evap} = chiller evaporator operating capacity [W]

The generator's heat input is also a function of several parameters. The primary input for determining the heat input requirements is the Generator Heat Input function of Part-Load Ratio Curve. The curve is a quadratic or cubic equation that determines the ratio of the generator heat input to the chiller's maximum capacity ($\dot{Q}_{evap, max}$) and is solely a function of part-load ratio. Typical generator heat input ratios at full load (i.e., $PLR=1$) are between 1 and 2. Two additional curves are available to modify the heat input requirement based on the condenser inlet water temperature and the evaporator outlet water temperature.

$$GeneratorHIR = a + b(PLR) + c(PLR)^2 + d(PLR)^3$$

$$GenfCondT = e + f(T_{generator}) + g(T_{generator})^2 + h(T_{generator})^3$$

$$GenfEvapT = i + j(T_{evaporator}) + k(T_{evaporator})^2 + l(T_{evaporator})^3$$

where

$GeneratorHIR$ = ratio of generator heat input to chiller operating capacity

$GenfCondT$ = heat input modifier based on generator temperature

$GenfEvapT$ = heat input modifier based on evaporator temperature

The Pump Electric Use function of Part-Load Ratio Curve is a quadratic or cubic equation that determines the ratio of the actual absorber pumping power to the nominal pumping power.

$$ElectricInputRatio = a + b(PLR) + c(PLR)^2 + d(PLR)^3$$

If the chiller operating part-load ratio is greater than the minimum part-load ratio, the chiller will run the entire time step and cycling will not occur (i.e. $CyclingFrac = 1$). If the operating

part-load ratio is less than the minimum part-load ratio, the chiller will be on for a fraction of the time step equal to *CyclingFrac*. Generator heat input and pump electrical energy use are also calculated using the chiller part-load cycling fraction.

$$CyclingFrac = MIN\left(1, \frac{PLR}{PLR_{min}}\right)$$

$$\dot{Q}_{generator} = GeneratorHIR(\dot{Q}_{evap,max})(GenfCondT)(GenfEvapT)(CyclingFrac)$$

$$\dot{Q}_{generator} = ElectricInputRatio(P_{pump})(CyclingFrac)$$

where

CyclingFrac = chiller part-load cycling fraction

PLR_{min} = chiller minimum part-load ratio

$\dot{Q}_{generator}$ = generator heat input [W]

\dot{Q}_{pump} = chiller pumping power [W]

The evaporator water mass flow rate is calculated based on the Chiller Flow Mode as follows.

Constant Flow Chillers:

$$\dot{m}_{evap} = \dot{m}_{evap,max}$$

Variable Flow Chillers:

$$\Delta T_{evap} = T_{evap,in} - T_{evap,SP}$$

$$\dot{m}_{evap} = \frac{\dot{Q}_{evap}}{C_{p,evap}(\Delta T_{evap})}$$

where

\dot{m}_{evap} = chiller evaporator water mass flow rate (kg/s)

$\dot{m}_{evap,max}$ = chiller design evaporator water mass flow rate (kg/s)

ΔT_{evap} = chiller evaporator water temperature difference (°C)

$T_{evap,in}$ = chiller evaporator inlet water temperature (°C)

$T_{evap,SP}$ = chiller evaporator outlet water setpoint temperature (°C)

$C_{p,evap}$ = specific heat of water entering evaporator (J/kg °C)

The evaporator outlet water temperature is then calculated based on the cooling effect produced and the evaporator entering water temperature.

$$T_{evap,out} = T_{evap,in} + \frac{\dot{Q}_{evap}}{C_{p,evap} (\dot{m}_{evap})}$$

where

$T_{evap,out}$ = chiller evaporator outlet water temperature [°C]

$T_{evap,in}$ = chiller evaporator inlet water temperature [°C]

$C_{p,evap}$ = specific heat of chiller evaporator inlet water [J/kg/°C]

\dot{m}_{evap} = chiller evaporator water mass flow rate [kg/s]

The condenser heat transfer and condenser leaving water temperature are also calculated.

$$\dot{Q}_{cond} = \dot{Q}_{evap} + \dot{Q}_{generator} + \dot{Q}_{pump}$$

$$T_{cond,out} = T_{cond,in} + \frac{\dot{Q}_{cond}}{C_{p,cond} (\dot{m}_{cond})}$$

where

\dot{Q}_{cond} = chiller condenser heat transfer rate [W]

$T_{cond,out}$ = chiller condenser outlet water temperature [°C]

$T_{cond,in}$ = chiller condenser inlet water temperature [°C]

$C_{p,cond}$ = specific heat of chiller condenser inlet water [J/kg/°C]

\dot{m}_{cond} = chiller condenser water mass flow rate [kg/s]

The absorption chiller can model the impact of steam or hot water entering the generator, although the connection of the steam (hot water) nodes to a plant is not actually required. The calculations specific to the generator depend on the type of fluid used and are described here in further detail.

Steam Loop Calculations

When a steam loop is used and the inlet and outlet node names are specified (i.e. the nodes are connected to a steam loop), the generator outlet node steam mass flow rate and temperature are calculated based on the generator heat input, latent heat of steam, the specific heat of water, and the amount of subcooling in the steam generator. The model assumes dry saturated steam enters the generator and exits the generator as a subcooled liquid. The temperature leaving the generator is calculated based on the user entered amount of liquid subcooling in the generator. The effect of subcooling of the liquid (condensate) in the pipe returning to the boiler is also modeled using the user entered amount of steam condensate loop subcooling.

$$\dot{m}_{steam} = \frac{\dot{Q}_{generator}}{h_{fg} + c_{p,water} \times \Delta T_{sc}}$$

$$T_{generator,out} = T_{generator,in} - \Delta T_{sc}$$

$$T_{loop,out} = T_{generator,out} - \Delta T_{sc,loop}$$

where

\dot{m}_{steam} = chiller steam mass flow rate [kg/s]

h_{fg} = latent heat of steam [J/kg]

$c_{p,water}$ = specific heat of water [J/Kg °C]

ΔT_{sc} = amount of subcooling in steam generator [°C]

$\Delta T_{sc,loop}$ = amount of condensate subcooling in steam loop [°C]

$T_{generator,out}$ = generator steam outlet node temperature [°C]

$T_{generator,in}$ = generator steam inlet node temperature [°C]

Hot Water Loop Calculations

When a hot water loop is used and the inlet and outlet node names are specified (i.e. the nodes are connected to a hot water loop), the generator outlet node temperature is calculated based on the generator heat input, mass flow rate of water, and the specific heat of water entering the hot water generator. The calculations are based on the Chiller Flow Mode as follows.

Constant Flow Chillers:

$$\dot{m}_{generator} = \dot{m}_{generator,max}$$

Variable Flow Chillers:

$$\dot{m}_{generator} = \frac{\dot{Q}_{generator}}{C_{p,water} (\Delta T_{generator})}$$

$$T_{generator,out} = T_{generator,in} - \frac{\dot{Q}_{generator}}{\dot{m}_{generator} (C_{p,water})}$$

where

$\dot{m}_{generator}$ = generator hot water mass flow rate [kg/s]

$\dot{m}_{generator,max}$ = generator design hot water mass flow rate (kg/s)

$\Delta T_{generator}$ = generator design hot water temperature difference (°C)

Combustion Turbine Chiller

The input object Chiller:CombustionTurbine provides a chiller model that is the empirical model from the Building Loads and System Thermodynamics (BLAST) program. The model fits catalog data to third order polynomial equations used for chiller performance curves. Three sets of coefficients are required.

The gas turbine-driven chiller is an open centrifugal chiller driven directly by a gas turbine. The BLAST model of an open centrifugal chiller is modeled as standard vapor compression refrigeration cycle with a centrifugal compressor driven by a shaft power from an engine. The centrifugal compressor has the incoming fluid entering at the eye of a spinning impeller that throws the fluid by centrifugal force to the periphery of the impeller. After leaving the compressor, the refrigerant is condensed to liquid in a refrigerant to water condenser. The heat from the condenser is rejected to a cooling tower, evaporative condenser, or well water condenser depending on which one is selected by the user based on the physical parameters of the plant. The refrigerant pressure is then dropped through a throttling valve so that fluid can evaporate at a low pressure that provides cooling to the evaporator. The evaporator can chill water that is pumped to chilled water coils in the building. For more information, see the Input/Output Reference Document.

This chiller is modeled like the electric chiller with the same numerical curve fits and then some additional curve fits to model the turbine drive. Shown below are the definitions of the curves that describe this model.

The chiller's temperature rise coefficient which is defined as the ratio of the required change in condenser water temperature to a given change in chilled water temperature, which maintains the capacity at the nominal value. This is calculated as the following ratio:

$$\frac{TCEnt_{required} - TCEnt_{rated}}{TELV_{required} - TELV_{rated}}$$

Where:

$TCEnt_{required}$ = Required entering condenser air or water temperature to maintain rated capacity.

$TCEnt_{rated}$ = Rated entering condenser air or water temperature at rated capacity.

$TELV_{required}$ = Required leaving evaporator water outlet temperature to maintain rated capacity.

$TELV_{rated}$ = Rated leaving evaporator water outlet temperature at rated capacity.

The Capacity Ratio Curve is a quadratic equation that determines the Ratio of Available Capacity to Nominal Capacity. The defining equation is:

$$AvailToNominalCapacityRatio = C_1 + C_2\Delta_{temp} + C_3\Delta_{temp}^2$$

Where the Delta Temperature is defined as:

$$\Delta_{Temp} = \frac{TempCondIn - TempCondInDesign}{TempRiseCoefficient} - (TempEvapOut - TempEvapOutDesign)$$

TempCondIn = Temperature entering the condenser (water or air temperature depending on condenser type).

TempCondInDesign = Temp Design Condenser Inlet from User input above.

TempEvapOut = Temperature leaving the evaporator.

TempEvapOutDesign = Temp Design Evaporator Outlet from User input above.

TempRiseCoefficient = User Input from above.

The following three fields contain the coefficients for the quadratic equation.

The Power Ratio Curve is a quadratic equation that determines the Ratio of Full Load to Power. The defining equation is:

$$\text{FullLoadtoPowerRatio} = C_1 + C_2 \text{AvailToNominalCapRatio} + C_3 \text{AvailToNominalCapRatio}^2$$

The Full Load Ratio Curve is a quadratic equation that determines the fraction of full load power. The defining equation is:

$$\text{FracFullLoadPower} = C_1 + C_2 \text{PartLoadRatio} + C_3 \text{PartLoadRatio}^2$$

The Fuel Input Curve is a polynomial equation that determines the Ratio of Fuel Input to Energy Output. The equation combines both the Fuel Input Curve Coefficients and the Temperature Based Fuel Input Curve Coefficients. The defining equation is:

$$\text{FuelEnergyInput} = \text{PLoad} * (\text{FIC}_1 + \text{FIC}_2 \text{Rload} + \text{FIC}_3 \text{Rload}^2) * (\text{TBFIC}_1 + \text{TBFIC}_2 \text{AT}_{\text{air}} + \text{TBFIC}_3 \text{AT}_{\text{air}}^2)$$

Where FIC represents the Fuel Input Curve Coefficients, TBFIC represents the Temperature Based Fuel Input Curve Coefficients, Rload is the Ratio of Load to Combustion Turbine Engine Capacity, and AT_{air} is the difference between the current ambient and design ambient temperatures.

The Exhaust Flow Curve is a quadratic equation that determines the Ratio of Exhaust Gas Flow Rate to Engine Capacity. The defining equation is:

$$\text{ExhaustFlowRate} = \text{GTCapacity} * (C_1 + C_2 \text{AT}_{\text{air}} + C_3 \text{AT}_{\text{air}}^2)$$

Where GTCapacity is the Combustion Turbine Engine Capacity, and AT_{air} is the difference between the current ambient and design ambient temperatures.

The Exhaust Gas Temperature Curve is a polynomial equation that determines the Exhaust Gas Temperature. The equation combines both the Exhaust Gas Temperature Curve Coefficients (Based on the Part Load Ratio) and the (Ambient) Temperature Based Exhaust Gas Temperature Curve Coefficients. The defining equation is:

$$\text{ExhaustTemperature} = (C_1 + C_2 \text{Rload} + C_3 \text{Rload}^2) * (\text{TBC}_1 + \text{TBC}_2 \text{AT}_{\text{air}} + \text{TBC}_3 \text{AT}_{\text{air}}^2) - 273.15$$

Where C represents the Exhaust Gas Temperature Curve Coefficients, TBC are the Temperature Based Exhaust Gas Temperature Curve Coefficients, Rload is the Ratio of Load to Combustion Turbine Engine Capacity, and AT_{air} is the difference between the actual ambient and design ambient temperatures.

The Recovery Lubricant Heat Curve is a quadratic equation that determines the recovery lube energy. The defining equation is:

$$\text{RecoveryLubeEnergy} = \text{Pload} * (C_1 + C_2 \text{RL} + C_3 \text{RL}^2)$$

Where Pload is the engine load and RL is the Ratio of Load to Combustion Turbine Engine Capacity

The UA is an equation that determines the overall heat transfer coefficient for the exhaust gasses with the stack. The heat transfer coefficient ultimately helps determine the exhaust stack temperature. The defining equation is:

$$\text{UAtoCapacityRatio} = C_1 \text{GasTurbineEngineCapacity}^{C_2}$$

Constant COP Chiller

The input object Chiller:ConstantCOP provides a chiller model that is based on a simple, constant COP simulation of the chiller. In this case, performance does not vary with chilled water temperature or condenser conditions. The nominal capacity of the chiller and the COP are user specified along with the connections to the plant and condenser loop and mass flow rates. *Such a model is useful when the user does not have access to detailed performance data.*

The chiller power is calculated from the load divided by the COP. This chiller will meet the load as long as it does not exceed the nominal capacity specified by the user.

```
QEvaporator = Load
Power = Load / ConstCOPChiller(ChillNum)%COP
```

Then the evaporator temperatures are calculated from the load

```
EvapDeltaTemp = QEvaporator/EvapMassFlowRate/CPwater
EvapOutletTemp = Node(EvapInletNode)%Temp - EvapDeltaTemp
```

The condenser load and temperatures are calculated from the evaporator load and the power to the chiller.

```
QCondenser = Power + QEvaporator

IF (ConstCOPChiller(ChillNum)%CondenserType == WaterCooled) THEN
  IF (CondMassFlowRate > WaterMassFlowTol) THEN
    CondOutletTemp = QCondenser/CondMassFlowRate/CPCW(CondInletTemp) + CondInletTemp
  ELSE
    CALL ShowSevereError('CalcConstCOPChillerModel: Condenser flow = 0, for CONST COP Chiller='// &
      TRIM(ConstCOPChiller(ChillNum)%Name))
    CALL ShowContinueErrorTimeStamp(' ')
    CALL ShowFatalError('Program Terminates due to previous error condition.')
  END IF
ELSE ! Air Cooled or Evap Cooled
  ! Set condenser outlet temp to condenser inlet temp for Air Cooled or Evap Cooled
  ! since there is no CondMassFlowRate and would divide by zero
  CondOutletTemp = CondInletTemp
END IF
```

See the InputOutput Reference for additional information.

Hot Water Heat Recovery from Chillers

The electric chillers (e.g., Chiller:Electric, Chiller:Electric:EIR, and Chiller:Electric:ReformulatedEIR) can have the option of having its condenser hooked up to a heat recovery loop or what is commonly known as a double bundled chiller. The Heat Recovery chiller is simulated as a standard vapor compression refrigeration cycle with a double bundled condenser. A double bundle condenser involves two separate flow paths through the condenser. One of these paths is to a standard cooling tower, the other path is to the heat recovery. After leaving the compressor, the refrigerant is condensed to liquid in a refrigerant to water condenser. The refrigerant pressure is then dropped through a throttling valve so that fluid can evaporate at a low pressure that provides cooling to the evaporator.

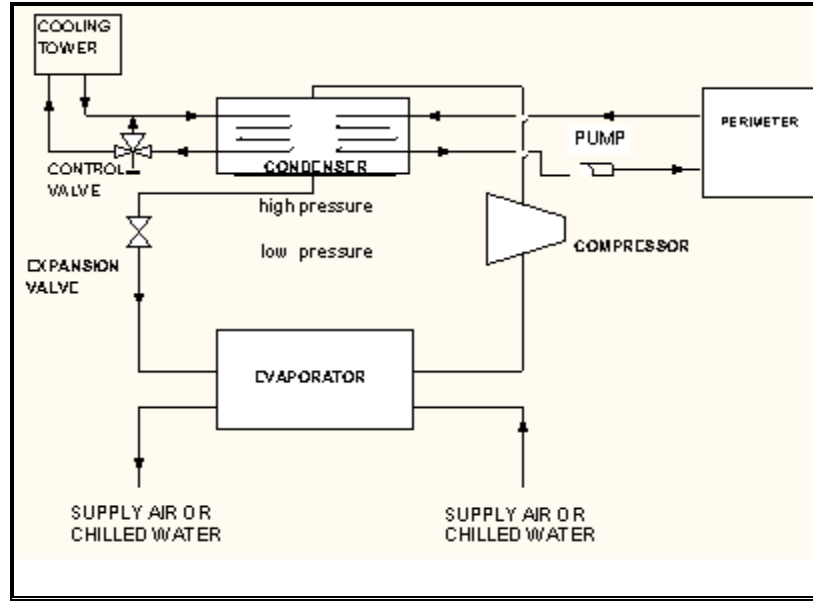


Figure 131. Diagram of Chiller:Electric with Heat Recovery

Algorithm for the Heat Recovery portion of the chiller needs to be determined from relatively simple inputs to estimate the amount of the heat that is recovered and then send the rest of the heat to the cooling tower.

$$Q_{Tot} = Q_{Evap} + Q_{Elec} = Q_{Cond} + Q_{Heat Rec}$$

$$Q_{Tot} = \dot{m}_{Heat Rec} * C_{p_{Heat Rec}} * (T_{Heat Rec Out} - T_{Heat Rec In}) + \dot{m}_{Cond} * C_{p_{Cond}} * (T_{Cond Out} - T_{Cond In})$$

Approximate the total heat transfer with average temperatures in and out.

$$Q_{Tot} = (\dot{m}_{Heat Rec} * C_{p_{Heat Rec}} + \dot{m}_{Cond} * C_{p_{Cond}}) * (T_{Avg Out} - T_{Avg In})$$

Then the inlet temperature can be flow ratio averaged to determine its conditions.

$$T_{Avg In} = \frac{(\dot{m}_{Heat Rec} * C_{p_{Heat Rec}} * T_{Heat Rec In} + \dot{m}_{Cond} * C_{p_{Cond}} * T_{Cond In})}{(\dot{m}_{Heat Rec} * C_{p_{Heat Rec}} + \dot{m}_{Cond} * C_{p_{Cond}})}$$

$$T_{Avg Out} = \frac{Q_{Tot}}{(\dot{m}_{Heat Rec} * C_{p_{Heat Rec}} + \dot{m}_{Cond} * C_{p_{Cond}})} + T_{Avg In}$$

Then the total heat can have an approximate fractional split between heat recovered and rejection through the tower by the following.

$$\frac{Q_{Heat Rec}}{Q_{Cond}} \cong Frac_{Heat Rec} = \frac{\dot{m}_{Heat Rec} * C_{p_{Heat Rec}} * (T_{Avg Out} - T_{Heat Rec In})}{\dot{m}_{Cond} * C_{p_{Cond}} * (T_{Avg Out} - T_{Cond In})}$$

Therefore the heat rejected to the water-cooled condenser and heat recovery is:

$$Q_{Cond} = (1 - Frac_{Heat Rec}) * Q_{Tot}$$

$$Q_{Heat Rec} = Frac_{Heat Rec} * Q_{Tot}$$

This allows for an approximate split of the heat recovery with some relationship to the mass flow rates and the inlet temperatures of each. Otherwise, the user has to pre-determine the heat recovery split with no relationship to actual simulation conditions.

Electric Chiller Model Based on Condenser Entering Temperature

Overview

This model (object name Chiller:Electric:EIR) simulates the performance of an electric liquid chiller. The model is based on the compression chiller model (COMREF) in the DOE-2.1 building energy simulation program. The EnergyPlus model contains all of the features of the DOE-2.1 chiller model, plus additional abilities for modeling evaporatively-cooled condensers and heat recovery for water heating.

This model simulates the thermal performance of the chiller and the power consumption of the compressor(s). It also models the power consumption of condenser fans if modeling an air-cooled or evaporatively-cooled condenser. This model does not simulate the thermal performance or the power consumption of associated pumps or cooling towers. This auxiliary equipment must be modeled using other EnergyPlus models (e.g. CoolingTower:SingleSpeed).

Model Description

The chiller model uses user-supplied performance information at reference conditions along with three performance curves (curve objects) for cooling capacity and efficiency to determine chiller operation at off-reference conditions. The three performance curves are:

8) Cooling Capacity Function of Temperature Curve

9) Energy Input to Cooling Output Ratio Function of Temperature Curve

10) Energy Input to Cooling Output Ratio Function of Part Load Ratio Curve

- The cooling capacity function of temperature curve is a biquadratic performance curve with two independent variables: the leaving chilled water temperature and the entering condenser fluid temperature. The output of this curve is multiplied by the reference capacity to give the full-load cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The biquadratic curve should be valid for the range of water temperatures anticipated for the simulation.

$$ChillerCapFTemp = a + b(T_{cw,l}) + c(T_{cw,l})^2 + d(T_{cond,e}) + e(T_{cond,e})^2 + f(T_{cw,l})(T_{cond,e})$$

where

ChillerCapFTemp = cooling capacity factor, equal to 1 at reference conditions

T_{cw,l} = leaving chilled water temperature, °C

T_{cond,e} = entering condenser fluid temperature, °C. For a water-cooled condenser this will be the water temperature returning from the condenser loop (e.g., leaving the cooling tower). For air- or evap-cooled condensers this will be the entering outdoor air dry-bulb or wet-bulb temperature, respectively.

- The energy input to cooling output ratio function of temperature curve is a biquadratic performance curve that parameterizes the variation of the energy input to cooling output ratio (EIR) as a function of the leaving chilled water temperature and the

entering condenser fluid temperature. The EIR is the inverse of the COP. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) to give the full-load EIR at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The biquadratic curve should be valid for the range of water temperatures anticipated for the simulation.

$$\text{ChillerEIRFTemp} = a + b(T_{cw,l}) + c(T_{cw,l})^2 + d(T_{cond,e}) + e(T_{cond,e})^2 + f(T_{cw,l})(T_{cond,e})$$

where

ChillerEIRFTemp = energy input to cooling output factor, equal to 1 at reference conditions

$T_{cw,l}$ = leaving chilled water temperature, °C

$T_{cond,e}$ = entering condenser fluid temperature, °C. For a water-cooled condenser this will be the water temperature returning from the condenser loop (e.g., leaving the cooling tower). For air- or evap-cooled condensers this will be the entering outdoor air dry-bulb or wet-bulb temperature, respectively.

- The energy input to cooling output ratio function of part-load ratio curve is a quadratic performance curve that parameterizes the variation of the energy input ratio (EIR) as a function of the part-load ratio. The EIR is the inverse of the COP, and the part-load ratio is the actual cooling load divided by the chiller's available cooling capacity. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) and the Energy Input to Cooling Output Ratio Function of Temperature Curve to give the EIR at the specific temperatures and part-load ratio at which the chiller is operating. This curve should have a value of 1.0 when the part-load ratio equals 1.0. The quadratic curve should be valid for the range of part-load ratios anticipated for the simulation.

$$\text{ChillerEIRFPLR} = a + b(PLR) + c(PLR)^2$$

where

ChillerEIRFPLR = energy input to cooling output factor, equal to 1 at reference conditions

PLR = part-load ratio = (cooling load) / (chiller's available cooling capacity)

All three of the performance curves are accessed through EnergyPlus' built-in performance curve equation manager (curve:quadratic and curve:biquadratic). It is not imperative that the user utilize all coefficients in the performance curve equations if their performance equation has fewer terms (e.g., if the user's ChillerEIRFPLR performance curve is linear instead of quadratic, simply enter the values for a and b, and set coefficient c equal to zero). Performance curves for more than 160 chillers, including the default DOE-2.1E reciprocating and centrifugal chillers, are provided in the EnergyPlus Reference DataSets (Chillers.idf and AllDataSets.idf).

Note: Chiller:Electric:EIR objects and their associated performance curve objects are developed using performance information for a specific chiller and should normally be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with caution.

For any simulation time step, the chiller's available cooling capacity is calculated as follows:

$$\dot{Q}_{avail} = \dot{Q}_{ref} (ChillerCapFTemp)$$

where

\dot{Q}_{ref} = chiller capacity at reference conditions (reference temperatures and flow rates defined by the user), W

\dot{Q}_{avail} = available chiller capacity adjusted for current fluid temperatures, W

The model then calculates the evaporator heat transfer rate required to bring the entering chilled water temperature down to the leaving chilled water setpoint temperature (established using a SetpointManager object and referenced in the PlantLoop object). If this calculated heat transfer rate is greater than the heat transfer rate being requested by the plant equipment operation scheme, then the evaporator heat transfer rate is reset to the requested cooling rate.

The evaporator heat transfer rate is then compared to the available capacity. If the available chiller capacity is sufficient to meet the evaporator heat transfer rate, the leaving chilled water temperature is set equal to the chilled water setpoint temperature. If the requested evaporator heat transfer rate is larger than the available capacity the chilled water leaving the evaporator is allowed to float upward. For this case, the exiting chilled water temperature is calculated based on the water temperature entering the evaporator, the available cooling capacity, and the evaporator mass flow rate as follows:

$$T_{cw,l} = T_{cw,e} - \left(\frac{\dot{Q}_{avail}}{\dot{m}_{evap} C_{p,evap}} \right)$$

where

$T_{cw,l}$ = water temperature leaving the evaporator, °C

$T_{cw,e}$ = water temperature entering the evaporator, °C

\dot{m}_{evap} = evaporator mass flow rate, kg/s

$C_{p,evap}$ = specific heat of water entering evaporator at $T_{cw,e}$, J/kg-°C

The part-load ratio is then calculated as the ratio of the evaporator heat transfer rate to the available chiller capacity. The part-load ratio is not allowed to be greater than the maximum part-load ratio specified by the user or less than zero as follows:

$$PLR = MAX \left(0.0, MIN \left(\frac{\dot{Q}_{evap}}{\dot{Q}_{avail}}, PLR_{max} \right) \right)$$

where

PLR = part-load ratio

\dot{Q}_{evap} = load to be met by the chiller, W

PLR_{max} = maximum part-load ratio (specified by the user in the input data file)

The model assumes that the cooling load is met through chiller unloading down to the minimum unloading ratio. False loading (e.g. hot-gas bypass) is assumed to occur between the minimum unloading ratio and the minimum part load ratio yielding constant electrical power consumption under these conditions. Below the minimum part load ratio, the chiller cycles on and off to meet very small loads and the power consumption during the on cycle is the same as when the chiller is operating at the minimum part load ratio. When the chiller part load ratio is less than the minimum part load ratio, the on-off cycling ratio of the chiller is calculated as follows and is available as an output variable.

$$ChillerCyclingRatio = MIN\left(\frac{PLR}{PLR_{min}}, 1.0\right)$$

To properly account for chiller electric power consumption when PLR is less than the minimum unloading ratio, the PLR is reset to the greater of the PLR calculated above and the PLR at the minimum unloading ratio. The result is available as the output variable Chiller Part Load Ratio.

$$PLR = MAX(PLR, PLR_{MinUnloadRatio})$$

This revised PLR accounts for the “false loading” (e.g., hot gas bypass) that is assumed to occur whenever the PLR (based on cooling load divided by available capacity) is less than the minimum unloading ratio specified. The amount of false loading on the chiller is calculated using this revised PLR and is reported as an output variable as follows:

$$\dot{Q}_{falseloading} = \left(\dot{Q}_{avail}\right)(PLR)(ChillerCyclingRatio) - \dot{Q}_{evap}$$

The electrical power consumption for the chiller compressor(s) for any simulation time step is then calculated using the following equation:

$$P_{chiller} = \left(\dot{Q}_{avail}\right)\left(\frac{1}{COP_{ref}}\right)(ChillerEIRFTemp)(ChillerEIRFPLR)(ChillerCyclingRatio)$$

where

$P_{chiller}$ = chiller compressor power, W

COP_{ref} = reference coefficient of performance, W/W

Heat rejected by the chiller condenser includes the heat transferred in the evaporator plus a portion or all of the compressor electrical energy consumption. For electric chillers with hermetic compressors, all compressor energy consumption is rejected by the condenser (compressor motor efficiency = $eff_{motor} = 1.0$). For chillers with semi-hermetic or open compressors, only a portion of the compressor energy use is rejected by the condenser. The heat transfer rate for the chiller condenser is calculated as follows:

$$\dot{Q}_{cond} = (P_{chiller} * eff_{motor}) + \dot{Q}_{evap} + \dot{Q}_{falseloading}$$

where

- \dot{Q}_{cond} = condenser heat transfer rate, W
 eff_{motor} = compressor motor efficiency = fraction of compressor electrical energy consumption rejected as condenser heat

For water-cooled chillers, the water temperature leaving the condenser is then calculated as shown below.

$$T_{cond,l} = T_{cond,e} + \frac{\dot{Q}_{cond}}{\dot{m}_{cond} * C_{p,cond}}$$

where:

- $T_{cond,l}$ = water temperature leaving the condenser, °C
 $T_{cond,e}$ = water temperature entering the condenser, °C
 \dot{m}_{cond} = mass flow rate through the condenser, kg/s
 $C_{p,cond}$ = specific heat of water entering the condenser at $T_{cond,e}$, J/kg-°C

For air- and evaporatively-cooled condensers, the exiting air temperature is not calculated and is set equal to the entering air or wet-bulb temperature, respectively.

The model then calculates the condenser fan energy for air- and evaporatively-cooled condensers. The amount of condenser fan energy is assumed to be proportional to the chiller cycling ratio and is calculated as follows:

$$P_{cond} = \dot{Q}_{ref} (P_{condfanratio}) (ChillerCyclingRatio)$$

where

- P_{cond} = chiller condenser fan electric power, W
 $P_{condfanratio}$ = condenser fan power ratio, W/W

The final calculations determine the total heat transfer energy for the condenser and evaporator, as well as the total electric energy consumed by the chiller compressor motor(s) and condenser fan(s). The results are available as output variables.

$$Q_{cond} = \dot{Q}_{cond} * TimeStepSys * 3600$$

$$Q_{evap} = \dot{Q}_{evap} * TimeStepSys * 3600$$

$$E_{chiller} = P_{chiller} * TimeStepSys * 3600$$

$$E_{cond} = P_{cond} * TimeStepSys * 3600$$

where

Q_{cond}	= chiller condenser heat transfer, J
Q_{evap}	= chiller evaporator heat transfer, J
$E_{chiller}$	= chiller (compressor) electric consumption, J
E_{cond}	= chiller condenser fan electric consumption, J
$TimeStepSys$	= HVAC system simulation time step, hr
3600	= conversion factor, sec/hr

Electric EIR Chiller with Heat Recovery Option

Heat from the electric EIR chiller condenser may be recovered when a water-cooled condenser is selected for simulation. The heat recovery water flow rate is specified by the user along with the input and output nodes connected to the heat recovery loop. The algorithms are identical to those used for Chiller:Electric. Refer to the section entitled Chillers with Plant Heat Recovery for details.

Electric Chiller Model Based on Condenser Leaving Temperature

Overview

This model (object name Chiller:Electric:ReformulatedEIR) simulates the thermal performance of an electric liquid chiller and the power consumption of its compressor(s). The model, developed by Hydeman et al. (2002) as part of the CoolTools™ project sponsored by Pacific Gas and Electric Company (PG&E), is an empirical model similar to EnergyPlus' Chiller:Electric:EIR model. The model uses performance information at reference conditions along with three curve fits for cooling capacity and efficiency to determine chiller operation at off-reference conditions. The model has the same capabilities as the Chiller:Electric:EIR model, but can potentially provide significant accuracy improvement over the Chiller:Electric:EIR model for chillers with variable-speed compressor motor drives and/or variable condenser water flow applications.

Chiller performance curves can be generated by fitting manufacturer's catalog data or measured data. Performance curves developed primarily from manufacturer's performance data are provided in the EnergyPlus Reference DataSets (Chillers.idf and AllDataSets.idf). This chiller model can be used to predict the performance of various chiller types (e.g., reciprocating, screw, scroll, and centrifugal) with water-cooled condensers. The model does not simulate the thermal performance or the power consumption of associated pumps or cooling towers. This auxiliary equipment must be modeled using other EnergyPlus models (e.g. CoolingTower:SingleSpeed).

The main difference between this model and the Chiller:Electric:EIR model is the condenser fluid temperature used in the associated performance curves: the Chiller:Electric:ReformulatedEIR model uses the LEAVING condenser water temperature while the Chiller:Electric:EIR model uses the ENTERING condenser water temperature. In addition, the Energy Input to Cooling Output Function of Part Load Ratio curve for this reformulated EIR chiller model includes the condenser leaving water temperature as an independent variable in addition to part-load ratio. Since the leaving condenser water temperature is a function of load, chiller performance, and condenser entering water temperature, EnergyPlus must iterate to converge on a solution for each simulation time step.

Model Description

The chiller model uses user-supplied performance information at reference conditions along with three performance curves (curve objects) for cooling capacity and efficiency to determine chiller operation at off-reference conditions. The three performance curves are:

- 1) Cooling Capacity Function of Temperature Curve
- 2) Energy Input to Cooling Output Ratio Function of Temperature Curve
- 3) Energy Input to Cooling Output Ratio Function of Part Load Ratio Curve

- The cooling capacity function of temperature curve is a biquadratic performance curve with two independent variables: the leaving chilled water temperature and the leaving condenser water temperature. The output of this curve is multiplied by the reference capacity to give the full-load cooling capacity at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The biquadratic curve should be valid for the range of water temperatures anticipated for the simulation (otherwise the program issues warning messages).

$$\text{ChillerCapFTemp} = a + b(T_{cw,l}) + c(T_{cw,l})^2 + d(T_{cond,l}) + e(T_{cond,l})^2 + f(T_{cw,l})(T_{cond,l})$$

where

ChillerCapFTemp = Cooling capacity factor, equal to 1 at reference conditions

$T_{cw,l}$ = leaving chilled water temperature, °C

$T_{cond,l}$ = leaving condenser water temperature, °C. This will be the water temperature entering the condenser loop (e.g., entering the cooling tower).

- The energy input to cooling output ratio function of temperature curve is a biquadratic performance curve that parameterizes the variation of the energy input to cooling output ratio (EIR) as a function of the leaving chilled water temperature and the leaving condenser water temperature. The EIR is the inverse of the COP. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) to give the full-load EIR at specific temperature operating conditions (i.e., at temperatures different from the reference temperatures). The curve should have a value of 1.0 at the reference temperatures and flow rates specified in the input data file by the user. The biquadratic curve should be valid for the range of water temperatures anticipated for the simulation (otherwise the program issues warning messages).

$$\text{ChillerEIRFTemp} = a + b(T_{cw,l}) + c(T_{cw,l})^2 + d(T_{cond,l}) + e(T_{cond,l})^2 + f(T_{cw,l})(T_{cond,l})$$

here

ChillerEIRFTemp = Energy input to cooling output factor, equal to 1 at reference conditions

$T_{cw,l}$ = leaving chilled water temperature, °C

$T_{cond,l}$ = leaving condenser water temperature, °C. This will be the water temperature entering the condenser loop (e.g., entering the cooling tower).

- The energy input to cooling output ratio function of part-load ratio curve is a bicubic performance curve that parameterizes the variation of the energy input ratio (EIR) as a function of the leaving condenser water temperature and the part-load ratio. The EIR is the inverse of the COP, and the part-load ratio is the actual cooling load divided by the chiller's available cooling capacity. The output of this curve is multiplied by the reference EIR (inverse of the reference COP) and the Energy Input to Cooling Output Ratio Function of Temperature Curve to give the EIR at the specific temperatures and part-load ratio at which the chiller is operating. This curve should have a value of 1.0 at the reference leaving condenser water temperature with part-load ratio equal to 1.0. It is recommended that this performance curve be developed using both full- and part-load performance data. The bicubic curve should be valid for the range of condenser water temperatures and part-load ratios anticipated for the simulation (otherwise the program issues warning messages).

$$\text{ChillerEIRFPLR} = a + b(T_{\text{cond},l}) + c(T_{\text{cond},l})^2 + d(\text{PLR}) + e(\text{PLR})^2 + f(T_{\text{cond},l})(\text{PLR}) + g(\text{PLR})^3$$

where

ChillerEIRFPLR = Energy input to cooling output factor, equal to 1 at the reference leaving condenser water temperature and PLR = 1.0

$T_{\text{cond},l}$ = leaving condenser water temperature, °C. This will be the water temperature entering the condenser loop (e.g., entering the cooling tower).

PLR = Part load ratio = (cooling load) / (chiller's available cooling capacity)

Note: Although a bicubic curve requires 10 coefficients (ref. Curve:Bicubic), coefficients 7, 9 and 10 are typically not used in the performance curve described here and should be entered as 0 unless sufficient performance data and regression accuracy exist to justify the use of these terms of the bicubic curve.

All three of the performance curves are accessed through EnergyPlus' built-in performance curve equation manager (curve:biquadratic and curve:bicubic). Note that the above three performance curves use the leaving condenser water temperature as an independent variable, instead of the entering condenser water temperature used in the performance curves for the Chiller:Electric:EIR model. Since the leaving condenser water temperature is calculated based on the condenser heat transfer rate, which is a function of the load to be met by the chiller, chiller compressor power, and the false loading (detailed calculations are given below), iterative calculations are required to determine the actual (converged) leaving condenser water temperature. The program uses the leaving condenser water temperature from the previous iteration to calculate values for each of the three performance curves described above. After obtaining the condenser heat transfer rate, the leaving condenser water temperature is recalculated. When the difference between the leaving condenser water temperature calculated on successive iterations is less than 0.0001°C, the solution is assumed to have converged. Warning messages are issued if the calculated solution for leaving condenser water temperature and/or part-load ratio falls outside the valid range specified for the chiller's performance curves. If these warnings are issued, the user may choose to extend the range for the performance curves (only if a small extension is required since model extrapolation may produce significant errors) or a different chiller and associated performance curves with extended performance range can be located and used for the simulation.

Note: Chiller:Electric:ReformulatedEIR objects and their associated performance curve objects are developed using performance information for a specific chiller and should almost always be used together for an EnergyPlus simulation. Changing the object input values, or swapping performance curves between chillers, should be done with extreme caution. For example, if the user wishes to model a chiller size that is different from the reference capacity, it is highly recommended that the reference flow rates be scaled proportionately to the change in reference capacity. Although this model can provide more accurate prediction than the Chiller:Electric:EIR model, it requires more performance data to develop the associated performance curves (at least 12 points from full-load performance and 7 points from part-load performance).

Although performance curve data sets for 160 chillers are provided in the EnergyPlus Reference DataSets (Chillers.idf and AllDataSets.idf), they may not meet the requirements for specific applications. One can develop performance curves from performance data using two available techniques (Hydeman and Gillespie 2002). The first technique is called the

Least-squares Linear Regression method and is used when sufficient performance data exist to employ standard least-square linear regression techniques. The second technique is called Reference Curve Method and is used when insufficient performance data exist to apply linear regression techniques. A detailed description of both techniques can be found in the reference mentioned above.

For any simulation time step, the chiller's available cooling capacity is calculated as follows:

$$\dot{Q}_{avail} = \dot{Q}_{ref} (ChillerCapFTemp)$$

where

\dot{Q}_{ref} = chiller capacity at reference conditions (reference temperatures and flow rates defined by the user), W

\dot{Q}_{avail} = available chiller capacity adjusted for current water temperatures, W

The model then calculates the evaporator heat transfer rate required to bring the entering chilled water temperature down to the leaving chilled water setpoint temperature (established using a SetpointManager object and referenced in the PlantLoop object). If this calculated heat transfer rate is greater than the heat transfer rate being requested by the plant equipment operation scheme, then the evaporator heat transfer rate is reset to the requested cooling rate.

The evaporator heat transfer rate is then compared to the available capacity. If the available chiller capacity is sufficient to meet the evaporator heat transfer rate, the leaving chilled water temperature is set equal to the chilled water setpoint temperature. If the requested evaporator heat transfer rate is larger than the available capacity the chilled water leaving the evaporator is allowed to float upward. For this case, the exiting chilled water temperature is calculated based on the water temperature entering the evaporator, the available cooling capacity, and the evaporator mass flow rate as follows:

$$T_{cw,l} = T_{cw,e} - \left(\frac{\dot{Q}_{avail}}{\dot{m}_{evap} C_{p,evap}} \right)$$

where

$T_{cw,l}$ = water temperature leaving the evaporator, °C

$T_{cw,e}$ = water temperature entering the evaporator, °C

\dot{m}_{evap} = evaporator mass flow rate, kg/s

$C_{p,evap}$ = specific heat of water entering evaporator at $T_{cw,e}$, J/kg-°C

The part-load ratio is then calculated as the ratio of the evaporator heat transfer rate to the available chiller capacity. The part-load ratio is not allowed to be greater than the maximum part-load ratio specified by the user or less than zero as follows:

$$PLR = MAX \left(0.0, MIN \left(\frac{\dot{Q}_{evap}}{\dot{Q}_{avail}}, PLR_{max} \right) \right)$$

where

PLR = part-load ratio

\dot{Q}_{evap} = load to be met by the chiller, W

PLR_{max} = maximum part-load ratio (specified by the user in the input data file)

Note that the maximum part-load ratio (PLR_{max} , specified in the Chiller:Electric:ReformulatedEIR object) used in the equation should be less than or equal to the maximum part-load ratio specified in the “Energy Input to Cooling Output Ratio Function of Part-Load Ratio” performance curve object.

The model assumes that the cooling load is met through chiller unloading down to the minimum unloading ratio. False loading (e.g. hot-gas bypass) is assumed to occur between the minimum unloading ratio and the minimum part-load ratio yielding constant electrical power consumption under these conditions. Below the minimum part-load ratio, the chiller cycles on/off to meet very small loads and the power consumption during the on cycle is the same as when the chiller is operating at the minimum part load ratio. When the chiller part-load ratio is less than the minimum part-load ratio, the on-off cycling ratio of the chiller is calculated as follows and is available as an output variable.

$$ChillerCyclingRatio = MIN\left(\frac{PLR}{PLR_{min}}, 1.0\right)$$

To properly account for chiller electric power consumption when PLR is less than the minimum unloading ratio, the PLR is reset to the greater of the PLR calculated above and the PLR at the minimum unloading ratio. The result is available as the output variable Chiller Part Load Ratio.

$$PLR = MAX(PLR, PLR_{MinUnloadRatio})$$

This revised PLR accounts for the “false loading” (e.g., hot-gas bypass) that is assumed to occur whenever the PLR (based on cooling load divided by available capacity) is less than the minimum unloading ratio specified. The amount of false loading on the chiller is calculated using this revised PLR and is reported as an output variable as follows:

$$\dot{Q}_{falseloading} = \left(\dot{Q}_{avail}\right)(PLR)(ChillerCyclingRatio) - \dot{Q}_{evap}$$

The electrical power consumption for the chiller compressor(s) for any simulation time step is then calculated using the following equation:

$$P_{chiller} = \left(\dot{Q}_{avail}\right)\left(\frac{1}{COP_{ref}}\right)(ChillerEIRFTemp)(ChillerEIRFPLR)(ChillerCyclingRatio)$$

where

$P_{chiller}$ = Chiller compressor power, W

COP_{ref} = Reference coefficient of performance, W/W

Heat rejected by the chiller condenser includes the heat transferred in the evaporator plus a portion or all of the compressor electrical energy consumption. For electric chillers with hermetic compressors, all compressor energy consumption is rejected by the condenser (compressor motor efficiency = $eff_{motor} = 1.0$). For chillers with semi-hermetic or open

compressors, only a portion of the compressor energy use is rejected by the condenser. The heat transfer rate for the chiller condenser is calculated as follows:

$$\dot{Q}_{cond} = (P_{chiller} * eff_{motor}) + \dot{Q}_{evap} + \dot{Q}_{falseloading}$$

where

\dot{Q}_{cond} = condenser heat transfer rate, W

eff_{motor} = compressor motor efficiency = fraction of compressor electrical energy consumption rejected as condenser heat

The above curve values are calculated based on the leaving condenser water temperature found through iteration. After obtaining the condenser heat transfer rate, the final leaving condenser water temperature is then calculated as:

$$T_{cond,l} = T_{cond,e} + \frac{\dot{Q}_{cond}}{\left(\dot{m}_{cond} * C_{p,cond} \right)}$$

where:

$T_{cond,l}$ = water temperature leaving the condenser, °C

$T_{cond,e}$ = water temperature entering the condenser, °C

\dot{m}_{cond} = mass flow rate through the condenser, kg/s

$C_{p,cond}$ = specific heat of water entering the condenser at $T_{cond,e}$, J/kg-°C

The final calculations determine the total heat transfer energy for the condenser and evaporator, as well as the total electric energy consumed by the chiller compressor motor(s) and condenser fan(s). The results are available as output variables.

$$Q_{cond} = \dot{Q}_{cond} * TimeStepSys * 3600$$

$$Q_{evap} = \dot{Q}_{evap} * TimeStepSys * 3600$$

$$E_{chiller} = P_{chiller} * TimeStepSys * 3600$$

$$E_{cond} = P_{cond} * TimeStepSys * 3600$$

where

Q_{cond} = chiller condenser heat transfer, J

Q_{evap} = chiller evaporator heat transfer, J

$E_{chiller}$ = chiller (compressor) electric consumption, J

E_{cond} = chiller condenser fan electric consumption, J

$TimeStepSys$ = HVAC system simulation time step, hr

3600 = conversion factor, sec/hr

Electric Reformulated EIR Chiller with Heat Recovery Option

Heat from the electric reformulated EIR chiller condenser may be recovered. The heat recovery water flow rate is specified by the user along with the input and output nodes connected to the heat recovery loop. The algorithms are identical to those used for Chiller:Electric and Chiller:Electric:EIR. Refer to the section entitled Chillers with Plant Heat Recovery for details.

References

Hydeman, M., N. Webb, P. Sreedharan, and S. Blanc. 2002. Development and Testing of a Reformulated Regression-Based Electric Chiller Model. ASHRAE Transactions HI-02-18-2.

Hydeman, M. and K.L. Gillespie. 2002. Tools and Techniques to Calibrate Electric Chiller Component Models. ASHRAE Transactions AC-02-9-1.

Hydeman, M., K. Gillespie, and R. Kammerud. 1997. PG&E's CoolTools project: A toolkit to improve evaluation and operation of chilled water plants. Presented at the CoolSense National Forum on Integrated Chilled Water Retrofits, Sep. 1997. Berkeley California: Lawrence Berkeley National Laboratory.

Engine Driven Chiller

The engine driven chiller (Object name: Chiller:EngineDriven) is the empirical model from the Building Loads and System Thermodynamics (BLAST) program. Fitting catalog data to a third order polynomial equations generates the chiller performance curves. Three sets of coefficients are required to model the open centrifugal chiller. Additional curve fits are required to model the engine. Because the model inherently involves the lower heating value of the fuel, a reference temperature is also involved in the calculations, which manufacturers present at 25°C.

The centrifugal chiller model Capacity Ratio Curve is a quadratic equation that determines the Ratio of Available Capacity to Nominal Capacity. The defining equation is:

$$CapRatio = \frac{Available\ Chiller\ Capacity}{Nominal\ Chiller\ Capacity} = A_1 + A_2\Delta T_{chiller} + A_3\Delta T_{chiller}^2$$

Where the Delta Temperature is defined as:

$$\Delta T_{chiller} = \frac{T_{cond,in} - T_{cond,in,design}}{TempRiseRatio} - (T_{evap,out} - T_{evap,out,design})$$

where the temperature rise coefficient which is defined as the ratio of the required change in condenser water temperature to a given change in chilled water temperature, which maintains the capacity at the nominal value. This is calculated as the following ratio:

$$TempRiseRatio = \frac{T_{cond,in,required} - T_{cond,in,rated}}{T_{evap,out,required} - T_{evap,out,rated}} - (T_{evap,out} - T_{evap,out,design})$$

Where:

$T_{cond,in,required}$ = Required entering condenser air or water temperature to maintain rated capacity.

$T_{cond,in,rated}$ = Rated entering condenser air or water temperature at rated capacity.

$T_{evap,out,required}$ = Required leaving evaporator water outlet temperature to maintain rated capacity.

$T_{evap,out,rated}$ = Rated leaving evaporator water outlet temperature at rated capacity.

The Power Ratio Curve is a quadratic equation that determines the Ratio of Full Load to Power. The defining equation is:

$$PowerRatio = B_1 + B_2PLR + B_3PLR^2$$

where the part load ratio, PLR is defined as:

$$PLR = \frac{chiller\ cooling\ load}{nominal\ chiller\ capacity}$$

The Load Ratio Curve is a quadratic equation that determines the Ratio of Full Load to Power. The defining equation is:

$$LoadRatio = C_1 + C_2CapRatio + C_3CapRatio^2$$

The evaporator heat transfer rate and the power required by the chiller are then calculated as:

$$Q_{evap} = AvailableChillerCap * PLR$$

$$Power = PowerRatio * LoadRatio * \frac{AvailableChillerCap}{RatedCOP}$$

The engine model was also developed for the BLAST program. It was adapted for use in EnergyPlus. This model is used for both the engine driven generator and the engine driven chiller. It uses the following set of equations all of which are quadratic fits to the PLR (Part Load Ratio) of the generator. The coefficients must be derived from manufacturers data.

$$\frac{chiller\ cooling\ load}{fuel\ energy\ input} = \frac{chiller\ cooling\ load}{(\dot{m}_{fuel} \{kg/s\} \cdot LHV \{J/kg\})} = a_1 + a_2PLR + a_3PLR^2$$

The exhaust gas temp and flow rate are used if a stack heat exchanger is used to recover waste heat from the exhaust. This temperature is the inlet temperature to the heat exchanger which is modeled in a UA-effectiveness form:

$$\frac{Total\ Exhaust\ heat}{fuel\ energy\ input} = \frac{Total\ Exhaust\ heat}{(\dot{m}_{fuel} \{kg/s\} \cdot LHV \{J/kg\})} = d_1 + d_2PLR + d_3PLR^2$$

$$\frac{Exhaust\ Gas\ Temp \{K\}}{fuel\ energy\ input} = \frac{Exhaust\ Gas\ Temp \{K\}}{(\dot{m}_{fuel} \{kg/s\} \cdot LHV \{J/kg\})} = e_1 + e_2PLR + e_3PLR^2$$

The exhaust flow rate is then calculated as:

$$\dot{m}_{exhaust} = \frac{Total\ Exhaust\ heat}{C_{p_{exhaust}} \cdot (T_{exhaust} - T_{reference})}$$

where $T_{\text{reference}}$ is the reference temperature for the fuel lower heating value, and is given as 25°C in manufacturer's data, and

$$T_{\text{stack}} = T_{\text{DesignMinExhaust}} + \frac{\left(T_{\text{exhaust}} - T_{\text{DesignMinExhaust}}\right)^2}{\exp\left(\frac{UA}{\dot{m}_{\text{exhaust}} C_{p_{\text{exhaust}}}}\right)}$$

Finally, heat recovered from the lube oil and the water jacket are accounted for as follows:

$$\frac{\text{Recoverable jacket heat}}{\text{fuel energy input}} = \frac{\text{Recoverable jacket heat}}{\left(\dot{m}_{\text{fuel}} \{ \text{kg} / \text{s} \} \cdot \text{LHV} \{ \text{J} / \text{kg} \} \right)} = b_1 + b_2 \text{PLR} + b_3 \text{PLR}^2$$

$$\frac{\text{Recoverable lube oil heat}}{\text{fuel energy input}} = \frac{\text{Recoverable lube oil heat}}{\left(\dot{m}_{\text{fuel}} \{ \text{kg} / \text{s} \} \cdot \text{LHV} \{ \text{J} / \text{kg} \} \right)} = c_1 + c_2 \text{PLR} + c_3 \text{PLR}^2$$

Ice Thermal Storage

Simple Ice Storage Model

Overview

This model (object name ThermalStorage:Ice:Simple) simulates the performance of an ice storage tank. The model is based on an integrated chiller and storage tank model developed for a special optimization project (Henze and Krarti 2002). This implementation removed the integrated chiller and allows the tank to be charged by any chiller configuration in an EnergyPlus plant loop.

This thermal storage model is based on a simple simulation of an ice storage tank with a fixed capacity. The tank is charged, or frozen, in an ice-on-coil configuration where ice builds up on the outside of the tubes carrying the brine or glycol solution from the chiller. There are two discharge (melt) options, internal or external. Internal melt uses the same fluid tubes for charging and discharging. External melt uses a separate fluid path for discharge such that the outer layers of ice melt first. The ice storage model includes an implied 3-way valve to control the amount of charge/discharge based on the incoming water temperature and the outlet node setpoint temperature. The tank is controlled using the PlantEquipmentOperation:ComponentSetpoint plant operation scheme, and requires that a setpoint be placed by a setpoint manager on the ice storage Plant Outlet Node. The model currently does not calculate any tank losses.

Model Description

Initialization

The storage tank is assumed to be fully charged (full of ice) at the beginning of each environment. The tank is then allowed to charge and discharge during the warmup days of the environment.

Control

The ice storage module is a passive component and will accept any flow rate placed on the inlet node. The load on the ice storage tank is determined by the inlet node flow rate, the inlet node temperature, and the outlet node setpoint temperature. A positive load indicates a request for cooling and the tank discharges if possible. A negative load indicates a request for charging the tank if possible. A zero load indicates the tank is dormant in which case all

flow bypasses the tank and the outlet node temperature is set to the inlet node temperature. The model includes an implied 3-way valve which controls the leaving water temperature to the outlet node setpoint temperature if possible.

Charging

When charging is requested, the following limits are calculated to determine the actual charging rate:

- 1) If the entering water temperature is greater than $-1C$, the charging rate is zero.
- 2) If the entering water temperature is greater than or equal to the outlet setpoint temperature, the charging rate is zero.
- 3) If the current fraction of ice stored is 1.0, the tank is fully charge, and the charging rate is zero.
- 4) If the requested charging rate times the current system timestep is more than the remaining uncharged storage capacity, the charging rate is limited to that required to completely fill the tank.
- 5) The maximum charging rate which the chilled water flow can provide is determined by the entering water temperature and flow rate and an outlet temperature which is the minimum of $-1C$ or the outlet setpoint temperature.
- 6) The maximum charging rate which the tank can accept is calculated by the following equations developed in the prior work (Henze and Krarti 2002):

$$UA_{IceCh} = (1.3879 - 7.6333*y + 26.3423*y^2 - 47.6084*y^3 + 41.8498*y^4 - 14.2948*y^5) * ITS_{NomCap} / TimeInterval / 10.0$$

where:

UA_{IceCh} = UA rate for charging

y = Current ice fraction stored

ITS_{NomCap} = Nominal storage capacity [GJ]

$TimeInterval$ = 3600 [s]

The smallest charging rate determined by the above rules is selected and the corresponding leaving water temperature is calculated.

Discharging

When discharging is requested, the following limits are calculated to determine the actual charging rate:

- 1) If the entering water temperature is less than $1C$, the discharge rate is zero.
- 2) If the entering water temperature is less than or equal to the outlet setpoint temperature, the discharge rate is zero.
- 3) If the current fraction of ice stored is 0.0, the tank is fully discharged, and the discharge rate is zero.
- 4) If the requested discharge rate times the current system timestep is more than the remaining charged storage capacity, the discharge rate is limited to that required to completely deplete the ice in the tank.
- 5) The maximum discharge rate which the chilled water flow can accept is determined by the entering water temperature and flow rate and an outlet temperature which is the maximum of $1C$ or the outlet setpoint temperature.
- 6) The maximum discharge rate which the tank can deliver is calculated by the following equations developed in the prior work (Henze and Krarti 2002):

For ice-on-coil internal melt:

$$UA_{IceDisCh} = (1.3879 - 7.6333*y + 26.3423*y^2 - 47.6084*y^3 + 41.8498*y^4 - 14.2948*y^5) * ITS_{NomCap} / TimeInterval / 10.0$$

For ice-on-coil external melt:

$$UA_{IceDisCh} = (1.1756 - 5.3689*y + 17.3602*y^2 - 30.1077*y^3 + 25.6387*y^4 - 8.5102*y^5) * ITS_{NomCap} / TimeInterval / 10.0$$

where:

$UA_{IceDisCh}$ = UA rate for discharging

y = 1 - Current ice fraction stored

ITS_{NomCap} = Nominal storage capacity [GJ]

$TimeInterval$ = 3600 [s]

The smallest discharge rate determined by the above rules is selected and the corresponding leaving water temperature is calculated.

References

Henze, Gregor P. and Moncef Krarti. 2002. Predictive Optimal Control of Active and Passive Building Thermal Storage Inventory, Final Report for Phase I: Analysis, Modeling, and Simulation. U.S. Department of Energy National Energy Technology Laboratory Cooperative Agreement DE-FC-26-01NT41255, December 2002.

Detailed Ice Storage Model

The following section describes how the detailed ice storage model works in EnergyPlus (object name ThermalStorage:Ice:Detailed).

Charging and Discharging Equation

The actual performance of the ice storage unit depends on the physical geometry, materials, and characteristics of the ice storage unit. In analyzing performance data trends from ice storage manufacturers, it was determined that the following equation would work well for the discharging process:

$$q^* = \left[C1 + C2(1 - P_c) + C3(1 - P_c)^2 \right] + \left[C4 + C5(1 - P_c) + C6(1 - P_c)^2 \right] \Delta T_{lm}^*$$

or

$$q^* = \left[C1 + C2(P_d) + C3(P_d)^2 \right] + \left[C4 + C5(P_d) + C6(P_d)^2 \right] \Delta T_{lm}^*$$

where:

$$q^* \equiv \frac{q \Delta t}{Q_{stor}}$$

$$\Delta T_{lm}^* \equiv \frac{\Delta T_{lm}}{\Delta T_{nominal}}$$

$$\Delta T_{lm} \equiv \frac{T_{brine,in} - T_{brine,out}}{\ln \left(\frac{T_{brine,in} - T_{brine,freeze}}{T_{brine,out} - T_{brine,freeze}} \right)}$$

q is the instantaneous heat transfer rate,

Q_{stor} is the total latent storage capacity,

Δt is a time step used in the curve fit (usually one hour),

$\Delta T_{nominal}$ is a nominal temperature difference ($18^\circ\text{F} = 10^\circ\text{C}$),

$T_{brine,in}$ is the tank brine inlet temperature,

$T_{brine,out}$ is the tank brine outlet temperature,

T_{freeze} is the freezing temperature of water or the latent energy storage material,

P_c is the fraction charged, and

P_d is the fraction discharged.

Likewise, the charging process of the ice storage device can be characterized by the following equation that is similar in form to the discharging equation:

$$q^* = \left[C1 + C2(P_c) + C3(P_c)^2 \right] + \left[C4 + C5(P_c) + C6(P_c)^2 \right] \Delta T_{lm}^*$$

Note that the time step might differ from the time step used within the EnergyPlus simulation. These are actually two separate time steps and are kept separate.

Charging Algorithm

During charging, manufacturers have stated that they attempt to charge the unit at the maximum rate until the unit is completely charged. This, of course, occurs during off-peak electric hours. Thus, once the setpoint has been scheduled for charging, the unit will charge at the maximum possible rate. This means that the flow rate through the ice storage device equals the flow to the component (or no bypass). The only time flow to the ice storage unit would be reduced is at the end of the charge cycle when more ice making capacity is available in a particular time step than is needed to fully charge the tank. In this case, the flow to the tank would be reduced appropriately to top off the tank storage capacity. We also have a setpoint goal for the outlet temperature of the ice storage device as defined by the setpoint schedule.

In solving the performance of the ice storage unit, we have effectively two equations (one of which is non-linear) and two unknowns. The equations are:

$$q^* = \left[C1 + C2(P_c) + C3(P_c)^2 \right] + \left[C4 + C5(P_c) + C6(P_c)^2 \right] \Delta T_{lm}^*$$

from which we can obtain the component load (q) and

$$q = \dot{m} C_p (T_i - T_o)$$

Both of these equations have q and T_o as unknowns. However, since the setpoint temperature is the goal for T_o , we can use this as an initial guess for T_o . Below is an outline of the algorithm:

- Initialize $T_o = T_{set}$
- Calculate LMTD*
- Calculate q^* from charging equation for the current percent charged (We will assume that the EnergyPlus time step is sufficiently small so that we do not need to find the average

percent charged for the time step. This was necessary when one hour time steps were used as in BLAST, but EnergyPlus generally uses relatively short time steps. Since there is already some iteration involved in the solution, we would like to avoid another layer of iteration if at all possible. One alternative that could be implemented would be to make a second pass with a closer average value based on what happens during the time step. This would effectively double the execution time for the model and would need to be justified before implementation.)

- Calculate $T_{o,new}$ and compare it to T_o
- Use $T_{o,new}$ to calculate a new LMTD* and iterate until T_o converges to some acceptable level

Charging would continue in subsequent time steps until the final state of the ice storage unit at the end of a particular time step is fully charged. If running a chiller would “overcharge” the tank, then the flow to the tank would be reduced (greater than zero bypass flow) while maintaining the same setpoint temperature coming out of the tank (though not necessarily out of the component).

Discharging Algorithm

During discharging, we cannot assume that all of the flow is sent through the ice storage unit and thus some of it may be bypassed around it locally. This ice storage model includes a built-in bypass leg to accommodate this without requiring the user to enter this additional information. This also allows the bypass leg/valve to be controlled by the ice storage unit.

While we cannot assume that all of the flow is sent through the ice storage unit, we can use that as an initial guess in order to determine the current performance of the ice storage system. Most of the discharging algorithm then becomes very similar to the charging process.

In solving the performance of the ice storage unit, we have effectively two equations (one of which is non-linear) and two unknowns. The equations are:

$$q^* = \left[C1 + C2(P_d) + C3(P_d)^2 \right] + \left[C4 + C5(P_d) + C6(P_d)^2 \right] \Delta T_{lm}^*$$

from which we can obtain the component load (q) and

$$q = \dot{m} C_p (T_i - T_o)$$

Both of these equations have q and T_o as unknowns. However, since the setpoint temperature is the goal for T_o , we can use this as an initial guess for T_o . Below is an outline of the algorithm:

- Initialize $T_o = T_{set}$
- Calculate LMTD*
- Calculate q^* from charging equation for the current percent charged (we will assume that the EnergyPlus time step is sufficiently small so that we do not need to find the average percent charged for the time step; this was necessary when one hour time steps were used as in BLAST, but EnergyPlus generally uses relatively short time steps)
- Calculate $T_{o,new}$ and compare it to T_o
- Use $T_{o,new}$ to calculate a new LMTD* and iterate until T_o converges to some acceptable level
- Once T_o has converged, we need to compare this value again to T_{set} . If T_o is greater than or equal to T_{set} , then we either just met the load (equal) or can't quite meet it (greater than). In this case, we simply leave T_o as is because we cannot meet the setpoint temperature.

- If T_o is less than T_{set} , then we have more capacity available than we need. In this case, we need to bypass some of the flow. Since the load on the ice storage device and the outlet temperature are not changing (we are just reducing the flow), we only need to split the flow and do not need to recalculate the action of the ice storage device. Some systems may be slightly dependent on the actual flow through the device itself. However, in an actual application, this only means that a slightly different amount will bypass the device. The overall energy impact will be the same and thus it is not necessary to be concerned about flow rate dependence.

Discharging would continue in subsequent time steps until the final state of the ice storage unit at the end of a particular time step is fully discharged.

References

Strand, R.K. 1992. "Indirect Ice Storage System Simulation," M.S. Thesis, Department of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign.

Coils

Chilled-Water-Based Air Cooling Coil

The input object Coil:Cooling:Water is simpler than the detailed geometry model. The simple model is useful in early stages of simulation because in order to provide this simulation capability, a coil model that predicts changes in air and water flow variables across the coil based on the coil geometry is required, but for the user is not required to input all the details of the coil geometry: Area, Number of tubes, Fin spacing etc, is a very cumbersome task especially in early stages of simulation. A good simulation program should ask the user bare minimum number of inputs keeping into account accuracy of the model, hence a model is required which is able to convert the large number of coil geometric details into a small set of auto sized thermodynamic inputs. A greatly simplified schematic of enthalpy and temperature conditions in a counter flow cooling/dehumidifying coil is shown in the following schematic figure 1. The input required to model the coil includes only a set of Thermodynamic design inputs, which require no specific manufacturer's data. The coil simulation model is essentially the modification of one presented by Elmahdy and Mitalas (1977), TRNSYS, 1990 and Threlkeld, J.L. 1970. The model calculates the UA values required for a Dry, Wet and Part Wet & Part Dry Coil and iterates between the Dry and Wet Coil to output the fraction wet. There are two modes of flow operation for this model: Cross Flow, which is widely applicable in HVAC systems and the second being Counter flow mode. The default value in program is set up for Cross Flow. In addition the coil has two modes of analysis: Simple Analysis and Detailed Analysis. The Simple analysis mode operates the coil as either wet or dry while the detailed mode simulates the coil as part wet part-dry, however the execution time for detailed mode is noticeably higher. The default input for the Analysis is **SimpleAnalysis**.

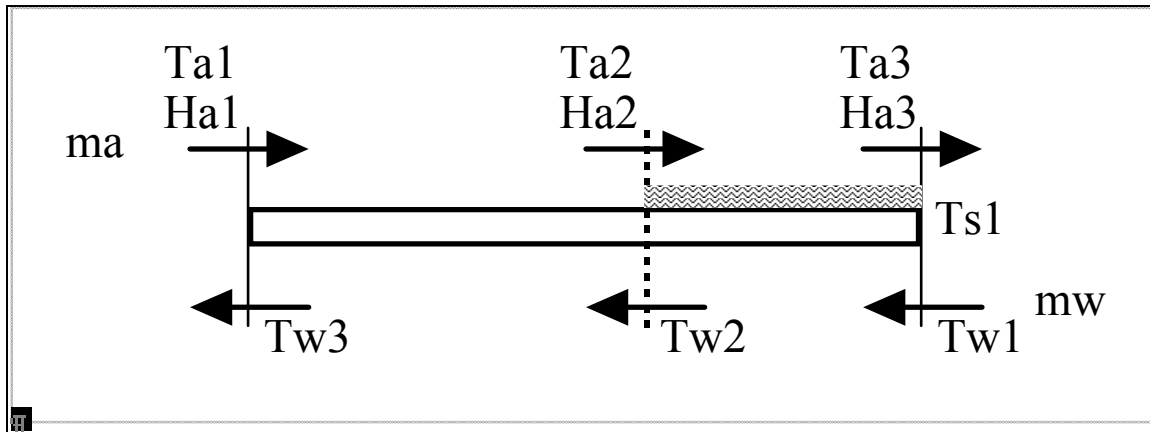


Figure 132. Simplified Schematic of Cooling/Dehumidifying Coil

Heat Transfer and Energy Balance

The cooling coil may be completely dry, completely wet with condensation, or it may have wet and dry sections. The actual condition of the coil surface depends on the humidity and temperature of the air passing over the coil and the coil surface temperature. The part-wet part-dry case represents the most general scenario for the coil surface conditions. There are subroutines present in the model for both the dry and wet regions of the coil, and a subroutine that iterates between the dry and wet subroutines to calculate the fraction surface wet of the coil. For each region the heat transfer rate from air to water may be defined by the rate of enthalpy change in the air and in the water. The rates must balance between each medium for energy to be conserved.

Model Description

The Model has two blocks: 1st = Design Block with the Design Inputs. This block calculates the Design U-Factor Times Area Value (UA) values required by the model. Using these UA values the model simulates the operating conditions. The operating block is the one containing the operating conditions, the conditions at which the coil operates. Following is the list of Design and Operating inputs and subsequently the Design and Operating variables used in the model.

Table 40. Design Inputs (User Inputs)

Input Field	Description
DesWaterVolFlowRate:	Maximum Water Volume Flow Rate
DesAirVolFlowRate:	Maximum Air Volume Flow Rate
DesInletWaterTemp:	Inlet Water Temperature at Design Condition
DesInletAirTemp:	Inlet Air Temperature at Design Condition
DesOutletAirTemp:	Outlet Air Temperature at Design Condition
DesInletAirHumRat:	Inlet Air Humidity Ratio at Design Conditions
DesOutletAirHumRat:	Outlet Air Humidity Ratio at Design Conditions.

Table 41. Operating Conditions (From Nodes -- not user inputs)

Condition Variable	Description
InletWaterMassFlowRate:	Entering Water Mass Flow Rate at operating condition
InletWaterTemp:	Inlet Water Temperature at operating condition

InletAirMassFlowRate:	Entering Air Mass Flow Rate at operating condition
InletAirTemp:	Inlet Air Temperature at operating condition
InletAirHumRat:	Entering air humidity ratio at operating conditions

Intermediate calculated U-Factor Times Area Values: The Crux of the Model

The various U-Factor Times Area values (UA) required by this model are calculated from the above inputs, which are explained later in the document. The various UA are:

Table 42. UA Descriptions of Model

UA Variable Name	Description
UACoilTotal:	Overall heat transfer coefficient (W/C)
UACoilInternal:	Overall internal UA (W/C)
UACoilExternal:	Overall external UA (W/C)
UACoilInternalPerUnitArea:	Internal overall heat transfer coefficient (W/m ² C)
UAWetExtPerUnitArea:	External overall heat transfer coefficient (W/m ² C)
UADryExtPerUnitArea:	External overall heat transfer coefficient (W/m ² C)

The UA values are calculated assuming Wet Coil at Design Condition. Following are few important calculations to understand the working of the model. The model is basically divided into Two Blocks: the Design Block and the Operating Block.

The Design Block is a One Time Calculation and in an Hourly simulation the code is skipped subsequently. The aim of the Design Block is to calculate the Coil UA for use in the operating Block.

Design Block Calculations:

The design block has the code for calculating the six Coil UA values required by the operating block. Reasonable assumptions have been made in the calculations to maintain the simplicity of the model.

Calculating Coil UA Enthalpy Based: Heat Transfer in Wet Coil is based on Enthalpy rather than temperature to take into account latent effects. While heat transfer rates are commonly expressed as the product of an overall heat transfer coefficient, UA, and a temperature difference, the use of enthalpy-based heat transfer calculations requires an enthalpy-based heat transfer coefficient which is defined as = DesUACoilTotalEnth and hence the equation.

$Q = \text{DesUACoilTotalEnth} * (H_1 - H_2)$, The value of Q is calculated using product of air mass flow rate and difference in inlet and outlet air enthalpies at design conditions.

Where $\text{DesUACoilTotalEnth} = \text{CoilUA} / \text{cp}$ and CoilUA is the conventional heat transfer coefficient and cp = specific heat across enthalpy difference. We use the following two heat transfer equations for calculating the DesUACoilTotalEnth.

$$\dot{Q}_{\text{Maximum}} = \text{Min}(\text{Air}, \text{Water})_{\text{Capacity}} * (\text{DesInletAirEnth} - \text{DesSatEnthAtWaterInTemp}) \quad (255)$$

$$\dot{Q}_{\text{Maximum}} = \text{CoilUA}_{\text{EnthalpyBased}} * (\text{DesInletAirEnth} - \text{DesSatEnthAtWaterInTemp}) \quad (256)$$

Iterate on equation (255) and (256), i.e., on the two value of maximum Heat Transfer till convergence achieved to calculate the value Coil UA Enthalpy Based. The variables used in above equations are calculated as below.

$$\text{DesSatEnthAtWaterInTemp} = \text{PsyHFnTdbW}(\text{DesInletWaterTemp}, \text{DesHumRatioAirInlet})$$

Psy**Fn** are the Psychrometric functions from Energy Plus for calculating Psychrometric properties of air. The above equation (0.1) and (0.2) give the value for Coil UA Enthalpy Based. We now calculate the value of Coil ByPass Factor based on enthalpies of the stream at various corresponding temperatures.

$$\text{DesByPassFactor} = \frac{\text{DesOutletAirEnth} - \text{DesAirApparatusDewPtEnth}}{\text{DesInletAirEnth} - \text{DesAirApparatusDewPtEnth}} \quad (257)$$

The variable DesAirApparatusDewPtEnth is calculated from DesAirTempApparatusDewPt which is intern calculated by the following procedure as shown below.

$$\text{Temp}_{\text{ApparatusDewPtEstimate}} = \text{Des}_{\text{InletAirTemp}} - \text{Slope} \times (\text{W}_{\text{AirIn}} - \text{W}_{\text{ApparatusDewPt}}) \quad (258)$$

$$\text{Where Slope} = \frac{\text{DesInlet}_{\text{AirTemp}} - \text{DesOutlet}_{\text{AirTemp}}}{\text{DesInlet}_{\text{AirHumRatio}} - \text{DesOutlet}_{\text{AirHumRatio}}}, \text{ and } \text{W} = \text{Humidity Ratio} \quad (259)$$

Iterate between -- equation (258) and the initial guess value of DesAirTempApparatusDewPt till convergence is achieved. The initial guess value of the above variable is

$$\text{DesAirTempApparatusDewPt} = \text{PsyTdpFnWPb}(\text{DesOutletAirHumRat}, \text{Patm})$$

Where the variables used have usual meaning.

Calculating Coil UA External:

The procedure for calculating the variable Coil UA External is best described by the familiar Fortran programming -- **If Then Else** block, as follows. External UA, which is the airside coefficient, is calculated assuming that surface temperature is at apparatus dewpoint.

IF (DesAirApparatusDewPtEnth ≤ DesSatEnthAtWaterInTemp) **THEN**

$$\text{CoilUA}_{\text{External}} = \text{DesCoilUA}_{\text{EnthalpyBased}} * \text{Cp}_{\text{Air}} \quad (260)$$

ELSE

$$\text{CoilUA}_{\text{External}} = -\text{Log}(\text{ByPass}_{\text{Factor}}) * (\text{Cap}_{\text{Air}}) \quad (261)$$

END IF

Where Cap_Air is the product of Design Air Mass Flow rate and Specific Heat Capacity of air and Cp_air is the specific heat capacity of air at particular condition.

Calculating Coil UA Internal:

The liquid-side coefficient, UA Coil Internal is calculated from enthalpy-based overall coefficient and airside coefficient. The Enthalpy Based UA is calculated as shown in equation (255) and (256) and airside coefficient which is the Coil UA external is calculated in equation (260) and (261).

$$\text{CoilUA}_{\text{Internal}} = \frac{\text{CpSat}_{\text{Intermediate}}}{\frac{1}{\text{DesCoilUA}_{\text{EnthalpyBased}}} - \frac{\text{Cp}_{\text{Air}}}{\text{CoilUA}_{\text{External}}}} \quad (262)$$

In the above equation the variables Cp_air, Coil UA External, Coil UA Enthalpy Based have already been calculated and are known from previous equations. The variable CpSat_Intermediate is an intermediate fictitious specific heat and is estimated using entering air dewpoint and water temperature. The calculation is shown below.

$$CpSat_{Intermediate} = \frac{EnthAir_{DesDewPoint} - EnthSat_{DesWaterInTemp}}{DewPt_{DesTempAir} - TempWater_{DesInlet}} \quad (263)$$

The variables used in the above equation are calculated from known and earlier calculated temperatures, using the psychrometric functions in Energy Plus.

Calculating Overall Heat Transfer Coefficient:

The overall heat transfer coefficient is calculated from a combination of series or parallel resistances, similar to thermal resistances. In the case of our coil model, total UA is a parallel combination of External and Internal UA values which are calculated in equations (260) to equation (262). Hence the equation for UA Coil Total is as follows.

$$CoilUA_{Total} = \frac{1}{\frac{1}{CoilUA_{Internal}} + \frac{1}{CoilUA_{External}}} \quad (264)$$

In order to calculate the value of Heat Transfer coefficients per unit area we need to get a good approximate for the coil area required. This is achieved by using the simple heat transfer correlation

$$Q_{Des} = CoilUA_{Total} \times CoilArea \times LMTD_{Cross,Counter-Flow} \quad (265)$$

All the variables in the above equation are calculated from design conditions, the variable LMTD can be calculated for Cross and Counter flow mode of coil depending on choice. The default set up in the program is for cross flow since most air-conditioning applications are in that mode.

Calculating Coil UA Internal/Unit area of Coil:

Internal overall heat transfer coefficient (W/m² C) is approximated as overall internal UA divided by Total surface area of coil.

$$CoilUA_{InternalPerUnitArea} = \frac{CoilUA_{Internal}}{Area} \quad (266)$$

Calculating Dry UA External/Unit area of Coil:

The value of Dry UA external is calculated from rearranging the equation (266):

$$DryUA = \frac{Area}{\frac{1}{DryUA_{ExtPerUnitArea}} + \frac{1}{CoilUA_{Internal}}} \quad (267)$$

Rearranging equation (267) and making a reasonable assumption that DryUA of Coil is equivalent of the Coil UA Total for this model. This assumption yields reasonably close results to the detailed coil, which calculates all UA based on precise geometric inputs. We get the following equation (268) for the Dry UA external per unit area for the coil:

$$DryUA_{ExtPerUnit Area} = \frac{CoilUA_{Total} \times CoilUA_{Internal}}{Area \times CoilUA_{Internal} - CoilUA_{Total}} \quad (268)$$

Calculating Wet UA External/Unit Area:

Since the Coil UA's are calculated assuming it is Wet Coil at design condition, we make a reasonable assumption as in equation (269) to calculate the value of Wet UA External per unit area. This assumption gives remarkably close values to what is expected if detailed geometric variables are used to calculate the same wet coil UA per unit area.

$$WetUA_{ExternalPerUnit Area} = \frac{CoilUA_{Internal}}{Area} \quad (269)$$

The Design Block calculations end at this part, we now have all the essential UA values to proceed with the 5 operating conditions to calculate the coils performance, the outlet states of water and air given the inlet conditions and the Heat Transfer: sensible and total. Hence the design block in the coil model is essentially being utilized to calculate the coil UA so that the operating block could use those values of UA.

Operating Block Calculations:

There are two modes of coil analysis in the operating block. They are the Simple analysis mode and the detailed analysis mode. The simple analysis mode assumes the coil to be either all wet or either all dry and execute the model, on the other hand the detailed mode checks for part wet part dry mode of operation and reports surface area wet fraction of coil, however the program execution time in detailed mode is noticeably higher.

The operating block for Detailed Mode Analysis of this coil model is divided into three modes of coil performance. The modes being

- Coil is completely dry: There is no moisture condensation on the coil surface and the coil is a dry coil. This is an extreme condition when the entering air has very low humidity ratio or is dry air.
- Coil is completely wet: The entire coil is wet due to complete condensation on the surface of the coil.
- Part Wet Part Dry Mode: This is the usual/frequent mode of operation of coil, as shown in figure 1, where part of the coil at entry of air is dry and as air cools condensation occurs and part of the coil becomes wet.

The Part Wet Part Dry Mode of operation is essentially a function the Coil Completely Dry and Coil Completely Wet mode. This subroutine iterates between the Dry Coil and the Wet Coil to give outputs, a detailed explanation is given later in the document. The operating block requires 5 inputs, which are mentioned earlier in the document. These inputs are automatically generated from the node connections in Energy Plus. The user does not have to input any information to run this coil model.

The option to identify which mode of operation the coil should perform ie, for a given set of inputs would the coil be Dry, Wet or Part Wet Part Dry, is decided by set of conditions described below.

- **IF (Temperature Dewpoint Air < Water Inlet Temperature) THEN** the coil is Dry and we call the Subroutine Coil Completely Dry. In this case outlet temperature of air would be higher than the air dewpoint and hence there would be no condensation.
- **IF (Temperature Dewpoint Air > Water Inlet Temperature) THEN** the coil is completely wet, call subroutine Coil Completely Wet, it is assumed that moisture condensation occurs over completely surface of the coil. However we go ahead and check for the coil being partially wet with the following condition.

- **IF (AirDewPointTemp < AirInletCoilSurfTemp) THEN**, the coil is Partially Wet because there is possibility that air temperature will go below its dewpoint and moisture will condense on latter part of the cooling coil.

The Operating Block for Simple Mode Analysis is divided into two modes of coil performance, the two modes being

- Coil is completely dry: There is no moisture condensation on the coil surface and the coil is a dry coil.
- Coil is completely wet: The entire coil is wet due to complete condensation on the surface of the coil.

The option to identify which mode of operation the Simple mode analysis should perform ie, for a given set of inputs would the coil be Dry or Wet is decided by set of conditions described below.

- **IF (Temperature Dewpoint Air < Water Inlet Temperature) THEN** the coil is Dry and we call the Subroutine Coil Completely Dry. In this case outlet temperature of air would be higher than the air dewpoint and hence there would be no condensation.
- **IF (Temperature Dewpoint Air > Water Inlet Temperature) THEN** the coil is completely wet, call subroutine Coil Completely Wet, it is assumed that moisture condensation occurs over completely surface of the coil. However we go ahead and check for the coil being partially wet with the following condition.

The above is a simple mode of analysis and the results are very slightly different from the detailed mode of analysis. The algorithms used in Simple mode and the Detailed mode are identically similar. The surface area wet fraction in the coil is reported as 1.0 or 0.0 for wet or dry coil respectively. The program defaults to simple mode of analysis for enabling higher execution speed.

Effectiveness Equations:

There are two modes of flow for the coil, Counter Flow mode or the Cross Flow mode, default set up is as cross flow since most air condition applications have cross flow heat exchangers. According to the mode of flow the following NTU - Effectiveness relationships are used to calculate coil effectiveness, which is used later by all the three modes (Dry, Wet, Part Wet) for calculating air outlet conditions and heat transfer.

Following are the relations used for calculating effectiveness equation for the Heat exchangers.

Counter Flow Heat Exchanger: Effectiveness Equation:

$$\eta_{CounterFlow} = \frac{(1 - \text{Exp}(-NTU \times (1 - \text{Ratio}_{StreamCapacity})))}{1 - \text{Ratio}_{StreamCapacity} \times \text{Exp}(-NTU \times (1 - \text{Ratio}_{StreamCapacity}))} \quad (270)$$

In Equation (270) the variable Ratio_StreamCapacity is defined as below

$$\text{Ratio}_{StreamCapacity} = \frac{\text{MinCapacity}_{Stream}}{\text{MaxCapacity}_{Stream}} \quad (271)$$

In equation (271) capacity of stream is defined as below in equation (272)

$$(\text{Min}, \text{Max}) \text{ Capacity}_{Stream} = (\text{MassFlowRate} \times \text{Cp})_{air, water} \quad (272)$$

NTU: equation (270), is defined as the Number of Transfer Units, it is a function of Coil UA and the Minimum Capacity of Stream. The Coil UA is a variable in this equation and depends on which mode of the coil operation (Dry, Wet, Part Wet) is calling upon equation (270), i.e., if

it is Coil Completely Dry calling upon the effectiveness equation with the value of Dry UA total, which in our case is defined as CoilUA_total. Equation (273) gives definition for NTU.

$$NTU = \frac{CoilUA}{MinStream_{Capacity}} \quad (273)$$

Cross Flow Heat Exchanger: Effectiveness Equation:

$$\eta_{CrossFlow} = 1 - EXP \left\{ \frac{Exp(-NTU \times Ratio_{StreamCapacity} \times NTU^{-0.22}) - 1}{Ratio_{StreamCapacity} \times NTU^{-0.22}} \right\} \quad (274)$$

The variables in the above equation have already been defined earlier. Depending on the mode of operation of the coil model the cross or the counter flow equations are used to calculate the effectiveness.

Coil Outlet Conditions:

Calculating the Outlet Stream Conditions using the effectiveness value from equation (270) or (274) depending on the mode of flow. The energy difference between the outlet and inlet stream conditions gives the amount of heat transfer that has actually take place. Temperature of air and water at outlet to the coil is given as in following equations

$$TempAir_{Out} = TempAir_{inlet} - \eta_{cross,counter} \times \frac{MaxHeatTransfer}{StreamCapacity_{Air}} \quad (275)$$

$$TempWater_{Out} = TempWater_{Inlet} + \eta_{Cross,counter} \times \frac{MaxHeatTransfer}{StreamCapacity_{Water}} \quad (276)$$

In the above equations (275) and (276) the maximum heat transfer is calculated as shown in the following equation

$$MaxHeatTransfer = MinStream_{Capacity} \times (TempAir_{Inlet} - TempWater_{Inlet}) \quad (277)$$

Coil Completely Dry Calculations: (operating block)

Since the coil is dry, the sensible load is equal to latent load and the same with the humidity ratios at inlet and outlet, as in equation (278) and (279).

$$QSensible_{DryCoil} = QTotal_{DryCoil} \quad (278)$$

$$HumRatio_{Inlet} = HumRatio_{Outlet} \quad (279)$$

Total Heat Transfer in dry coil is as follows:

$$Q_{TotalDryCoil} = CapacityAir \times (AirTemp_{In} - AirTemp_{Outlet}) \quad (280)$$

The variables in the above equation are calculated in the earlier in equations (275) and (276) to give the total cooling load on the coil.

Coil Completely Wet Calculations: (operating block)

In wet coil we need to account for latent heat transfer, hence calculations are done using enthalpy of air and water instead of stream temperatures. Hence we need to define coil UA for the wet coil based on enthalpy of the operating streams and not design streams.

Similar to equations (275) and (276) we calculate the air outlet enthalpy and water outlet enthalpy i.e. by replacing temperature with enthalpy of the respective streams. The input variable for Coil UA in equation (273) for calculating NTU, in this case it would be enthalpy based and is given as shown in equation (281)

$$CoilUA_{EnthalpyBased} = \frac{1}{\left(\frac{CpSat_{intermediate}}{UACoil_{Internal}} + \frac{Cp_{Air}}{UACoil_{External}} \right)} \quad (281)$$

Total Coil Load in case of Wet Coil is the product of mass flow rate of air and enthalpy difference between the inlet and outlet streams as given in the following equation

$$Q_{Total} = \dot{M}_{air} \times (EnthAir_{Inlet} - EnthAir_{Outlet}) \quad (282)$$

Once the enthalpy is known the outlet temperatures and outlet humidity ratios of the wet coil are calculated as in equations below.

IF (TempCondensation < PsyTdpFnWPb(InletAirHumRat ,Patm)) THEN

$$AirTemp_{Out} = AirTemp_{inlet} - (AirTemp_{inlet} - Condensation_{Temp}) \times \eta \quad (283)$$

and

OutletAirHumidityRatio = PsyWFnTdbH(OutletAirTemp,EnthAirOutlet)

ELSE

There is no condensation and hence the inlet and outlet Hum Ratios are equal, and outlet temperature is a function of outlet air enthalpy as below

OutletAirTemp = PsyTdbFnHW (EnthalpyAirOutlet, OutletAirHumRat)

and

OutletAirHumRat = InletAirHumRat

ENDIF

Effectiveness eta used in equation (283) is defined in equation (284) and Condensation Temperature is calculated using Psychrometric function as in equation (285) and (286).

$$\eta = 1 - \exp \left\{ - \frac{CoilUA_{External}}{Capacitance_{Air}} \right\} \quad (284)$$

$$Condensation_{Temp} = PsyTsatFnHPb(Enth_{AirCondensateTemp}, Patm) \quad (285)$$

$$Enth_{AirCondensateTemp} = Enth_{AirInlet} - \frac{(Enth_{AirInlet} - Enth_{AirOutlet})}{\eta} \quad (286)$$

Once the air outlet temperature are known, then sensible load is calculated as a product of capacitance of air and temperature difference at inlet and outlet, as in equation (287)

$$Q_{Sensible} = Capacitance_{Air} \times (AirTemp_{Inlet} - AirTemp_{Outlet}) \quad (287)$$

Coil Part Wet Part Dry Calculations: (operating block)

The Coil would perform under part wet part dry conditions when Air Dewpoint Temperature is less than Coil surface temperature at inlet to air. In this case part of the coil used value of Dry UA for heat transfer and part the coil used Wet UA value for heat transfer.

This problem is solved utilizing the fact that the Exit conditions from the Dry Part of the Coil would become the inlet conditions to the wet part of the coil (Figure-1) and the coil model determines by iteration what fraction of the coil is wet and based on that it calculates the areas and subsequently the UA values of that dry and wet part, based on the area of the dry and wet part respectively. Explained below are the steps followed to the estimating the wet dry behavior of the coil.

- Iterate between the Dry Coil and the Wet Coil. First calculate Coil Completely Dry performance by estimating the wet dry interface water temperature using equation (288) and inputting this variable as the water inlet temperature to dry Coil.

$$WetDryInterface_{WaterTemp} = WaterTemp_{Inlet} + Area_{WetFraction} * (WaterTemp_{Outlet} - WaterTemp_{Inlet}) \quad (288)$$

The value of Surface Area Wet fraction is estimated initially as follows

$$Area_{WetFractionEstimate} = \frac{AirDewPt_{Temp} - InletWater_{Temp}}{OutletWater_{Temp} - InletWater_{Temp}} \quad (289)$$

For the above mentioned iteration the value of Coil UA for Wet and Dry part need to be varied according to the new respective area of the wet and dry parts. This estimate of Wet and Dry area is a product of the estimated Surface Area Fraction and total coil external area, which keeps varying as will be explained further in the document.

UA value for Dry part of the Coil is estimated as below.

$$DryUA_{external} = \frac{SurfArea_{Dry}}{\frac{1}{CoilUADry_{ExternalPerUnitArea}} + \frac{1}{CoilUAInternal_{PerUnitArea}}} \quad (290)$$

Where Surface Area Dry =(Total Coil Area – Wet Part Area), where the Wet part area is the product of Surface fraction Wet and Total Coil Area.

UA value for the Wet part of the Coil requires Wet UA external and Wet UA Internal, which are calculated as below.

$$WetPartUA_{External} = CoilUAWet_{ExternalPerUnitArea} \times SurfaceAreaWet \quad (291)$$

$$WetPartUA_{Internal} = CoilUAWet_{InternalPerUnitArea} \times SurfaceAreaWet \quad (292)$$

It is essential to remember that the mode of calculation for the coils remains the same as in completely wet and completely dry mode, only the UA values and water, air outlet and inlet values change.

Now Iterate between the Dry Coil and wet Coil with the above respective UA, and usual operating inputs except the variable water inlet temperature for dry Coil is replaced with Wet Dry Interface Water temperature, and in the Wet Coil the Outlet Air Temperature from dry Coil is the inlet air temperature to Wet Coil. The iteration proceeds till the Outlet Water

Temperature from Wet Coil equals the Wet Dry Interface Water Temp, which is the input to Dry Coil.

Dry Part Inputs: (changed operating inputs) :Iteration Case 1: Explained In Programming Fashion:

CALL CoilCompletelyDry (WetDryInterfcWaterTemp, InletAirTemp, DryCoilUA, & OutletWaterTemp, WetDryInterfcAirTemp, WetDryInterfcHumRat, & DryCoilHeatTranfer).

Input the calculated values calculated by Dry Coil above into Wet Coil below. The variables have been highlighted in color red and blue.

CALLCoilCompletelyWet (InletWaterTemp, WetDryInterfcAirTemp, WetDryInterfcHumRat WetPartUAIinternal,WetPartUAExternal, & EstimateWetDryInterfcWaterTemp, OutletAirTemp, OutletAirHumRat, & WetCoilTotalHeatTransfer, WetCoilSensibleHeatTransfer, & EstimateSurfAreaWetFraction, WetDryInterfcSurfTemp)

Iterate Between the above two Wet and Dry Coil calls until the two variables in blue ie **WetDryInterfcWaterTemp = EstimateWetDryInterfcWaterTemp**. The key is to have the difference between the variables (WetDryInterfcWaterTemp – OutletWaterTemp) in Dry Coil equal to (InletWaterTemp–EstimatedWetDryInterfcWaterTemp) in Wet Coil. This equality quantized the relative part of coil that is dry and part that is wet on the basis of heat transfer that has occurred.

After the above convergence check for the coil being dry otherwise iterate to calculate surface fraction area wet.

IF

$$\left\{ (AreaFraction_{Wet} \leq 0.0) \text{ and } (WetDryInterface_{SurfTemp} > AirDewPt) \right\} \quad (293)$$

THEN CoilCompletelyDry

If Equation (293) is satisfied then Coil is Dry and simply output the value for Dry Coil calculated else the coil is partially wet and then iterate to find the surface fraction area wet. Start with the initially guess value of surface area fraction (equation (289)) wet and iterate on the entire loop starting from equation (289) until the Wet Dry Interface Temperature equals the Air Dewpoint Temperature. The value of Surface Area fraction wet at which the interface air temperature equals is dewpoint is the transition point from wet to dry and gives the % of coil that is dry and % that is wet.

Graphs Showing the Performance of the coil model at optimum operating conditions are shown below. All values of variable used have been normalized.

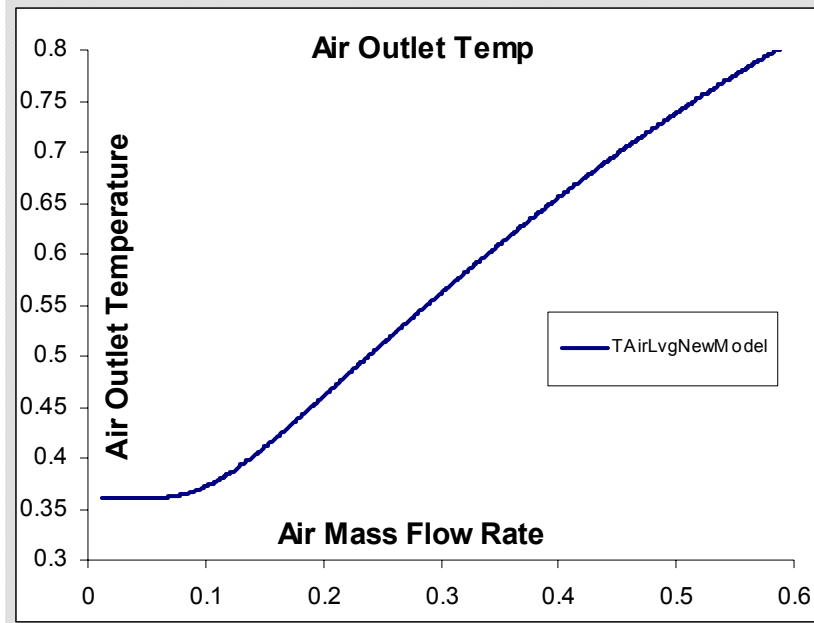


Figure 133. Air Outlet Temperature Vs Air Mass Flow Rate

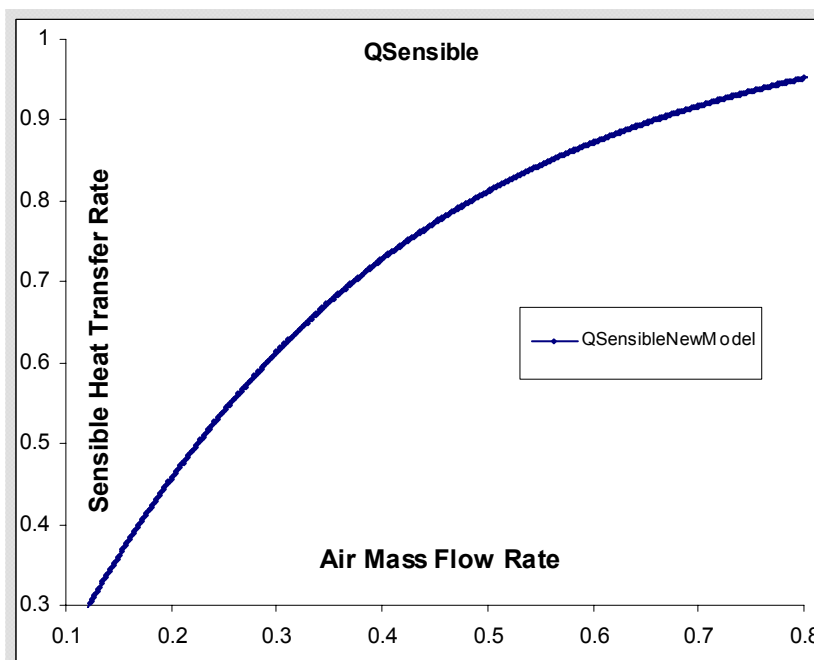


Figure 134. Sensible Load variations Vs Air mass Flow Rate

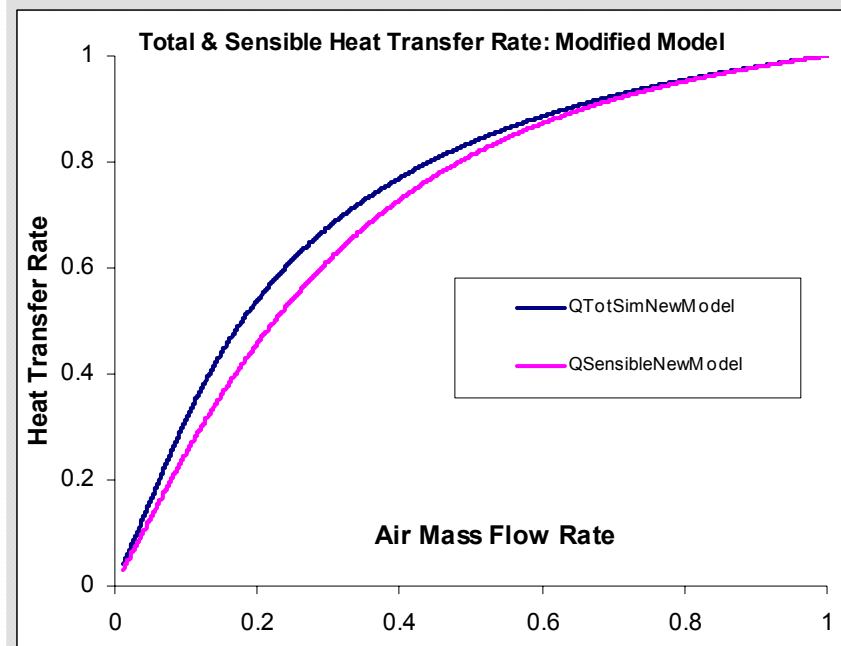


Figure 135. Total and Sensible Load variations Vs Air Mass Flow Rate

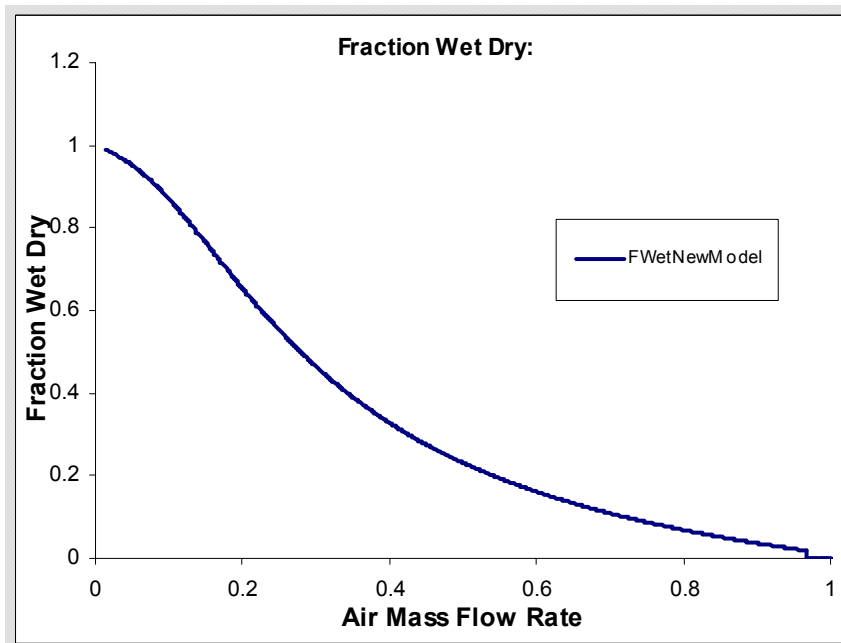


Figure 136. Surface Area Fraction Wet Vs Air Mass Flow Rate

References

IBPSA BuildSim-2004. 2004. Colorado Boulder: An Improvement of Ashrae Secondary HVAC toolkit Simple Cooling Coil Model for Building Simulation, Rahul J Chillar, Richard J Liesen M&IE, UIUC.

Stoecker, W.F. <dates unspecified> Design of Thermal Systems,; ME 423 Class Notes, M&IE Dept UIUC.

Brandemeuhl, M. J. 1993. HVAC2 Toolkit: Algorithms and Subroutines for Secondary HVAC Systems Energy Calculations, ASHRAE.

Elmahdy, A.H. and Mitalas, G.P. 1977. "A Simple Model for Cooling and Dehumidifying Coils for Use In Calculating Energy Requirements for Buildings ASHRAE Transactions, Vol.83 Part 2, pp. 103-117.

Threlkeld, J.L. 1970. Thermal Environmental Engineering, 2nd Edition, Englewood Cliffs: Prentice-Hall, Inc. pp. 254-270.

ASHRAE Secondary HVAC Toolkit TRNSYS. 1990. A Transient System Simulation Program: Reference Manual. Solar Energy Laboratory, Univ. Wisconsin-Madison, pp. 4.6.8-1 - 4.6.8-12.

Kays, W.M. and A.L. London. 1964. Compact Heat Exchangers, 2nd Edition, New York: McGraw-Hill.

Clark, D.R.. 1985. HVACSIM+ Building Systems and Equipment Simulation Program Reference Manual, Pub. No. NBSIR 84-2996, National Bureau of Standards, U.S. Department of Commerce, January, 1985

Elmahdy, A.H. 1975. Analytical and Experimental Multi-Row Finned-Tube Heat Exchanger Performance During Cooling and Dehumidifying Processes, Ph.D. Thesis, Carleton University, Ottawa, Canada, December, 1975.

Elmahdy, A.H., and Mitalas, G.P. 1977. "A Simple Model for Cooling and Dehumidifying Coils for Use in Calculating Energy Requirements for Buildings," ASHRAE Transactions, Vol. 83, Part 2, pp. 103-117.

Chilled-Water-Based Detailed Geometry Air Cooling Coil

The input object Coil:Cooling:Water:DetailedGeometry provides a coil model that predicts changes in air and water flow variables across the coil based on the coil geometry. A greatly simplified schematic of enthalpy and temperature conditions in a counterflow cooling/dehumidifying coil is shown in the following schematic figure. In addition, the variables required to model a cooling/dehumidifying coils and their definitions are extensively listed in "Table 43. Coil Geometry and Flow Variables for Coils". The input required to model the coil includes a complete geometric description that, in most cases, should be derivable from specific manufacturer's data. The coil simulation model is essentially the one presented by Elmahdy and Mitalas (1977) and implemented in HVACSIM+ (Clark 1985), a modular program also designed for energy analysis of building systems. The model solves the equations for the dry and wet sections of the coil using log mean temperature and log mean enthalpy differences between the liquid and the air streams. Elmahdy and Mitalas state that crossflow counterflow coils with at four rows or more are approximated well by this model. This does not constitute a major limitation since cooling and dehumidifying coils typically have more than four rows.

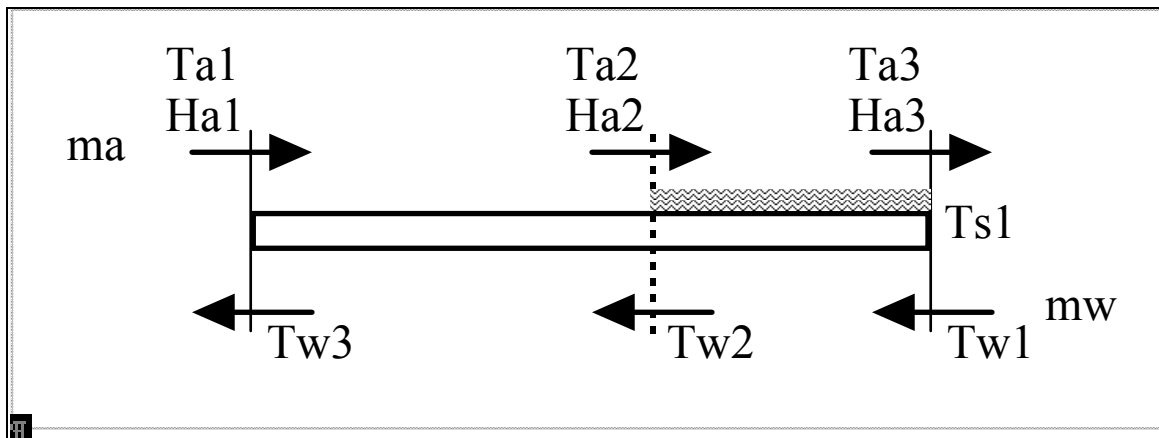


Figure 137. Simplified Schematic of Cooling/Dehumidifying Coil

Heat Transfer and Energy Balance

The cooling coil may be completely dry, completely wet with condensation, or it may have wet and dry sections. The actual condition of the coil surface depends on the humidity and temperature of the air passing over the coil and the coil surface temperature. The partly wet-partly dry case represents the most general scenario for the coil surface conditions. The all dry and all wet cases can be considered as limiting solutions of the wet or dry areas respectively going to zero. In the general case, equations are written for both the dry and wet regions of the coil. For each region the heat transfer rate from air to water may be defined by the rate of enthalpy change in the air and in the water. The rates must balance between each medium for energy to be conserved. Equations (294) through (297) express the energy balance between the water and the air for the case of dry and wet coils respectively. Equations (298) and (299) represent the heat transfer rate between water and air based on the actual performance of the coil. The UA parameter can be calculated from the parameters in the following table.

Table 43. Coil Geometry and Flow Variables for Coils

A	area	LMHD	log mean enthalpy difference
A	air, air side	LMTD	log mean temperature difference
aa, bb	coeff. in enthalpy approximation	\dot{m}	mass flow rate
C1, C2	coeff. in air side film coeff.	mf	metal and fouling
Cp	specific heat	μ	viscosity
D	diameter, effective diameter	o	outside (air side)
D _{hdr}	hydraulic diameter on air side	Pr	Prandtl number
D	dry region	\dot{Q}	heat transfer rate
δ	thickness	R	overall thermal resistance
Δ	spacing	Re	Reynolds number
F	heat transfer film coefficient	ρ	ratio of diameters
F _{ai}	variable in fin eff. calculation	s	surface, outside of metal
fin, fins	air side fin geometry	St	Stanton number
H	enthalpy	T	temperature

η	efficiency	tube	water tube
I0()	mod Bessel fn, 1st kind, ord 0	UAdry	dry heat xfer coeff. * dry area
I1()	mod Bessel fn, 1st kind, ord 1	UcAw	wet heat xfer coeff. * wet area
K0()	mod Bessel fn, 2nd kind, ord 0	ub, ue	variables in fin eff. calculation
K1()	mod Bessel fn, 2nd kind, ord 1	V	average velocity
I	inside (water side)	w	water, water side, or wet region
K1	variable in sol'n form of eq.	wa	humidity ratio
K	thermal conductivity	Z	variables in sol'n form of eq.
L	length	1, 2, 3	positions (see diagram)

Equations (294) through (299) represent two sets of three equations with 7 unknowns: \dot{Q}_d , $T_{a,1}$, $T_{a,2}$, $T_{w,2}$, $T_{w,3}$, \dot{m}_a , \dot{m}_w . However, normally at least four of these variables are specified, for example: inlet water temperature, outlet air temperature, water flow rate, air flow rate, so that the system of equations is effectively closed.

$$\dot{Q}_d = m_a C p_a (T_{a,1} - T_{a,2}) \quad (294)$$

$$\dot{Q}_d = m_w C p_w (T_{w,3} - T_{w,2}) \quad (295)$$

$$\dot{Q}_d = (U A_{dry}) (LMTD) \quad (296)$$

$$\dot{Q}_w = m_a (H_{a,2} - H_{a,3}) \quad (297)$$

$$\dot{Q}_w = m_w C p_w (T_{w,2} - T_{w,1}) \quad (298)$$

$$\dot{Q}_w = (U_c A_w) (LMHD) \quad (299)$$

In order to manipulate these equations, the log mean temperature and enthalpy differences are expanded as shown in Equations (300) and (301). Finally, a linear approximation of the enthalpy of saturated air over the range of surface temperature is made using Equation (302). Note that in Equation (301) H_w refers to the enthalpy of saturated air at the water temperature.

$$LMTD = \frac{(T_{a,1} - T_{w,3}) - (T_{a,2} - T_{w,2})}{\ln \frac{T_{a,1} - T_{w,3}}{T_{a,2} - T_{w,2}}} \quad (300)$$

$$LMHD = \frac{(H_{a,2} - H_{w,2}) - (H_{a,3} - H_{w,1})}{\ln \frac{H_{a,2} - H_{w,2}}{H_{a,3} - H_{w,1}}} \quad (301)$$

$$H_w = aa + bb T_w \quad (302)$$

Equation (303) is derived from the above equations and is used to solve for the coil conditions when all of the inlet conditions are given as input. Operating in this manner, the coil does not have a controlled outlet air temperature.

$$T_{w,2} = \frac{(1-Z)(H_{a,1} - aa - K1 Cp_a T_{a,1}) + Z T_{w,1} \left(bb - \frac{m_w Cp_w}{m_a} \right)}{bb - Z \frac{m_w Cp_w}{m_a} - (1-Z) K1 Cp_a} \quad (303)$$

An alternative solution method is to define the coil leaving air temperature as an input with a variable water flow rate. In this case Equations (304) and (305) are more convenient. Equations (306) through (308) define terms that are used to simplify Equations (303), (304) and (305).

$$T_{w,2} = \frac{(1-Z)(H_{a,3} - aa) + T_{w,1} \left(\frac{m_w Cp_w}{m_a} - bb Z \right)}{\frac{m_w Cp_w}{m_a} - bb} \quad (304)$$

$$T_{w,2} = \frac{(Z_d - 1) T_{a,1} Cp_a + T_{w,3} \left(Cp_a - Z_d \frac{m_w Cp_w}{m_a} \right)}{Z_d \left(Cp_a - \frac{m_w Cp_w}{m_a} \right)} \quad (305)$$

$$Z = \exp \left(U_c A_w \left(\frac{1}{m_a} - \frac{bb}{m_w Cp_w} \right) \right) \quad (306)$$

$$K1 = \frac{Z_d - 1}{Z_d - \frac{m_a Cp_a}{m_w Cp_w}} \quad (307)$$

$$Z_d = \exp \left(U_c A_{dry} \left(\frac{1}{m_a Cp_a} - \frac{1}{m_w Cp_w} \right) \right) \quad (308)$$

Underlying Correlations, Properties, and Assumptions

Overall heat transfer coefficients are calculated from the specified coil geometry and by using empirical correlations from fluid mechanics and heat transfer. For the water side, Equation (309) gives the film heat transfer coefficient in SI units:

$$f_i = 1.429(1 + 0.0146 T_w) V_w^{0.8} D_i^{-0.2} \quad (309)$$

This is valid for Reynolds numbers greater than 3100 based on water flow velocity and pipe inside diameter and is given in Elmahdy and Mitalas (1977) as recommended in the standard

issued by the Air-Conditioning and Refrigeration Institute (1972) for air-cooling coils. The definition of overall inside thermal resistance follows directly as shown in Equation(310).

$$R_i = \frac{1}{f_i A_i} \quad (310)$$

Equation (311) gives the film coefficient for the air side. Another form of the same equation is Equation (312), which is familiar from the data presented in Kays and London (1984). For coil sections that have a wet surface due to condensation, the air side film coefficient is modified according to Equation (313). The correction term, a function of air Reynolds number, is valid for Reynolds numbers between 400 and 1500. The coefficients in Equation (311) and (312) are calculated by Equations (314) and (315) that are functions of the coil geometry. Elmahdy (1977) explains the modifier for the wet surface and coefficients for the film coefficient. Equations (316) through (319) show definitions and values of common parameters and properties.

$$f_o = C_1 Re_a^{C_2} \frac{m_a}{A_{a_min_flow}} C_{p_a} Pr_a^{2/3} \quad (311)$$

$$C_1 Re_a^{C_2} = St_a Pr_a^{2/3} \quad (312)$$

$$f_{o,w} = f_o \left(1.425 - 5.1 \times 10^{-4} Re_a + 2.63 \times 10^{-7} Re_a^2 \right) \quad (313)$$

$$C_1 = 0.159 \left(\frac{\delta_{fin}}{D_{hdr}} \right)^{-0.065} \left(\frac{\delta_{fin}}{L_{fin}} \right)^{0.141} \quad (314)$$

$$C_2 = -0.323 \left(\frac{\Delta_{fins}}{L_{fin}} \right)^{0.049} \left(\frac{D_{fin}}{\Delta_{tube_rows}} \right)^{0.549} \left(\frac{\delta_{fin}}{\Delta_{fins}} \right)^{-0.028} \quad (315)$$

$$D_{hdr} = \frac{4A_{a_min_flow} \delta_{coil}}{A_{s_total}} \quad (316)$$

$$Re_a = \frac{4\delta_{coil} (1 + w_a) m_a}{A_{s_total} \mu_a} \quad (317)$$

$$Pr_a = 0.733 \quad (318)$$

$$\mu_a = 1.846 \times 10^{-5} \quad (319)$$

The film coefficients above act on the extended surface of the air side, that is the area of the fins and the tubes. Therefore, the fin efficiency must also be considered in calculating the overall thermal resistance on the outside. Gardner (1945) gives the derivation of Equation (320), used as a curve fit to find the fin efficiency as a function of film coefficient. This equation is based on circular fins of constant thickness. To model a coil with flat fins, an

effective diameter -- that of circular fins with the same fin area -- is used. Equations (321) through (324) define variables used in Equation (320). The overall efficiency of the surface is shown by Equation (325). Note that the efficiency is found by the same equations for the wet surface using the wet surface film coefficient.

$$\eta_{fin} = \frac{-2\rho}{fai(1+\rho)} \left[\frac{I_1(u_b)K_1(u_e) - K_1(u_b)I_1(u_e)}{I_0(u_b)K_1(u_e) + K_0(u_b)I_1(u_e)} \right] \quad (320)$$

$$fai = \frac{(D_{fin} - D_{tube})}{2} \sqrt{\frac{2f_o}{k_{fin} \delta_{fin}}} \quad (321)$$

$$\rho = \frac{D_{tube}}{D_{fin}} \quad (322)$$

$$u_e = \frac{fai}{1-\rho} \quad (323)$$

$$u_b = u_e \rho \quad (324)$$

$$\eta_o = 1 - (1 - \eta_{fin}) \frac{A_{fins}}{A_{s_total}} \quad (325)$$

The definition of overall outside thermal resistance is given in Equation (326) as a function of fin efficiency and film coefficient. For a wet coil surface the resistance must be defined differently because the heat transfer equations are based on enthalpy rather than temperature differences, as shown in Equation (327).

$$R_o = \frac{1}{f_o \eta_o A_{s_total}} \quad (326)$$

$$R_{o,w} = \frac{Cp_a / bb}{f_{o,w} \eta_{o,w} A_{s_total}} \quad (327)$$

Equation (328) gives the last two overall components of thermal resistance. They represent the metal tube wall and internal fouling. The fouling factor, due to deposits of dirt and corrosion of the tube inside surfaces, is assumed to be $5 \times 10^{-5} \text{ m}^2 \cdot \text{K/W}$. All components of thermal resistance are added in series to produce the overall heat transfer coefficients shown in Equations (329) and (330).

$$R_{mf} = \frac{\delta_{tube}}{k_{tube} A_i} + \frac{Fl}{A_i} \quad (328)$$

$$UA_{dry} = \frac{A_{dry}}{A_{s,total}} \left[\frac{1}{R_i + R_{mf} + R_o} \right] \quad (329)$$

$$U_c A_w = \frac{A_w}{A_{s,total}} \left[\frac{1/bb}{R_i + R_{mf} + R_{o,w}} \right] \quad (330)$$

Solution Method of Model

The complicated equations derived above were implemented in a successive substitution solution procedure to calculate the coil performance based on the input parameters. The MODSIM implementation of a cooling coil, the TYPE12 subroutine, was the motivation for this approach; the method used there has been retained with modifications for the uncontrolled coil model. Clark (1985) contains notes about the MODSIM routine.

In the general case, the cooling coil is only partially wet. For an uncontrolled coil, Equation (303) is used to find the water temperature at the boundary. Several simple equations in the loop adjust the boundary point until the dry surface temperature at the boundary is equal to the dewpoint of the inlet air. For the controlled coil, Equations (304) and (305) give two calculations of the boundary temperature, and the water flow rate and boundary position are adjusted until the two equations agree.

Special cases occur when the coil is all wet or all dry. The coil is solved as if it were all wet before the general case is attempted. If the wet surface temperatures at the coil inlet and outlet are both below the dewpoint, no further solution is required. However, to ensure a continuous solution as flow variables are changed, when the surface is all dry or when it is wet with only the dry surface equations yielding a surface temperature below the dewpoint at the water outlet, the general solution is used to calculate the unknowns. In the solution of the controlled coil the outlet air enthalpy, given some resulting dehumidification, must correspond to the enthalpy at the specified outlet air temperature.

Application of Cooling Coil Model to Heating Coils

The implementation of detailed heating coil models in IBLAST was another important aspect of the system/plant integration. The same kind of loops exist to provide hot water to the heating coils from the boilers as exist to supply the cooling coils with chilled water from the chillers. Some simplifications can be made, however, since the enthalpy change of the air flowing over a heating coil is entirely sensible. There is no condensation in a heating coil. In order to allow heating and cooling coils to be specified using the same geometric parameters, a heating coil simulation was developed from the cooling coil model described above by eliminating the wet surface analysis.

In addition, it was concluded that, since much simpler and less computationally expensive heating coil simulations are possible, an option was provided in IBLAST for a heating coil design using only the UA value of the coil, the product of heat transfer coefficient and coil area. This model was largely based on the TYPE10 subroutine implemented in MODSIM. The equations used to model the performance of the TYPE10 heating coil are as follows:

$$T_{a,out} = T_{a,in} + (T_{w,in} - T_{a,in}) \varepsilon \left(\frac{\min(C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w)}{C_{p,a} \dot{m}_a} \right) \quad (331)$$

$$T_{w,out} = T_{w,in} - (T_{a,out} - T_{a,in}) \left(\frac{C_{p,a} \dot{m}_a}{C_{p,w} \dot{m}_w} \right)$$

where the coil effectiveness is given by:

$$\varepsilon = 1 - \exp \left(\frac{\left\{ \exp \left[- \left(\frac{\min \{ C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w \}}{\max \{ C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w \}} \right) \{ NTU \}^{0.78} \right] - 1 \right\}}{\left(\frac{\min \{ C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w \}}{\max \{ C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w \}} \right) \{ NTU \}^{-2.2}} \right) \quad (332)$$

The parameter NTU is the number of transfer units and is defined as a function of the UA value of the coil as follows:

$$NTU = \frac{UA}{\min(C_{p,a} \dot{m}_a, C_{p,w} \dot{m}_w)} \quad (333)$$

Hot-Water-Based Air Heating Coil

Overview

The input object Coil:Heating:Water provides a model that uses an NTU–effectiveness model of a static heat exchanger. The model is an inlet – outlet model: given the inlet conditions and flow rates and the UA, the effectiveness is calculated using the formula for the effectiveness of a cross-flow heat exchanger with both fluid streams unmixed. The effectiveness then allows the calculation of the outlet conditions from the inlet conditions.

The inputs to the model are: (1) the current inlet temperatures and flow rates of the air and water fluid streams and (2) the UA of the coil. Note that the UA is fixed in this model and is not a function of the flow rates.

There are 2 alternative user inputs for the component: the user may input the design water volumetric flow rate and the UA directly; or the user may choose to input the more familiar design heating capacity plus design inlet & outlet temperatures and let the program calculate the design UA. These alternative user inputs are fully described in the EnergyPlus Input Output Reference document.

Model Description

The air and water capacitance flows are defined as:

$$\dot{C}_{air} = c_{p,air} \cdot \dot{m}_{air}$$

$$\dot{C}_{water} = c_{p,water} \cdot \dot{m}_{water}$$

The minimum and maximum capacity flows are then:

$$\dot{C}_{min} = \min(\dot{C}_{air}, \dot{C}_{water})$$

$$\dot{C}_{max} = \max(\dot{C}_{air}, \dot{C}_{water})$$

The capacitance flow ratio is defined as:

$$Z = \dot{C}_{min} / \dot{C}_{max}$$

The number of transfer units (*NTU*) is:

$$NTU = UA / \dot{C}_{\min}$$

The effectiveness is:

$$\varepsilon = 1 - \exp\left(\frac{e^{-NTU \cdot Z \cdot \eta} - 1}{Z \cdot \eta}\right)$$

Where $\eta = NTU^{-0.22}$.

The outlet conditions are then:

$$T_{air,out} = T_{air,in} + \varepsilon \cdot \dot{C}_{\min} \cdot (T_{water,in} - T_{air,in}) / \dot{C}_{air}$$

$$T_{water,out} = T_{water,in} - \dot{C}_{air} \cdot (T_{air,out} - T_{air,in}) / \dot{C}_{water}$$

The output of the coil in watts is:

$$\dot{Q}_{coil} = \dot{C}_{water} \cdot (T_{water,in} - T_{water,out})$$

The above formulas are given in the following reference, along with further references.

References

Wetter, M. 1999. Simulation Model: Finned Water-to-Air Coil Without Condensation. LBNL-42355.

This document can be downloaded from <http://simulationresearch.lbl.gov>.

Single-Speed Electric DX Air Cooling Coil

Overview

This model (object names Coil:Cooling:DX:SingleSpeed and Coil:Cooling:DX:TwoStageWithHumidityControlMode, with CoilPerformance:DX:Cooling) simulates the performance of an air-cooled or evaporative-cooled direct expansion (DX) air conditioner. The model uses performance information at rated conditions along with curve fits for variations in total capacity, energy input ratio and part-load fraction to determine the performance of the unit at part-load conditions (Henderson et al. 1992, ASHRAE 1993). Sensible/latent capacity splits are determined by the rated sensible heat ratio (SHR) and the apparatus dewpoint (ADP)/bypass factor (BF) approach. This approach is analogous to the NTU-effectiveness calculations used for sensible-only heat exchanger calculations, extended to a cooling and dehumidifying coil.

This model simulates the thermal performance of the DX cooling coil and the power consumption of the outdoor condensing unit (compressor, fan, crankcase heater and evap condenser water pump). The total amount of heat rejected by the condenser is also calculated and stored for use by other waste heat recovery models (e.g., Coil:Heating:Desuperheater). The performance of the indoor supply air fan varies widely from system to system depending on control strategy (e.g., constant fan vs. AUTO fan, constant air volume vs. variable air volume, etc.), fan type, fan motor efficiency and pressure losses through the air distribution system. Therefore, this DX system model does not account for the thermal effects or electric power consumption of the indoor supply air fan. EnergyPlus contains separate models for simulating the performance of various indoor fan configurations, and these models can be easily linked with the DX system model described here to simulate

the entire DX air conditioner being considered (e.g., see AirLoopHVAC:Unitary:Furnace:HeatCool, AirLoopHVAC:UnitaryHeatCool, ZoneHVAC:WindowAirConditioner or AirLoopHVAC:UnitaryHeatPump:AirToAir).

Model Description

The user must input the total cooling capacity, sensible heat ratio (SHR), coefficient of performance (COP) and the volumetric air flow rate across the cooling coil at rated conditions. The capacity, SHR and COP inputs should be “gross” values, excluding any thermal or energy impacts due to the indoor supply air fan. The rated conditions are considered to be air entering the cooling coil at 26.7°C drybulb/19.4°C wetbulb and air entering the outdoor condenser coil at 35°C drybulb/23.9°C wetbulb. The rated volumetric air flow should be between 0.00004027 m³/s and 0.00006041 m³/s per watt of rated total cooling capacity (300 – 450 cfm/ton).

The user must also input five performance curves that describe the change in total cooling capacity and efficiency at part-load conditions:

- 7) Total cooling capacity modifier curve (function of temperature)
- 8) Total cooling capacity modifier curve (function of flow fraction)
- 9) Energy input ratio (EIR) modifier curve (function of temperature)
- 10) Energy input ratio (EIR) modifier curve (function of flow fraction)
- 11) Part load fraction correlation (function of part load ratio)
 - The total cooling capacity modifier curve (function of temperature) is a biquadratic curve with two independent variables: wet-bulb temperature of the air entering the cooling coil, and dry-bulb temperature of the air entering the air-cooled condenser coil (wet-bulb temperature if modeling an evaporative-cooled condenser). The output of this curve is multiplied by the rated total cooling capacity to give the total cooling capacity at the specific entering air temperatures at which the DX coil unit is operating (i.e., at temperatures different from the rating point temperatures).

$$TotCapTempModFac = a + b(T_{wb,i}) + c(T_{wb,i})^2 + d(T_{c,i}) + e(T_{c,i})^2 + f(T_{wb,i})(T_{c,i}) \quad (334)$$

where

$T_{wb,i}$ = wet-bulb temperature of the air entering the cooling coil, °C

$T_{c,i}$ = dry-bulb temperature of the air entering an air-cooled condenser or wet-bulb temperature of the air entering an evaporative-cooled condenser, °C

- The total cooling capacity modifier curve (function of flow fraction) is a quadratic (or cubic) curve with the independent variable being the ratio of the actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated total cooling capacity and the total cooling capacity modifier curve (function of temperature) to give the total cooling capacity at the specific temperature and air flow conditions at which the DX unit is operating.

$$TotCapFlowModFac = a + b(ff) + c(ff)^2 \quad (335)$$

or

$$TotCapFlowModFac = a + b(ff) + c(ff)^2 + d(ff)^3$$

where

$$ff = \text{flow fraction} = \left(\frac{\text{Actual air mass flow rate}}{\text{Rated air mass flow rate}} \right)$$

Note: The actual volumetric air flow rate through the cooling coil for any simulation time step where the DX unit is operating must be between 0.00002684 m³/s and .00006713 m³/s per watt of rated total cooling capacity (200 - 500 cfm/ton). The simulation will issue a warning message if this air flow range is exceeded.

- The energy input ratio (EIR) modifier curve (function of temperature) is a biquadratic curve with two independent variables: wet-bulb temperature of the air entering the cooling coil, and dry-bulb temperature of the air entering the air-cooled condenser coil (wet-bulb temperature if modeling an evaporative-cooled condenser). The output of this curve is multiplied by the rated EIR (inverse of the rated COP) to give the EIR at the specific entering air temperatures at which the DX coil unit is operating (i.e., at temperatures different from the rating point temperatures).

$$EIRTempModFac = a + b(T_{wb,i}) + c(T_{wb,i})^2 + d(T_{c,i}) + e(T_{c,i})^2 + f(T_{wb,i})(T_{c,i}) \quad (336)$$

where

$T_{wb,i}$ = wet-bulb temperature of the air entering the cooling coil, °C

$T_{c,i}$ = dry-bulb temperature of the air entering an air-cooled condenser or wet-bulb temperature of the air entering an evaporative-cooled condenser, °C

- The energy input ratio (EIR) modifier curve (function of flow fraction) is a quadratic (or cubic) curve with the independent variable being the ratio of the actual air flow rate across the cooling coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated EIR (inverse of the rated COP) and the EIR modifier curve (function of temperature) to give the EIR at the specific temperature and air flow conditions at which the DX unit is operating.

$$EIRFlowModFac = a + b(ff) + c(ff)^2 \quad (337)$$

or

$$EIRFlowModFac = a + b(ff) + c(ff)^2 + d(ff)^3$$

where

$$ff = \text{flow fraction} = \left(\frac{\text{Actual air mass flow rate}}{\text{Rated air mass flow rate}} \right)$$

- The part load fraction correlation (function of part load ratio) is a quadratic or a cubic curve with the independent variable being part load ratio (sensible cooling load / steady-state sensible cooling capacity). The output of this curve is used in combination with the rated EIR and EIR modifier curves to give the “effective” EIR for a given simulation time step. The part load fraction (PLF) correlation accounts for efficiency losses due to compressor cycling.

$$PartLoadFrac = PLF = a + b(PLR) + c(PLR)^2 \quad (338)$$

or

$$PartLoadFrac = a + b(PLR) + c(PLR)^2 + d(PLR)^3 \quad (339)$$

where

$$PLR = \text{part - load ratio} = \left(\frac{\text{sensible cooling load}}{\text{steady - state sensible cooling capacity}} \right)$$

The part-load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the compressor(s) run continuously for the simulation time step). For PLR values between 0 and 1 ($0 \leq PLR < 1$), the following rules apply:

$$PLF \geq 0.7 \quad \text{and} \quad PLF \geq PLR$$

If $PLF < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the coil is defined as PLR/PLF . If $PLF < PLR$, then a warning message is issued and the runtime fraction of the coil is limited to 1.0.

A typical part load fraction correlation for a conventional, single-speed DX cooling coil (e.g., residential or small commercial unit) would be:

$$PLF = 0.85 + 0.15(PLR)$$

All five part-load curves are accessed through EnergyPlus' built-in performance curve equation manager (curve: quadratic, curve:cubic and curve:biquadratic). It is not imperative that the user utilize all coefficients shown in equations (334) through (338) if their performance equation has fewer terms (e.g., if the user's PartLoadFrac performance curve is linear instead of quadratic, simply enter the values for a and b, and set coefficient c equal to zero).

For any simulation time step, the total (gross) cooling capacity of the DX unit is calculated as follows:

$$\dot{Q}_{total} = \dot{Q}_{total, rated} (TotCapTempModFac)(TotCapFlowModFac) \quad (340)$$

In a similar fashion, the electrical power consumed by the DX unit (compressors plus outdoor condenser fans) for any simulation time step is calculated using the following equation:

$$Power = \left(\dot{Q}_{total} \right) (EIR)(RTF) \quad (341)$$

where

$$\dot{Q}_{total} = \text{Total cooling capacity, W -- ref. equation (340)}$$

$$EIR = \text{Energy input ratio} = \left(\frac{1}{COP_{rated}} \right) (EIRTempModFac)(EIRFlowModFac)$$

COP_{rated} = Coefficient of performance at rated conditions (user input)

$RTF = \left(\frac{PLR}{PartLoadFrac} \right)$ = runtime fraction of the cooling coil

The total amount of heat rejected by the condenser is then calculated and stored for use by other waste heat recovery models (e.g., Coil:Heating:Desuperheater).

$$\dot{Q}_{cond} = \dot{Q}_{total} (1 + EIR)$$

where

\dot{Q}_{cond} = total amount of heat rejected by the condenser (W)

The crankcase heater is assumed to operate when the cooling coil's compressor is OFF and the outdoor dry-bulb temperature is below the maximum outdoor temperature for crankcase heater operation. The average crankcase heater power for the simulation time step is calculated as follows:

$$P_{crankcase} = \dot{Q}_{cap,crankcase} (1 - RTF) \quad (342)$$

where

$P_{crankcase}$ = DX cooling coil crankcase heater power, W

$\dot{Q}_{cap,crankcase}$ = crankcase heater capacity, W

If this cooling coil is used as part of an air-to-air heat pump (Ref. UnitarySystem:HeatPump:AirToAir), the crankcase heater defined for this DX cooling coil is disregarded and the associated output variable is omitted. Instead, the crankcase heater defined for the DX heating coil (Coil:DX:HeatingEmpirical) is enabled during the time that the compressor is not running for either heating or cooling. In this instance, RTF in the above equations would be the runtime fraction of the heat pump's heating coil or cooling coil, whichever is greater.

In addition to calculating the total cooling capacity provided by the DX air conditioner, it is important to properly determine the break down of total cooling capacity into its sensible (temperature) and latent (dehumidification) components. The model computes the sensible/latent split using the rated SHR and the ADP/BF approach (Carrier et al. 1959). When the DX coil model is initially called during an EnergyPlus simulation, the rated total capacity and rated SHR are used to calculate the coil bypass factor (BF) at rated conditions. The rated total capacity and rated SHR are first used to determine the ratio of change in air humidity ratio to air dry-bulb temperature:

$$SlopeRated = \left(\frac{\omega_{in} - \omega_{out}}{T_{db,in} - T_{db,out}} \right)_{rated} \quad (343)$$

where

ω_{in} = humidity ratio of the air entering the cooling coil at rated conditions, kg/kg

ω_{out} = humidity ratio of the air leaving the cooling coil at rated conditions, kg/kg

$T_{db,in}$ = dry-bulb temperature of the air entering the cooling coil at rated conditions, °C

$T_{db,out}$ = dry-bulb temperature of the air leaving the cooling coil at rated conditions, °C

Along with the rated entering air conditions, the algorithm then searches along the saturation curve of the psychrometric chart until the slope of the line between the point on the saturation curve and the inlet air conditions matches *SlopeRated*. Once this point, the apparatus dewpoint, is found on the saturation curve the coil bypass factor at rated conditions is calculated as follows:

$$BF_{rated} = \frac{h_{out,rated} - h_{ADP}}{h_{in,rated} - h_{ADP}} \quad (344)$$

where

$h_{out,rated}$ = enthalpy of the air leaving the cooling coil at rated conditions, J/kg

$h_{in,rated}$ = enthalpy of the air entering the cooling coil at rated conditions, J/kg

h_{ADP} = enthalpy of saturated air at the coil apparatus dewpoint, J/kg

The coil bypass factor is analogous to the “ineffectiveness” (1-ε) of a heat exchanger, and can be described in terms of the number of transfer of unit (NTU).

$$BF = e^{-NTU} = e^{-\left(\frac{UA}{c_p}\right)/\dot{m}} = e^{-A_o/\dot{m}} \quad (345)$$

For a given coil geometry, the bypass factor is only a function of air mass flow rate. The model calculates the parameter A_o in equation (345) based on BF_{rated} and the rated air mass flow rate. With A_o known, the coil BF can be determined for non-rated air flow rates.

For each simulation time step when the DX air conditioner operates to meet a cooling load, the total cooling capacity at the actual operating conditions is calculated using equation (340) and the coil bypass factor is calculated based on equation (345). The coil bypass factor is used to calculate the operating sensible heat ratio (SHR) of the cooling coil using equations (346) and (347).

$$h_{ADP} = h_{in} - \frac{(\dot{Q}_{total} / \dot{m})}{1 - BF} \quad (346)$$

$$SHR = Minimum \left(\left(\frac{h_{Tin,wADP} - h_{ADP}}{h_{in} - h_{ADP}} \right), 1 \right) \quad (347)$$

where

h_{in} = enthalpy of the air entering the cooling coil, J/kg

h_{ADP} = enthalpy of air at the apparatus dewpoint condition, J/kg

$h_{Tin,wADP}$ = enthalpy of air at the entering coil dry-bulb temperature and humidity ratio at ADP, J/kg

\dot{m} = air mass flow rate, kg/s

With the SHR for the coil at the current operating conditions, the properties of the air leaving the cooling coil are calculated using the following equations:

$$h_{out} = h_{in} - \frac{\dot{Q}_{total}}{\dot{m}} \quad (348)$$

$$h_{T_{in},\omega_{out}} = h_{in} - (1 - SHR)(h_{in} - h_{out}) \quad (349)$$

$$\omega_{out} = PsyWFnTdbH(T_{in}, h_{T_{in},\omega_{out}}) \quad (350)$$

$$T_{db,out} = PsyTdbFnHW(h_{out}, \omega_{out}) \quad (351)$$

where

h_{out} = enthalpy of the air leaving the cooling coil, J/kg

$h_{T_{in},\omega_{out}}$ = enthalpy of air at the entering coil dry-bulb temperature and leaving air humidity ratio, J/kg

ω_{out} = leaving air humidity ratio, kg/kg

$T_{db,out}$ = leaving air dry-bulb temperature, °C

PsyWFnTdbH = EnergyPlus psychrometric function, returns humidity ratio given dry-bulb temperature and enthalpy

PsyTdbFnHW = EnergyPlus psychrometric function, returns dry-bulb temperature given enthalpy and humidity ratio

Dry Coil Conditions

If the model determines that the cooling coil is dry ($\omega_{in} < \omega_{ADP}$), then equations (340) and (341) are invalid since they are functions of entering wet-bulb temperature. Under dry-coil conditions, coil performance is a function of dry-bulb temperature rather than wet-bulb temperature. In this case, the model recalculates the performance of the DX cooling unit using the calculation procedure described above but with $\omega_{in} = \omega_{dry}$, where ω_{dry} is the inlet air humidity ratio at the coil dry-out point ($SHR = 1.0$).

Condenser Options: Air Cooled vs. Evaporatively Cooled

As described previously, this model can simulate the performance of air-cooled or evaporative-cooled DX air conditioners. The following paragraphs describe three modeling options.

If the user wants to model an air-cooled condenser, they should simply specify AirCooled in the field Condenser Type. In this case, the Total Cooling Capacity Modifier Curve (function of temperature) and the Energy Input Ratio Modifier Curve (function of temperature) (equations (334) and (336) above) will utilize the outdoor dry-bulb temperature.

If the user wishes to model an evaporatively-cooled condenser AND they have performance curves that are a function of the wet-bulb temperature of air entering the condenser coil, then the user should specify Condenser Type = EvaporativelyCooled and the evaporative condenser effectiveness value should be entered as 1.0. In this case, the Total Cooling Capacity Modifier Curve (function of temperature) and the Energy Input Ratio Modifier Curve (function of temperature) (equations (334) and (336) above) will utilize the outdoor wet-bulb temperature.

If the user wishes to model an air-cooled condenser that has evaporative media placed in front of it to cool the air entering the condenser coil, then the user should specify Condenser Type = EvaporativelyCooled. The user must also enter the appropriate evaporative effectiveness for the media. In this case, the Total Cooling Capacity Modifier Curve (function

of temperature) and the Energy Input Ratio Modifier Curve (function of temperature) will utilize the condenser inlet air temperature as calculated below:

$$T_{c,i} = (T_{wb,o}) + (1 - EvapCondEffectiveness)(T_{db,o} - T_{wb,o})$$

where

$T_{c,i}$ = the temperature of the air entering the condenser coil, °C

$T_{wb,o}$ = the wet-bulb temperature of the outdoor air, °C

$T_{db,o}$ = the dry-bulb temperature of the outdoor air, °C

In this case, the Total Cooling Capacity Modifier Curve (function of temperature) and the Energy Input Ratio Modifier Curve (function of temperature) input fields for this object should reference performance curves that are a function of outdoor dry-bulb temperature. Be aware that the evaporative media will significantly reduce the dry-bulb temperature of the air entering the condenser coil, so the Total Cooling Capacity and EIR Modifier Curves must be valid for the expected range of dry-bulb temperatures that will be entering the condenser coil.

If an evaporatively-cooled condenser is modeled, the power requirements for the water pump are calculated as follows:

$$P_{evapcondpump} = \dot{Q}_{cap,evapcondpump} (RTF)$$

where

$P_{evapcondpump}$ = DX cooling coil evap condenser pump electric power, W

$\dot{Q}_{cap,evapcondpump}$ = evaporative condenser pump rated power consumption, W

Water consumption for the evaporative-cooled condenser is calculated using the difference in air humidity level across the evaporative media and the condenser air mass flow rate:

$$V_{water} = \frac{\dot{m}_{air} (\omega_{evapcond,out} - \omega_{evapcond,in})}{\rho_{water}} (RTF) (TimeStepSys) (3600.)$$

where

V_{water} = DX cooling coil evap condenser water consumption, m³

\dot{m}_{air} = evaporative condenser air mass flow rate, kg/s

$\omega_{evapcond,in}$ = humidity ratio of outdoor air entering the evap condenser, kg/kg

$\omega_{evapcond,out}$ = humidity ratio of air leaving the evap condenser, kg/kg

ρ_{water} = density of water at the outdoor dry-bulb temperature, kg/m³

$TimeStepSys$ = HVAC system simulation time step, hr

Condenser Inlet Air Temperature

The air temperature entering the outdoor condenser is based on the weather data used during a simulation. This temperature is either taken directly from the weather data, or can be adjusted based on the height of the outdoor condenser. The input for Condenser Air Inlet Node Name can be used to control this optional feature. If this input is left blank, the air temperature entering the condenser is based solely on the weather data. If this input is not blank, then the node name specified must also be listed in an OutdoorAir:Node or OutdoorAir:NodeList object. When the node name is listed in an OutdoorAir:NodeList object, the air temperature entering the condenser is based solely on the weather data. When the node name is listed in an OutdoorAir:Node object, the height of the node determines the air temperature entering the outdoor condenser (see description of Local Outdoor Air Temperature Calculation in the Atmospheric Variation section of this document for further details).

Supply Air Fan Control: Cycling vs. Continuous

One of the inputs to the DX cooling coil model is the supply air fan operation mode: cycling fan, cycling compressor (CyclingFanAndCompressor) or continuous fan, cycling compressor (ContinuousFanWithCyclingCompressor). The first operation mode is frequently referred to as “AUTO fan”, where the compressor(s) and supply air fan operate in unison to meet the zone cooling load, and cycle off together when the cooling load has been met. The second operation mode is often referred to as “fan ON”, where the compressor(s) cycle on and off to meet the zone cooling load but the supply air fan operates continuously regardless of compressor operation.

The fan operation mode is input in one of two ways. If the DX cooling coil is used with AC equipment that does not specify a fan operation mode schedule (e.g. AirLoopHVAC:UnitaryCoolOnly), then the selection made in the DX cooling coil is used and remains fixed throughout the simulation. If the DX cooling coil is used with AC equipment that does specify a fan operation mode schedule (e.g. AirLoopHVAC:Unitary:Furnace:HeatCool), then the selection made in the DX cooling coil is ignored and the fan operation mode schedule value determines the fan operation mode for each time step throughout the simulation. A fan operation mode schedule value of 0 specifies AUTO fan mode operation while values other than 0 specify fan ON operation. The use of a schedule allows the fan operation mode to change based on time-of-day or with changes in season.

The EnergyPlus methodology for determining the impact that HVAC equipment has on an air stream is to calculate the mass flow rate and air properties (e.g., enthalpy, dry-bulb temperature, humidity ratio) exiting the equipment. These exiting conditions are passed along as inlet conditions to the next component model in the air stream. Eventually the flow rate and properties of the air being supplied to the conditioned zone are used in the zone energy balance to determine the resulting zone air temperature and humidity ratio. With this methodology, the determination of the air mass flow rate and air properties for the two different supply air fan operation modes is slightly different.

Cycling Fan Mode:

For the case of cycling fan/cycling compressor when humidity control is not specified, the conditions of the air leaving the cooling coil are the steady-state values calculated using equations (348), (350) and (351) above. However the air mass flow rate passed along to the next component (and eventually to the conditioned zone) is the average air mass flow rate for the system simulation time step (determined by the cooling system; see ZoneHVAC:WindowAirConditioner, AirLoopHVAC:Unitary:Furnace:HeatCool, AirLoopHVAC:UnitaryHeatCool, or AirLoopHVAC:UnitaryHeatPump:AirToAir).

For the case of cycling fan/cycling compressor when humidity control is specified, the conditions of the air leaving the cooling coil are calculated as the average conditions during the fan operating period. When the compressor operates in tandem with the fan (i.e., compressor part-load ratio [PLR] is equal to the fan PLR), the outlet conditions are identical to the calculations described above. When the compressor operates for a shorter duration

than the fan (i.e., the compressor PLR is less than the heating/fan PLR), the air properties leaving the cooling coil are calculated as the average conditions during the fan operating period. In this case the calculation of exiting air conditions is analogous to the calculations for continuous fan mode described below except that PLR in the equations represents the ratio of the compressor to the fan operating period. For cycling fan systems, the fan will only operate longer than the compressor, and therefore latent degradation may be modeled (user input), when humidity control is specified, a moisture load exists (i.e., the zone air humidistat senses a moisture load), and a heating load exists where the heating PLR is greater than the cooling PLR.

Continuous Fan Mode:

For the case of continuous fan/cycling compressor, the air mass flow rate is constant. However, the air properties leaving the cooling coil are calculated as the average conditions during the system simulation time step. The model assumes that the exiting air conditions are the steady-state values calculated using equations (348), (350) and (351) above when the compressor(s) operate. For the remainder of the system simulation time step, it is assumed that the air exiting the DX coil has the same properties as the air entering the coil. For this supply air fan operating strategy, the leaving air properties are calculated as follows:

$$h_{out,ContFanCycComp} = h_{out}(PLR) + h_{in}(1 - PLR) \quad (352)$$

$$\omega_{out,ContFanCycComp} = \omega_{out}(PLR) + \omega_{in}(1 - PLR) \quad (353)$$

$$T_{db,out,ContFanCycComp} = P_{sy}T_{dbFnHW}(h_{out,ContFanCycComp}, \omega_{out,ContFanCycComp}) \quad (354)$$

Latent Capacity Degradation

The latent (dehumidification) capacity of a direct-expansion (DX) cooling coil is strongly affected by part-load, or cyclic, operation. This is especially true in applications where the supply air fan operates continuously while the cooling coil cycles on and off to meet the cooling load. During constant fan operation, moisture condenses on the cooling coil when the compressor operates, but part or all of the moisture that is held by the coil evaporates back into the airstream when the cooling coil is deactivated (Figure 138). The net effect is that the amount of moisture removed from the air is degraded at part-load conditions as compared to steady-state conditions when the compressor operates continuously (Figure 139).

EnergyPlus is able to model latent capacity degradation based on algorithms developed by Henderson and Rengarajan (1996). The model is applicable to single-stage cooling units, like residential and small commercial air conditioners or heat pumps with less than 19 kW of nominal cooling capacity. The model inputs are described in the EnergyPlus Input/Output Reference for the object Coil:Cooling:DX:SingleSpeed. The model is enabled only if the four numerical inputs are defined (values greater than zero, see IO Reference).

The following discussion applies to both cycling fan and continuous fan operation when the fan operates for a longer period of time than the compressor and air continues to flow over the moisture laden cooling coil after compressor operation has terminated.

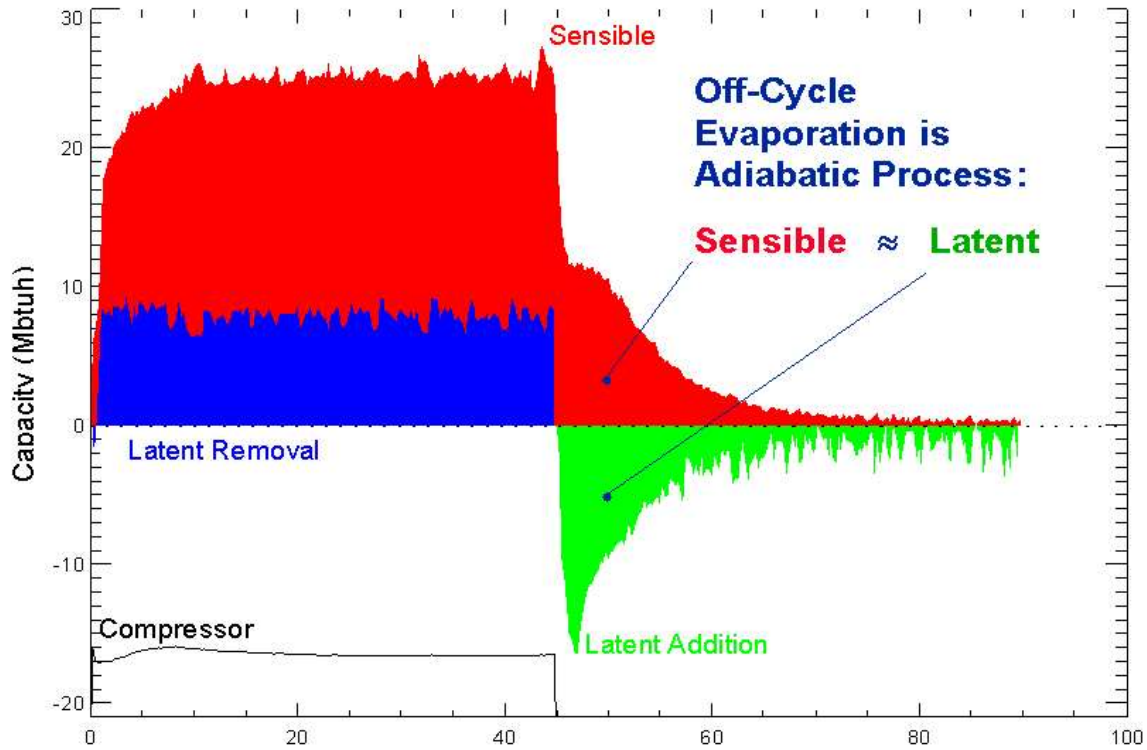


Figure 138. Transient Sensible and Latent Capacity of a Cooling Coil Over an Operating Cycle

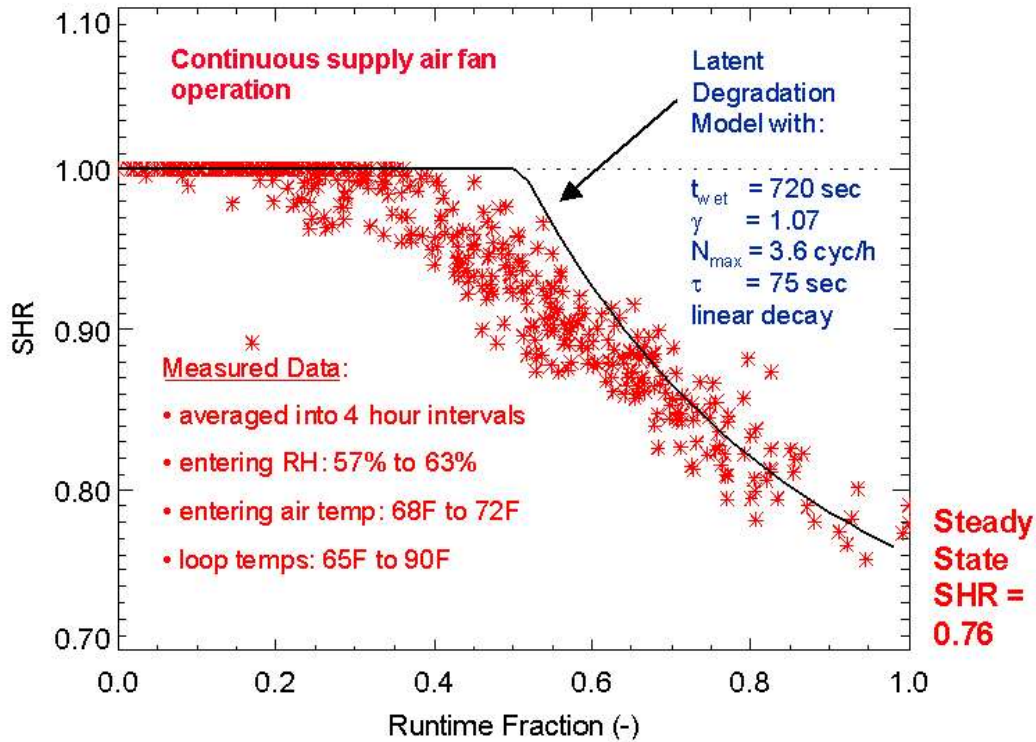


Figure 139. Field Data Showing the Net Impact of Part-Load Operation on Sensible Heat Ratio

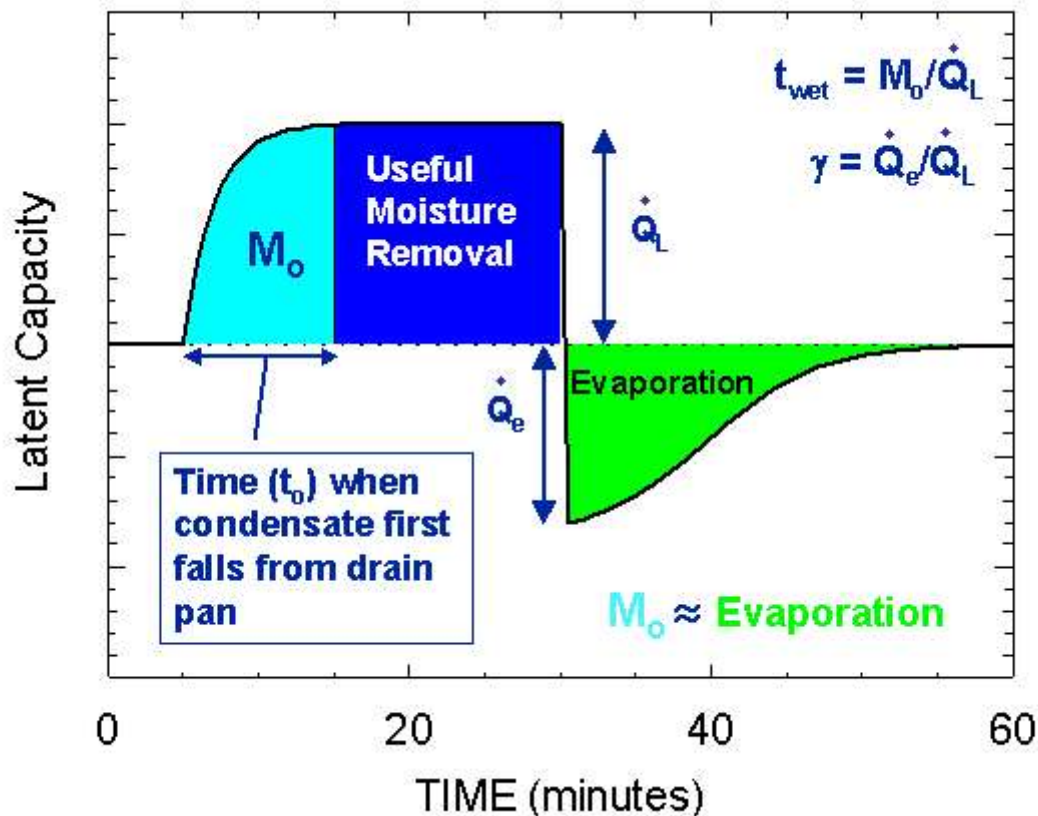


Figure 140. Concepts of Moisture Buildup and Evaporation

Figure 140 graphically depicts the latent degradation concepts and defines several key model parameters. After the cooling coil starts to operate, the coil temperature is eventually reduced below the dewpoint temperature of the entering air. Moisture from the air then builds on the surface of the coil until time t_0 has elapsed and the total moisture mass on the coil is M_0 . After this time (t_0), moisture begins to fall from the coil and all of the latent capacity provided by the coil is “useful” since this condensate is collected and removed from the unit. When the coil cycles off and the supply air fan continues to operate, the initial moisture mass buildup on the coil (M_0) evaporates back into the supply air stream. If the cooling coil cycles back on before all of the moisture has evaporated, then the time until the first condensate removal (t_0) is shorter for this cooling cycle since the coil is already partially wetted.

Figure 140 also shows several parameters that are used in the latent degradation model. The ratio of the coil’s moisture holding capacity (M_0) and the steady-state latent capacity (\dot{Q}_L) is defined as t_{wet} : the nominal time for moisture to fall from the coil (ignoring transient effects at startup and starting with a dry coil). The ratio of the initial moisture evaporation rate (\dot{Q}_e) and the steady-state latent capacity (\dot{Q}_L) is defined as γ . Both t_{wet} and γ at the rated air volume flow rate and temperature conditions are required model inputs. Two other model inputs are the Maximum ON/OFF Cycling Rate (cycles per hour, N_{max}) and the time constant (τ , in seconds) for the cooling coil’s latent capacity to reach steady state after startup. The

development of the latent degradation model is fully described by Henderson and Rengarajan (1996). The model implemented in EnergyPlus is for their “linear decay” evaporation model.

During the simulation, all of the steady-state calculations described previously in equations (334) through (347) are completed. The latent degradation model then modifies the steady-state sensible heat ratio for the coil as shown below. The value of t_{wet} at the current air volume flow rate and entering air conditions is first calculated based on the rated value of t_{wet} entered by the user:

$$t_{wet} = \text{Minimum} \left(t_{wet,rated} \left(\frac{\dot{Q}_{latent,rated}}{\dot{Q}_{latent}} \right), t_{wet,max} \right)$$

where

t_{wet} = nominal time for condensate removal to begin at the current airflow and entering air conditions, starting with a dry coil (sec)

$t_{wet,rated}$ = nominal time for condensate removal to begin at the coil’s rated airflow and entering air conditions, starting with a dry coil (sec)

$\dot{Q}_{latent,rated}$ = cooling coil latent capacity at the rated airflow and temperature conditions, W

\dot{Q}_{latent} = cooling coil latent capacity at the current airflow and temperature conditions, W

$t_{wet,max}$ = maximum allowed value for t_{wet} (9999.0 sec)

Likewise, the value of γ at the current air volume flow rate and entering air conditions is calculated based on the rated value of γ entered by the user:

$$\gamma = \gamma_{rated} \left(\frac{\dot{Q}_{latent,rated}}{\dot{Q}_{latent}} \right) \left(\frac{T_{db,i} - T_{wb,i}}{T_{db,rated} - T_{wb,rated}} \right)$$

where:

γ = ratio of the initial moisture evaporation rate from the cooling coil (when the compressor first turns off, in Watts) and the coil’s steady-state latent capacity (Watts) at the current air volume flow rate and entering air conditions

γ_{rated} = γ at rated air flow and entering air conditions

$T_{db,i}$ = dry-bulb temperature of the air entering the cooling coil, °C

$T_{wb,i}$ = wet-bulb temperature of the air entering the cooling coil, °C

$T_{db,rated}$ = dry-bulb temperature of air entering the cooling coil at rated conditions (26.7°C)

$T_{wb,rated}$ = wet-bulb temperature of air entering the cooling coil at rated conditions (19.4°C)

The cooling coil on and off times are then calculated based on the maximum number of cycles per hour and the calculated run-time fraction for the coil.

$$t_{on} = \frac{3600}{4N_{max}(1-X)}$$

$$t_{off} = \frac{3600}{4N_{max}X}$$

where

t_{on}	= duration of cooling coil on-cycle (sec)
N_{max}	= maximum on/off cycles per hour (cph)
X	= cooling coil runtime fraction (-)
t_{off}	= duration of cooling coil off-cycle (sec)

For cycling fan systems, the duration of the heating coil on and off cycles are also calculated. When the heating coil operates for a longer duration of time than the cooling coil, latent degradation can also occur. For this case, the off-cycle time (i.e., the amount of time the fan operates longer than the cooling coil) is recalculated and based on the difference between the heating coil on-cycle time and the cooling coil on-cycle time. t_{on} and t_{off} for the heating coil are calculated in the same manner as shown above except that the heating run-time fraction is used for the calculations. This model assumes that the cycling rate of the cooling and heating coils are the same. In addition, since the heating coil cycling rate may be different than the cooling coil (based on run-time fractions), the heating coil may also turn on again before the next cooling coil on-cycle. The following equations are used when the heating coil operates for a longer time period than does the cooling coil.

$$t_{on, heating} = \frac{3600}{4N_{max}(1 - X_{heating})}$$

$$t_{off, heating} = \frac{3600}{4N_{max}(X_{heating})}$$

$$t_{on, heating} = t_{on, heating} + MAX\left(0, MIN\left(t_{on, heating}, \left((t_{on} + t_{off}) - (t_{on, heating} + t_{off, heating})\right)\right)\right)$$

$$t_{off} = MIN\left(t_{off}, t_{on, heating} - t_{on}\right)$$

where

$t_{on, heating}$	= duration of heating coil on-cycle (sec)
$X_{heating}$	= heating coil runtime fraction (-)
$t_{off, heating}$	= duration of heating coil off-cycle (sec)

The equation for calculating the time t_o when moisture first begins to fall from the cooling coil is shown below, and is solved iteratively by EnergyPlus:

$$t_o^{j+1} = \gamma t_{off} - \left(\frac{\gamma^2}{4t_{wet}}\right) t_{off}^2 - \tau \left(e^{\frac{t_o^j}{\tau}} - 1\right), t_{off} \leq \left(\frac{2t_{wet}}{\gamma}\right)$$

where

t_o	= time where condensate removal begins (sec)
-------	--

τ = latent capacity time constant at start-up (sec)

j = iteration number

The part-load latent heat ratio of the cooling coil is then calculated with t_o , t_{on} and τ , which is in turn used to calculate the “effective” sensible heat ratio of the cooling including part-load latent degradation effects.

$$\frac{LHR}{LHR_{ss}} = \text{Maximum} \left(\left(\frac{t_{on} - t_o}{t_{on} + \tau \left(e^{\left(\frac{-t_{on}}{\tau} \right)} - 1 \right)} \right), 0.0 \right)$$

$$SHR_{eff} = 1 - (1 - SHR_{ss}) \left(\frac{LHR}{LHR_{ss}} \right)$$

where

LHR = part-load latent heat ratio

LHR_{ss} = latent heat ratio at steady-state conditions ($1 - SHR_{ss}$ with SHR_{ss} from eqn. (347))

SHR_{eff} = part-load sensible heat ratio ($SHR_{ss} \leq SHR_{eff} \leq 1.0$)

SHR_{ss} = steady-state sensible heat ratio (from eqn. (347))

With the “effective” SHR for the coil at the current operating conditions, including the impacts of latent degradation, equations (348) through (351) are then used to calculate the properties of the air leaving the cooling coil when it operates. Finally, equations (352) through (354) are used to calculate the average leaving air conditions (average when the coil is on and off) for the simulation time step.

Special Calculations for Coil: Cooling: DX: TwoStageWithHumidityControlMode with CoilPerformance: DX: Cooling

The multimode DX coil uses 1 to 4 set of performance specifications which all feed into the main DX coil model as described above. The multimode DX coil may have 1 or 2 capacity stages and may have 1 enhanced dehumidification mode in addition to its “normal” or base mode performance. Any mode may specify that a portion of the total airflow is bypassed.

Bypass Operation

When a mode has coil bypass, the non-bypassed air flow fraction is used to model the DX coil performance as described above. Then the bypassed air stream is mixed with the conditioned air stream and these conditions are placed on the coil outlet node.

Multi-Stage Operation

Multi-stage control is modeled with the DX coil. If 2-stage operation has been specified, stage 1 is assumed to operate first and continue to operate if stage 2 is required. If latent degradation is active, only one of the stages will be degraded. If stage 1 operates for less than the full time step, then latent degradation is applied. If stage 2 operates, then stage 1 is running for the full time step and latent degradation is applied only to the portion of the latent load which is attributed to stage 2 operation.

Enhanced Dehumidification Mode

If enhanced dehumidification mode is available, this is controlled by the parent object of the DX coil, such as DXSystem:AirLoop. A dehumidification mode flag is passed to the coil model which is used to select a different set of performance specifications from a CoilPerformance:DX:CoolingBypassFactorEmpirical input object. Then the simulation of the DX coil proceeds as described above including any bypass or multi-stage operation.

References

ASHRAE. 1993. HVAC2 Toolkit: A Toolkit for Secondary HVAC System Energy Calculation. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Carrier, W.H., R.E. Cherne, W.A. Grant, and W.H. Roberts. 1959. *Modern air conditioning, heating and ventilating*, 3d ed. New York: Pitman Publishing Corporation.

Henderson, H.I. Jr., K. Rengarajan, and D.B. Shirey III. 1992. The impact of comfort control on air conditioner energy use in humid climates. *ASHRAE Transactions* 98(2): 104-113.

Henderson, H.I. Jr. and K. Rengarajan. 1996. A Model to Predict the Latent Capacity of Air Conditioners and Heat Pumps at Part-Load Conditions with Constant Fan Operation. *ASHRAE Transactions* 102(2): 266-274.

Henderson, H. 1998. The impact of part load air conditioner operation on dehumidification performance: Validating a latent capacity degradation model. Presented at ASHRAE's IAQ & Energy '98 conference. New Orleans, LA. October.

Henderson, H. and D. Shirey. 2003. Understanding the Dehumidification Performance of Air-Conditioning Equipment at Part-Load Conditions. Paper presented at Joint CIBSE/ASHRAE Conference, *Building Sustainability, Value and Profit*. September. Edinburgh, Scotland.

Multi-Speed Electric DX Air Cooling Coil

Overview

This model (object name Coil:Cooling:DX:MultiSpeed) simulates the performance of an air-to-air direct expansion (DX) cooling system. The main difference compared to the other cooling coil models, such as Coil:Cooling:DX:SingleSpeed, is that this cooling coil allows modeling of two to four discrete compressor speeds. Each speed has a set of corresponding performance information at rated conditions along with curve fits for variations in total capacity, SHR, energy input ratio and part-load fraction to determine the performance of the unit at part-load conditions (DOE 1982). The full load supply airflow rate is dependent on the speed number and provided by its parent object (Ref. AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed). The part-load impact on coil energy use is automatically applied to the lowest speed. A choice is provided to determine whether part-load impacts on coil energy use are applied when the coil is operating at speeds greater than speed 1.

This model simulates the thermal performance of the indoor DX cooling coil, and the power consumption of the outdoor unit (multispeed compressor, fans, and crankcase heaters). The performance of the indoor supply air fan varies widely from system to system depending on control strategy (e.g., constant fan vs. AUTO fan), fan type, fan motor efficiency and pressure losses through the air distribution system. Therefore, this DX system model does not account for the thermal effects or electric power consumption of the indoor supply air fan. EnergyPlus contains separate models for simulating the performance of various indoor fan configurations, and these models can be easily linked with the DX system model described here to simulate the entire DX system being considered. For the time being, this coil model can only be called by the parent object AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed.

When the model determines performance at Speed 1 (the lowest speed) or cycling between OFF and Speed 1, its performance is almost the same as the performance for the Coil:Cooling:DX:SingleSpeed model. However, the outlet conditions are calculated slightly differently. Therefore, the Coil:Cooling:DX:SingleSpeed model may be considered as a subset of the model described here. When the multispeed coil model determines performance at higher speeds (above 1), the model linearly interpolates the performance at

two consecutive speeds (n-1 and n) as needed to meet the cooling load, with the fraction of time at each speed established by the speed ratio.

Model Inputs

The model inputs are also very similar to the inputs of the Coil:Cooling:DX:SingleSpeed object. The main difference is that this multispeed model requires a set of fields at each speed, such as rated capacity, rated SHR, rated COP, two capacity modifiers, two energy input ratio modifiers, part-load correction, and latent degradation inputs. The inputs also include waste heat fraction at the rated conditions and modifier as a function of temperature to calculate recoverable waste heat for heat recovery, which are not available in the similar Coil:Cooling:DX:SingleSpeed object

Speed 1 Operation

The calculation procedures in this model are identical to the Coil:Cooling:DX:SingleSpeed object (Ref: Coil:Cooling:DX:SingleSpeed) with one exception: outlet node condition calculation when the supply air fan operation mode is ContinuousFanWithCyclingCompressor. The following procedure provides the detailed description of the exception.

■ Total delivered cooling capacity

The total delivered cooling capacity for speed 1 operating at the cycling ratio needed to meet the requested cooling load is:

$$Q_{coil,cycling} = \dot{m}_{Speed1} * CycRatio * (h_{inlet} - h_{outlet,full})$$

where,

$Q_{coil,cycling}$ = delivered total cooling capacity for Speed 1 operating at a specific cycling ratio [W]

\dot{m}_{Speed1} = air mass flow rate through cooling coil at Speed 1 as set by the parent object [kg/s]

$h_{outlet,full}$ = specific enthalpy of the coil outlet air during full-load operation at Speed 1 (no cycling) [J/kg]

h_{inlet} = specific enthalpy of the coil inlet air [J/kg]

$CycRatio$ = cycling ratio at Speed 1, ratio of requested heating load to the full-load capacity of the coil at Speed 1 [dimensionless]

It is assumed that the coil provides no cooling capacity when the coil is OFF, even if the supply air fan continues to operate.

■ Outlet air specific enthalpy

The average specific enthalpy of the coil outlet air is then calculated based on the total delivered cooling capacity and the average air mass flow rate entering the coil:

$$h_{outlet,average} = h_{inlet} - \frac{Q_{coil,cycling}}{\dot{m}_{inlet}}$$

where

$h_{outlet,average}$ = averaged specific enthalpy at the coil outlet [J/kg]

h_{inlet} = specific enthalpy at the coil inlet [J/kg]

$Q_{coil,cycling}$ = total capacity at full load [W]

\dot{m}_{inlet} = mass flow rate at the inlet to the coil as established by the parent object (Ref. UnitarySystem:MultiSpeedHeatPump:AirToAir, Mass Flow Rate Calculation). This flow rate is the average value determined by the parent object, accounting for the specified flow rate when the cooling coil is ON and the specified flow rate when the cooling coil is OFF for the time step being simulated.

■ Sensible capacity

The minimum humidity ratio (HR_{min}) is based on humidity ratios between inlet and full load outlet as:

$$HR_{min} = \text{Minimum}(HR_{inlet}, HR_{full})$$

where

HR_{inlet} = Humidity ratio at the inlet [kg/kg]

HR_{full} = Full load humidity ratio at the outlet [kg/kg]

The coil sensible capacity may be calculated as:

$$Q_{coil,sens} = \dot{m}_{Speed1} * CycRatio * [h_{inlet}(T_{inlet}, HR_{min}) - h_{outlet,full}(T_{outlet,full}, HR_{min})]$$

where

$Q_{coil,sens}$ = delivered sensible cooling capacity [W]

$h_{outlet,full}$ = full load specific enthalpy at the coil outlet as a function of outlet dry-bulb temperature at the full load, and the minimum humidity ratio [J/kg]

h_{inlet} = specific enthalpy at the coil inlet [J/kg]

■ Latent capacity

The latent capacity is the difference between total and sensible capacities

$$Q_{coil,latent} = Q_{coil,cycling} - Q_{coil,sens}$$

where

$Q_{coil,latent}$ = delivered latent cooling capacity [W]

■ Average outlet air humidity ratio

The averaged outlet HR can be calculated as:

$$HR_{outlet,average} = HR_{inlet} - \frac{Q_{coil,latent}}{\lambda \dot{m}_{inlet}}$$

where

λ = heat of vaporization as a function of HR_{min} and $CycRatio * T_{outlet,full} + (1 - CycRatio) * T_{inlet}$ [J/kg]

■ Average outlet air temperature

Using the above averaged outlet humidity ratio and specific enthalpy, the averaged outlet temperature can be calculated using the psych function of PsyTdbFnHW.

The main reason using the above approach is that outlet conditions are calculated in the same way in low and high speed operation.

The crankcase heater defined for this DX cooling coil is enabled during the time that the compressor is not running for either heating or cooling. The crankcase heater power use from either heating or cooling is reported in the heating coil (Coil:Heating:DX:MultiSpeed).

Higher Speed Operation

This section describes how higher speed operation is simulated. When the required sensible load is less than the full load sensible capacity at Speed n (Speed Number > 1), the following calculations are performed:

- Bypass factor at Speed n-1 and Speed n

$$BypassFactor_n = f(RatedBypassFactor_n, RatedFlowRate_n, ActualFowRate_n)$$

$$BypassFactor_{n-1} = f(RatedBypassFactor_{n-1}, RatedFlowRate_{n-1}, ActualFowRate_{n-1})$$

where

BypassFactor_i = bypass factor at actual flow rate conditions at Speed i [dimensionless]

RatedBypassFactor_i = bypass factor at the rated conditions at Speed i [dimensionless]

RatedFowRate_i = air mass flow rate at the rated conditions at Speed i [kg/s]

ActualFowRate_i = actual air mass flow rate at Speed i [kg/s]

i = Speed n or Speed n-1

The bypass factor at Speed n is a function of the bypass factor at the rated conditions, rated airflow rate, and actual flow rate at Speed n. The calculation is performed by a function, called AdjustCBF in the DXCoil module.

- Total capacity at Speed n-1 and Speed n

$$TotCap_{n-1} = f(RatedCap_{n-1}, TotCapTempModFac_{n-1}, TotCapFlowModFac_{n-1}, BypassFactor_{n-1})$$

$$TotCap_n = f(RatedCap_n, TotCapTempModFac_n, TotCapFlowModFac_n, BypassFactor_n)$$

where

TotCap_i = total cooling capacity at given temperatures and flow rates at Speed i [W]

RatedCap_i = cooling capacity at the rated conditions at Speed i [W]

TotCapTempModFac_i = total cooling capacity modifier as a function of indoor web-bulb temperature and outdoor air dry-bulb temperature at Speed i

TotCapFlowModFac_i = total cooling capacity modifier as a function of the ratio of the actual flow rate across the cooling coil to the rated airflow rate at Speed i

i = Speed n or Speed n-1

The calculation is performed by a subroutine, called CalcTotCapSHR in the DXCoil module.

- EIR at Speed n-1 and Speed n

$$EIR_{n-1} = RatedEIR_{n-1} * EIRTempModFac_{n-1} * EIRFlowModFac_{n-1}$$

$$EIR_n = RateEIR_n * EIRTempModFac_n * EIRFlowModFac_n$$

where

EIR_i = Energy input ratio at given temperatures and flow rates at Speed i [W]

RatedEIR_i = Energy input ratio at the rated conditions at Speed i [W]

EIRTempModFac_i = Energy input ratio modifier as a function of indoor and outdoor air dry-bulb temperature at Speed i

EIRFlowModFac_i = Energy input ratio modifier as a function of ratio of the actual flow rate across the heating coil to the rated airflow rate at Speed i

$i = n$ or $n-1$

- Full load outlet conditions at Speed $n-1$ and Speed n

The calculation procedure of full load outlet conditions at Speed $n-1$ and Speed n is the same as the calculation procedure used in the Coil:Cooling:DX:SingleSpeed model (Ref. Coil:Cooling:DX:SingleSpeed). The difference is that the outlet conditions at Speed $n-1$ are calculated based on the total cooling capacity and mass flow rate at Speed $n-1$, while the outlet conditions at Speed n are calculated based on the total cooling capacity and mass flow rate at Speed n .

- Effective total cooling capacity

$$Q_{coil,SpeedRatio} = (SpeedRatio) \dot{m}_{Speed\ n} (h_{inlet} - h_{outlet,full_Speed\ n}) + (1 - SpeedRatio) \dot{m}_{Speed\ n-1} (h_{inlet} - h_{outlet,full_Speed\ n-1})$$

where

$Q_{coil,SpeedRatio}$ = delivered sensible cooling capacity at a given speed ratio between two consecutive speeds [W]

$\dot{m}_{Speed\ n}$ = air mass flow rate through cooling coil at Speed n as set by the parent object [kg/s]

$\dot{m}_{Speed\ n-1}$ = air mass flow rate through cooling coil at Speed 1 as set by the parent object [kg/s]

h_{inlet} = specific enthalpy at the coil inlet [J/kg]

$h_{outlet,full_Speed\ n}$ = full load specific enthalpy at the coil outlet at Speed n [J/kg]

$h_{outlet,full_Speed\ n-1}$ = full load specific enthalpy at the coil outlet at Speed $n-1$ [J/kg]

- Average outlet air specific enthalpy

$$h_{outlet,average} = h_{inlet} - \frac{Q_{coil,SpeedRatio}}{\dot{m}_{inlet}}$$

where

$h_{outlet,average}$ = averaged specific enthalpy at the coil outlet [J/kg]

h_{inlet} = specific enthalpy at the coil inlet [J/kg]

\dot{m}_{inlet} = mass flow rate at the inlet to the coil as established by the parent object (Ref. AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed, Mass Flow Rate Calculation). This flow rate is the average value determined by the parent object, accounting for the specified flow rate when the heating coil is at Speed n and the specified flow rate when the heating coil is at Speed $n-1$ for the time step being simulated.

- Effective sensible cooling capacity

The minimum humidity ratio (HR_{min}) is calculated as

$$HR_{\min} = \text{Minimum}[HR_{\text{inlet}}, (\text{SpeedRatio})HR_{\text{full},n} + (1.0 - \text{SpeedRatio})HR_{\text{full},n-1}]$$

The effective sensible cooling capacity is expressed as:

$$Q_{\text{coil},\text{sens}} = \dot{m}_{\text{Speed } n} (\text{SpeedRatio}) [h_{\text{inlet}}(T_{\text{inlet}}, HR_{\min}) - h_{\text{outlet},\text{full_Speed } n}(T_{\text{outlet},n}, HR_{\min})] \\ + \dot{m}_{\text{Speed } n-1} (1 - \text{SpeedRatio}) [h_{\text{inlet}}(T_{\text{inlet}}, HR_{\min}) - h_{\text{outlet},\text{full_Speed } n-1}(T_{\text{outlet},n-1}, HR_{\min})]$$

where

$Q_{\text{coil},\text{sens}}$ = effective sensible cooling capacity [W]

$h_{\text{outlet},\text{full_Speed } n}$ = full load specific enthalpy at the coil outlet at Speed n as a function of outlet dry-bulb temperature at the full load, and the minimum humidity ratio [J/kg]

$h_{\text{outlet},\text{full_Speed } n-1}$ = full load specific enthalpy at the coil outlet at Speed n-1 as a function of outlet dry-bulb temperature at the full load, and the minimum humidity ratio [J/kg]

h_{inlet} = specific enthalpy at the coil inlet [J/kg]

■ Average outlet air humidity ratio and temperature

The effective latent cooling capacity is the difference between the total and sensible capacity:

$$Q_{\text{coil},\text{latent}} = Q_{\text{coil},\text{SpeedRatio}} - Q_{\text{coil},\text{sens}}$$

$Q_{\text{coil},\text{latent}}$ = effective latent cooling capacity [W]

The average outlet air HR can be calculated as:

$$HR_{\text{outlet},\text{average}} = HR_{\text{inlet}} - \frac{Q_{\text{coil},\text{latent}}}{\lambda \dot{m}_{\text{inlet}}}$$

where

λ = heat of vaporization as a function of HR_{\min} and $\text{SpeedRatio} * T_{\text{outlet},n} + (1 - \text{SpeedRatio}) * T_{\text{outlet},n-1}$ [J/kg]

At the given averaged outlet humidity ratio and specific enthalpy, the averaged outlet temperature can be calculated using the psych function of `PsyTdbFnHW`.

■ Calculate combined energy input

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is No in the object (equivalent to a single compressor), the combined energy output is calculated as follows:

$$\text{CoolingPower} = (\text{TotCap}_n)(\text{EIR}_n)(\text{SpeedRatio}) + (\text{TotCap}_{n-1})(\text{EIR}_{n-1})(1.0 - \text{SpeedRatio})$$

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is Yes in the object (equivalent to multiple compressors), the combined energy output is calculated as follows:

$$\text{CoolingPower} = (\text{TotCap}_n)(\text{EIR}_n)(\text{RTF}) + (\text{TotCap}_{n-1})(\text{EIR}_{n-1})(1.0 - \text{RTF})$$

where

CoolingPower = Power used in Watt

RTF = Run time fraction at Speed n

■ Latent degradation

When the supply fan operation mode is `ContinuousFanWithCyclingCompressor` and the input of the `Apply Latent Degradation to Speeds Greater than 1` is Yes, the latent degradation is included at Speed n. The calculation procedure is the same as one in the `Coil:Cooling:DX:SingleSpeed` object. The difference is that the rated values and run time

fraction at Speed n are used. The adjusted SHR is used to calculate full load outlet conditions at Speed n.

It is expected to have less latent degradation at Speed n than Speed 1. Therefore, smaller values of the latent degradation inputs at Speed n than those at Speed 1 are recommended.

- Crankcase heater

There is no power need at higher speed operation.

Waste heat calculation

The waste heat generated by this coil object is calculated as:

$$Q_{WasteHeat} = (Fraction)(TempModifier)(CoolingPower)$$

where

Fraction = rated waste heat fraction of the energy input

TempModifier = waste heat modifier as a function of indoor and outdoor air dry-bulb temperature

Two-Speed Electric DX Air Cooling Coil

Overview

The input object Coil:Cooling:DX:TwoSpeed provides a model that is meant to represent either a 2 speed (fan and compressor) DX cooling unit (compressor, evaporator, condenser) or a variable speed DX cooling unit in which the variation of compressor speed and air flow rate is nearly linear as a function of cooling part load. In EnergyPlus Coil:Cooling:DX:TwoSpeed is used as the cooling coil in packaged variable volume systems (PVAV). The model is based upon the single speed DX unit Coil:Cooling:DX:SingleSpeed. Basically the model for Coil:Cooling:DX:TwoSpeed employs a separate single speed model at high speed (full load) and low speed (minimum load) and interpolates between these 2 states to obtain the needed cooling output. Below minimum load the unit cycles on/off, just like the single speed unit.

Inputs and Data

The input is similar to that for Coil:Cooling:DX:SingleSpeed except there needs to be two complete sets of performance data. The user inputs a rated total cooling capacity, rated SHR, rated COP, and rated air volumetric flow rate for the high speed and low speed states. Performance curves – cooling capacity as a function of entering air wet-bulb temperature and outside dry-bulb temperature (wet-bulb if there is an evaporatively cooled condenser), EIR as a function of the same two temperatures – must be defined and referenced for both high and low speed states. The performance characteristics of the evaporative condenser, if present, also need to be given at high and low speed states: effectiveness, air volumetric flow rate, and pump power consumption. The full list of inputs is given in the Input/Output Reference document.

The data for Coil:Cooling:DX:TwoSpeed is stored in the array structure *DXCoil* in the program module *DXCoils*

Calculation

Calculation is performed in routine *CalcMultiSpeedDXCoil* in module *DXCoils*. The inputs to the calculation are the coil index (denoting a specific coil with its particular performance parameters), the speed ratio, and the cycling ratio. The speed ratio is an artificial parameter between 0 and 1. If the speed ratio (*SR*) is greater than zero, the performance of the unit will be:

$$SR \cdot HighSpeedPerformance + (1 - SR) \cdot LowSpeedPerformance$$

Here *HighSpeedPerformance* means the electricity consumption, cooling output, and outlet conditions of the unit if the unit were operating at high speed (full load). Similarly *LowSpeedPerformance* means the electricity consumption, cooling output, and outlet conditions if the unit were operating at low speed (minimum non-cycling load). The calculations for each state are the same as for the single speed, cycling DX unit Coil:Cooling:DX:SingleSpeed.

If the speed ratio is zero and the cycling ratio is > 0, the unit will be in cycling mode. The unit will be on for *cycling ratio* fraction of the time step, off for the remainder of the time step. While on, the unit will perform according to the low speed performance parameters.

Simulation and Control

Coil:Cooling:DX:TwoSpeed is not used by itself; it is used as part of an encompassing component or system that provides control for the unit. In setting up a PVAV system, for instance, Coil:Cooling:DX:TwoSpeed is part of the component AirLoopHVAC:UnitaryCoolOnly, which controls Coil:Cooling:DX:TwoSpeed. When used in the AirLoopHVAC:UnitaryCoolOnly component, the simulation allows for either temperature or temperature and dehumidification control based on a dehumidification control selection in the AirLoopHVAC:UnitaryCoolOnly component. If None is selected the simulation runs the DX unit to satisfy a supply air temperature setpoint. It typically varies the speed ratio in an iterative solution process to establish the operating point that will give the desired unit outlet air temperature. If CoolReheat is selected, the simulation first runs the DX unit to satisfy a supply air temperature setpoint. However, in this case if the DX unit outlet air humidity ratio is above the humidity ratio setpoint, the simulation will increase the speed and/or cycling ratio to meet the desired unit outlet air humidity ratio. The humidity ratio setpoint is specified through the use of a humidistat (ref. ZoneControl:Humidistat) and a setpoint manager (ref. SetpointManager:SingleZone:Humidity:Maximum or SetpointManager:OutdoorAirPretreat). When used with the AirLoopHVAC:UnitaryCoolOnly component, the dehumidification control type Multimode is not allowed.

References

See in this document.

Electric Air Heating Coil

The electric air heating coil (object name: Coil:Heating:Electric) is a simple capacity model with a user-input efficiency. In many cases, this efficiency for the electric coil will be 100%. This coil only has air nodes to connect it in the system. The coil can be used in the air loop simulation or in the zone equipment as a reheat coil. Depending on where it is used determines if this coil is temperature or capacity controlled. If used in the air loop simulation it will be controlled to a specified temperature on the setpoint node by the SetpointManager. If it is used in zone equipment, it will be controlled from the zone thermostat by meeting the zone demand. See Input Output Reference for additional information.

For a coil that is controlled to meet the zone demand and will meet the capacity necessary for that zone unless it exceeds the capacity of the coil specified by the user.

```

! Control output to meet load QCoilReq
IF((AirMassFlow .GT. 0.0 .AND. HeatingCoil(CoilNum)%NominalCapacity > 0.0) .and. &
(GetCurrentScheduleValue(HeatingCoil(CoilNum)%SchedPtr) .gt. 0.0) .and. &
(QCoilReq .gt. 0.0) .and. (TempSetPoint == 0.0)) THEN
!check to see if the Required heating capacity is greater than the user! specified capacity.
IF(QCoilReq > HeatingCoil(CoilNum)%NominalCapacity) Then
  QCoilCap = HeatingCoil(CoilNum)%NominalCapacity
Else
  QCoilCap = QCoilReq
End IF

TempAirOut=TempAirIn + QCoilCap/CapacitanceAir
HeatingCoilLoad = QCoilCap

!The HeatingCoilLoad is the change in the enthalpy of the Heating
HeatingCoil(CoilNum)%ElecUseLoad = HeatingCoilLoad/Effic

```

For a temperature setpoint coil the delta temperature from the coil inlet temperature to the setpoint is determined and the capacity of the coil is calculated and is met if less than the user specified capacity.

```

! Control coil output to meet a setpoint temperature.
Else IF((AirMassFlow .GT. 0.0 .AND. HeatingCoil(CoilNum)%NominalCapacity > 0.0) .and. &
(GetCurrentScheduleValue(HeatingCoil(CoilNum)%SchedPtr) .gt. 0.0) .and. &
(QCoilReq == 0.0) .and. &
(ABS(TempSetPoint-TempAirIn) .gt. TempControlTol) ) THEN
QCoilCap = CapacitanceAir*(TempSetPoint - TempAirIn)
! check to see if setpoint above entering temperature. If not, set
! output to zero.
IF(QCoilCap .LE. 0.0) THEN
  QCoilCap = 0.0
  TempAirOut = TempAirIn
!check to see if the Required heating capacity is greater than the user
! specified capacity.
Else IF(QCoilCap > HeatingCoil(CoilNum)%NominalCapacity) Then
  QCoilCap = HeatingCoil(CoilNum)%NominalCapacity
  TempAirOut=TempAirIn + QCoilCap/CapacitanceAir
Else
  TempAirOut = TempSetPoint
End IF

HeatingCoilLoad = QCoilCap

!The HeatingCoilLoad is the change in the enthalpy of the Heating
HeatingCoil(CoilNum)%ElecUseLoad = HeatingCoilLoad/Effic

```

Gas Air Heating Coil

The gas-powered heating coil (object name: Coil:Heating:Gas) is a simple capacity model with user inputted gas burner efficiency. The default for the gas burner efficiency is 80%. This coil only has air nodes to connect it in the system. The coil can be used in the air loop simulation or in the zone equipment as a reheat coil. Depending on where it is used determines if this coil is temperature or capacity controlled. If used in the air loop simulation it will be controlled to a specified temperature scheduled from the SetpointManager. If it is used in zone equipment, it will be controlled from the zone thermostat by meeting the zone demand. The gas coil has additional features that can add a part load correction and a parasitic gas or electric load.

The parasitic electric load associated with the gas coil operation, such as an inducer fan, etc. This will be modified by the PLR (or coil runtime fraction if a part-load fraction correlation is provided in the next input field) to reflect the time of operation in a simulation time step.

The parasitic gas load associated with the gas coil's operation (Watts), such as a standing pilot light. The model assumes that this parasitic load is consumed only for the portion of the simulation time step where the gas heating coil is not operating.

Field: Part Load Fraction Correlation (function of part load ratio)

The part load correction defines the name of a quadratic or cubic performance curve (Ref: Performance Curves) that parameterizes the variation of gas consumption rate by the heating coil as a function of the part load ratio (PLR, sensible heating load/nominal capacity of the heating coil). For any simulation time step, the nominal gas consumption rate (heating load/burner efficiency) is divided by the part-load fraction (PLF) if a part-load curve has been defined. The part-load curve accounts for efficiency losses due to transient coil operation.

The part-load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the heating coil runs continuously for the simulation time step). For PLR values between 0 and 1 ($0 \leq \text{PLR} < 1$), the following rules apply:

PLF \geq 0.7 and PLF \geq PLR

If $\text{PLF} < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the heating coil is defined a PLR/PLF . If $\text{PLF} < \text{PLR}$, then a warning message is issues and the runtime fraction of the coil is limited to 1.0.

A typical part load fraction correlation for a conventional gas heating coil (e.g., residential furnace) would be:

$$\text{PLF} = 0.8 + 0.2 \cdot \text{PLR}$$

For a better understanding of how the coil meets the temperature setpoint in the air loop or the zone demand as zone equipment, see Coil:Electric:Heating for additional information. Also see Input Output Reference for additional input information.

Single-Speed Electric Heat Pump DX Air Heating Coil**Overview**

This model (object name Coil:Heating:DX:SingleSpeed) simulates the performance of an air-to-air direct expansion (DX) heating system. The model uses performance information at rated conditions along with curve fits for variations in total capacity, energy input ratio and part-load fraction to determine the performance of the unit at part-load conditions (DOE 1982). Adjustment factors are applied to total capacity and input power to account for frost formation on the outdoor coil.

This model simulates the thermal performance of the indoor DX heating coil, and the power consumption of the outdoor unit (compressors, fans, crankcase heaters and defrost heaters). The performance of the indoor supply air fan varies widely from system to system depending on control strategy (e.g., constant fan vs. AUTO fan, constant air volume vs. variable air volume, etc.), fan type, fan motor efficiency and pressure losses through the air distribution system. Therefore, this DX system model does not account for the thermal effects or electric power consumption of the indoor supply air fan. EnergyPlus contains separate models for simulating the performance of various indoor fan configurations, and these models can be easily linked with the DX system model described here to simulate the entire DX system being considered (e.g., see AirLoopHVAC:UnitaryHeatPump:AirToAir).

Model Inputs

The user must input the total heating capacity, coefficient of performance (COP) and the volumetric airflow rate across the heating coil at rated conditions. The capacity and COP inputs should be “gross” values, excluding any thermal or energy impacts due to the indoor supply air fan. The rating condition is considered to be outdoor air at 8.33C dry-bulb and 6.11C wet-bulb temperatures (i.e., air entering the outdoor coil), with air entering the indoor DX heating coil at 21.11C dry-bulb and 15.55C wet-bulb temperatures. The rated volumetric air flow across the DX heating coil should be between 0.00004027 m³/s and 0.00006041 m³/s per watt of rated total heating capacity (300 – 450 cfm/ton).

Depending on the defrost strategy that is selected, the user must also input up to six performance curves that describe the change in total heating capacity and efficiency at part-load conditions, and efficiency during reverse-cycle defrosting:

- 1) The total heating capacity modifier curve (function of temperature) can be a function of both the outdoor and indoor air dry-bulb temperature or only the outdoor air dry-bulb temperature. User has the choice of a bi-quadratic curve with two independent variables or a quadratic curve as well as a cubic curve with a single independent variable. The bi-quadratic curve is recommended if sufficient manufacturer data is available as it provides sensitivity to the indoor air dry-bulb temperature and a more realistic output. The output of this curve is multiplied by the rated total heating capacity to give the total heating capacity at specific temperature operating conditions (i.e., at an outdoor or indoor air temperature different from the rating point temperature).

$$TotCapTempModFac = a + b(T_{db,o}) + c(T_{db,o})^2 \quad (355)$$

or

$$TotCapTempModFac = a + b(T_{db,o}) + c(T_{db,o})^2 + d(T_{db,o})^3 \quad (356)$$

$$TotCapTempModFac = a + b(T_{db,i}) + c(T_{db,i})^2 + d(T_{db,o}) + e(T_{db,o})^2 + f(T_{db,i})(T_{db,o}) \quad (357)$$

where

$T_{db,i}$ = dry-bulb temperature of the air entering the indoor coil, °C

$T_{db,o}$ = dry-bulb temperature of the air entering the outdoor coil, °C

- 1) The total heating capacity modifier curve (function of flow fraction) is a quadratic or cubic curve with the independent variable being the ratio of the actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated total heating capacity and the total heating capacity modifier curve (function of temperature) to give the total heating capacity at the specific temperature and air flow conditions at which the coil is operating.

$$TotCapFlowModFac = a + b(ff) + c(ff)^2 \quad (358)$$

or

$$TotCapFlowModFac = a + b(ff) + c(ff)^2 + d(ff)^3 \quad (359)$$

where

$$ff = \text{flow fraction} = \left(\frac{\text{Actual air mass flow rate}}{\text{Rated air mass flow rate}} \right)$$

Note: The actual volumetric airflow rate through the heating coil for any simulation time step where the DX unit is operating should be between 0.00002684 m³/s and .00008056 m³/s per watt of rated total heating capacity (200 - 600 cfm/ton). The simulation will issue a warning message if this airflow range is exceeded.

- 2) The energy input ratio (EIR) modifier curve (function of temperature) can be a function of both the outdoor and indoor air dry-bulb temperature or only the outdoor air dry-bulb temperature. User has the choice of a bi-quadratic curve with two independent variables or a quadratic curve as well as a cubic curve with a single independent variable. The bi-quadratic curve is recommended if sufficient manufacturer data is available as it provides sensitivity to the indoor air dry-bulb temperature and a more realistic output. The output of this curve is multiplied by the rated EIR (inverse of the rated COP) to give the EIR at specific temperature operating conditions (i.e., at an outdoor or indoor air temperature different from the rating point temperature).

$$EIRTempModFac = a + b(T_{db,o}) + c(T_{db,o})^2 \quad (360)$$

or

$$EIRTempModFac = a + b(T_{db,o}) + c(T_{db,o})^2 + d(T_{db,o})^3 \quad (361)$$

or

$$EIRTempModFac = a + b(T_{db,i}) + c(T_{db,i})^2 + d(T_{db,o}) + e(T_{db,o})^2 + f(T_{db,i})(T_{db,o}) \quad (362)$$

- 3) The energy input ratio (EIR) modifier curve (function of flow fraction) is a quadratic or cubic curve with the independent variable being the ratio of the actual air flow rate across the heating coil to the rated air flow rate (i.e., fraction of full load flow). The output of this curve is multiplied by the rated EIR (inverse of the rated COP) and the EIR modifier curve (function of temperature) to give the EIR at the specific temperature and air flow conditions at which the coil is operating.

$$EIRFlowModFac = a + b(ff) + c(ff)^2 \quad (363)$$

or

$$EIRFlowModFac = a + b(ff) + c(ff)^2 + d(ff)^3 \quad (364)$$

- 4) The part-load fraction correlation (function of part-load ratio) is a quadratic or cubic curve with the independent variable being part-load ratio (sensible heating load / steady-state heating capacity). The output of this curve is used in combination with the rated EIR and EIR modifier curves to give the “effective” EIR for a given simulation time step. The part-load fraction (PLF) correlation accounts for efficiency losses due to compressor cycling.

$$PartLoadFrac = PLF = a + b(PLR) + c(PLR)^2 \quad (365)$$

or

$$PartLoadFrac = a + b(PLR) + c(PLR)^2 + d(PLR)^3 \quad (366)$$

where

$$PLR = \text{part-load ratio} = \left(\frac{\text{sensible heating load}}{\text{steady-state sensible heating capacity}} \right)$$

The part-load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the compressor(s) run continuously for the simulation time step). For PLR values between 0 and 1 ($0 \leq PLR < 1$), the following rules apply:

$$PLF \geq 0.7 \quad \text{and} \quad PLF \geq PLR$$

If $PLF < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the coil is defined a PLR/PLF . If $PLF < PLR$, then a warning message is issued and the runtime fraction of the coil is limited to 1.0.

A typical part load fraction correlation for a conventional, single-speed DX heating coil (e.g., residential heat pump) would be:

$$PLF = 0.85 + 0.15(PLR)$$

- 5) The defrost energy input ratio (EIR) modifier curve (function of temperature) is a bi-quadratic curve with two independent variables: outdoor air dry-bulb temperature and the heating coil entering air wet-bulb temperature. The output of this curve is multiplied by the heating coil capacity, the fractional defrost time period and the runtime fraction of the heating coil to give the defrost power at the specific temperatures at which the coil is operating. This curve is only required when a reverse-cycle defrost strategy is specified.

$$DefrostEIRTempModFac = a + b(T_{wb,i}) + c(T_{wb,i})^2 + d(T_{db,o}) + e(T_{db,o})^2 + f(T_{wb,i})(T_{db,o}) \quad (367)$$

where

$T_{wb,i}$ = wet-bulb temperature of the air entering the indoor heating coil, °C

$T_{db,o}$ = dry-bulb temperature of the air entering the outdoor coil, °C

All six curves are accessed through EnergyPlus' built-in performance curve equation manager (curve:quadratic, curve:cubic and curve:biquadratic). It is not imperative that the user utilize all coefficients shown in the preceding equations {(355) through (367)} in items (1) through (6) if their performance equation has fewer terms (e.g., if the user's PartLoadFrac performance curve is linear instead of quadratic or cubic, simply enter the appropriate values for the coefficients a and b, and set the remaining coefficients to zero).

The next input item for the HeatingEmpirical DX coil is the supply air fan operation mode. Either the supply air fan runs continuously while the DX coil cycles on/off, or the fan and coil cycle on/off together. The next two inputs define the minimum outdoor dry-bulb temperature that the heat pump compressor will operate and the maximum outdoor dry-bulb temperature for defrost operation. Crankcase heater capacity and crankcase heater cutout temperature are entered in the following two inputs. The final four inputs cover the type of defrost strategy (reverse-cycle or resistive), defrost control (timed or on-demand), the fractional defrost time period (timed defrost control only), and the resistive defrost heater capacity if a resistive defrost strategy is selected.

Model Description

The general flow of the model is as follows:

- 1) If the outdoor air dry-bulb temperature is below the specified minimum temperature for compressor operation or the DX heating coil is not scheduled to operate, simply pass through the heating coil inlet air conditions as the coil outlet conditions, set power and heating rates equal to zero, and set crankcase heater power equal to the crankcase heater capacity value specified by the input file.
- 2) If the outdoor air dry-bulb temperature is above the specified minimum temperature for compressor operation and the DX heating coil is scheduled to operate, then:
 - a. If the outdoor dry-bulb temperature is below the specified maximum outdoor dry-bulb temperature for defrost operation, calculate a heating capacity multiplier, input power multiplier and fractional defrost time period depending on the defrost strategy and defrost control type specified for the heating coil.
 - b. Using the rated heating capacity and COP, the part-load curves specified for the DX heating coil, the defrost multipliers calculated above (if applicable), and the part-load ratio that is being requested of the heating coil, determine the following: heating coil exiting air conditions (dry-bulb temperature, humidity ratio and enthalpy), total DX coil heating rate, electric power during heating (compressors and outdoor fans), electric power during defrost, and crankcase heater power.

The following paragraphs give a detailed description of the model calculations that are performed when the DX heating coil is operating (i.e., scenario # 2 above).

Frost Adjustment Factors

Frost formation on the outdoor coil, and the need to periodically defrost this coil, has a significant impact on heating capacity and energy use by the DX heating system. If the outdoor air dry-bulb temperature is below the specified maximum temperature for defrost operation, then the model calculates adjustment factors for heating capacity and input power due to frost formation, and the fractional defrost time period, depending on the defrost strategy and defrost control type specified for the heating coil. This method of accounting for the impacts of frosting/defrost was taken from the model used in DOE-2.1E (ESTSC 2001, Miller and Jaster 1985).

The model first estimates the outdoor coil temperature according to a linear empirical relationship with outdoor air dry-bulb temperature as the independent variable.

$$T_{coil,out} = 0.82T_{db,o} - 8.589 \quad (368)$$

The difference between the outdoor air humidity ratio (from the weather file) and the saturated air humidity ratio at the estimated outdoor coil temperature is then calculated, and this value is used as an indication of frost formation on the outdoor coil.

$$\Delta\omega_{coil,out} = MAX \left[1.0E-6, \omega_{outdoor} - \omega_{sat} \left(T_{coil,out}, OutBaroPress \right) \right] \quad (369)$$

Frost formation on the outdoor coil must be periodically removed. The fraction of compressor runtime when the coil is being defrosted is either entered by the user (for timed defrost) or is calculated by the model (for on-demand defrost) using an empirical equation and $\Delta\omega_{coil,out}$. Adjustment factors to total heating coil capacity and input power due to frost formation on the outdoor coil are also calculated by empirical models with $\Delta\omega_{coil,out}$ or fractional defrost time period as the independent variable. The defrost time period fraction and adjustment factors due to frost formation on the outdoor coil vary depending on the defrost control type as shown below.

Timed Defrost:

$$\text{Fractional Defrost Time} = \text{time period specified by user} = t_{frac,defrost} \quad (370)$$

$$\text{Heating Capacity Multiplier} = 0.909 - 107.33(\Delta\omega_{coil,out}) \quad (371)$$

$$\text{Input Power Multiplier} = 0.9 - 36.45(\Delta\omega_{coil,out}) \quad (372)$$

On-Demand Defrost:

$$\text{Fractional Defrost Time} = \frac{1}{1 + \left(\frac{0.01446}{\Delta\omega_{coil,out}} \right)} = t_{frac,defrost} \quad (373)$$

$$\text{Heating Capacity Multiplier} = 0.875(1 - t_{frac,defrost}) \quad (374)$$

$$\text{Input Power Multiplier} = 0.954(1 - t_{frac,defrost}) \quad (375)$$

If the outdoor air dry-bulb temperature is above the specified maximum temperature for defrost operation, the fractional defrost time period is set to zero and the heating capacity/input power multipliers are set to unity.

Defrost Operation

If the fractional defrost time period is greater than zero for the simulation time step, then the model calculates the electrical power used during defrost. The method for calculating defrost power varies based on the defrost strategy specified (i.e., reverse-cycle or resistive). In the case of reverse-cycle defrost, the additional heating load due to defrost (indoor cooling during defrost) is also calculated so that it may be added to the existing heating load when calculating input power for the compressor(s) and outdoor coil fan(s).

Reverse-Cycle:

$$Q_{defrost} = 0.01(t_{frac,defrost})(7.222 - T_{db,o}) \left(\frac{Q_{total, rated}}{1.01667} \right) \quad (376)$$

$$P_{defrost} = \text{DefrostEIRTempModFac} \left(\frac{Q_{total, rated}}{1.01667} \right) (t_{frac,defrost})(RTF) \quad (377)$$

Resistive:

$$Q_{defrost} = 0.0 \quad (378)$$

$$P_{defrost} = (Q_{cap,defrost})(t_{frac,defrost})(RTF) \quad (379)$$

where:

$Q_{defrost}$ = additional indoor heating load due to reverse-cycle defrost (W)

$Q_{total,rated}$ = total full-load heating capacity of the coil at rated conditions (W)

$P_{defrost}$ = average defrost power for the simulation time step (W)

$Q_{cap,defrost}$ = capacity of the resistive defrost heating element (W)

$DefrostEIRTempModFac$ = energy input ratio modifier curve applicable during defrost

$RTF = \left(\frac{PLR}{PartLoadFrac} \right)$ = runtime fraction of the heating coil

Heating Operation

For any simulation time step, the total heating capacity of the DX unit is calculated as follows:

$$Q_{total} = Q_{total,rated} (TotCapTempModFac) (TotCapFlowModFac) \quad (380)$$

If the outdoor air dry-bulb temperature is below the maximum temperature for defrost operation, then the total heating capacity is further adjusted due to outdoor coil frost formation based on the results of Equation (380) and Equation (371) or (374).

$$Q_{total} = Q_{total} (HeatingCapacityMultiplier) \quad (381)$$

In a similar fashion, the electrical power draw by the DX unit (compressors plus outdoor coil fans) for any simulation time step is calculated. For a reverse-cycle defrost strategy, the additional heating load ($Q_{defrost}$) generated during defrost operation is added to the heating load being requested by adjusting the part-load ratio. If a resistive defrost strategy is selected, $Q_{defrost} = 0$. The part-load fraction correlation for the heating coil (user input, Equation (365) or (366)) is used in the calculation of electrical power draw to account for efficiency losses due to compressor cycling.

$$PLR = MIN \left(1.0, PLR + \left(\frac{Q_{defrost}}{Q_{total}} \right) \right) \quad (382)$$

$$PartLoadFrac = a + b(PLR) + c(PLR)^2 + d(PLR)^3 \quad (366)$$

$$P_{heating} = \frac{(Q_{total})(EIR)(PLR)}{PartLoadFrac} \times InputPowerMultiplier \quad (383)$$

where

$P_{heating}$ = average compressor and outdoor fan power for the simulation time step(W)

Q_{total} = total heating capacity W, Eqn. (381)

$EIR = Energy\ input\ ratio = \left(\frac{1}{COP_{rated}} \right) (EIRTempModFac) (EIRFlowModFac)$

COP_{rated} = coefficient of performance at rated conditions (user input)

$InputPowerMultiplier$ = power adjustment due to frost if applicable -Eqn. (372) or (375)

The crankcase heater is assumed to operate when the heating coil's compressor is OFF, and the average crankcase heater power for the simulation time step is calculated as follows:

$$P_{crankcase} = Q_{cap,crankcase} (1 - RTF) \quad (384)$$

$$RTF = \left(\frac{PLR}{PartLoadFrac} \right) = \text{runtime fraction of the heating coil} \quad (385)$$

where

$P_{crankcase}$ = average crankcase heater power for the simulation time step (W)

$Q_{cap,crankcase}$ = crankcase heater capacity (W)

If this heating coil is used as part of an air-to-air heat pump (Ref. AirLoopHVAC:UnitaryHeatPump:AirToAir), the crankcase heater defined for this DX heating coil is enabled during the time that the compressor is not running for either heating or cooling (and the crankcase heater power defined in the DX cooling coil object is disregarded in this case). In this instance, RTF in the above equations would be the runtime fraction of the heat pump's heating coil or cooling coil, whichever is greater.

The properties of the air leaving the heating coil at full-load operation are calculated using the following equations:

$$h_{outlet} = h_{inlet} + \frac{Q_{total}}{m} \quad (386)$$

$$\omega_{outlet} = \omega_{inlet} \quad (387)$$

$$T_{db,outlet} = PsyTdbFnHW(h_{outlet}, \omega_{outlet}) \quad (388)$$

where

h_{outlet} = enthalpy of the air leaving the heating coil (J/kg)

ω_{outlet} = leaving air humidity ratio (kg/kg)

$T_{db,outlet}$ = leaving air dry-bulb temperature (°C)

$PsyTdbFnHW$ = EnergyPlus psychrometric function, returns dry-bulb temp given enthalpy and humidity ratio

Condenser Inlet Air Temperature

The air temperature entering the outdoor condenser is based on the weather data used during a simulation. This temperature is either taken directly from the weather data, or can be adjusted based on the height of the outdoor condenser. Since this DX heating coil can only be used in conjunction with a DX cooling coil, the input for Condenser Air Inlet Node Name in the DX cooling coil object can be used to control this optional feature. If this input is left blank, the air temperature entering the condenser is based solely on the weather data. If this input is not blank, then the node name specified must also be listed in an OutdoorAir:Node or OutdoorAir:NodeList object. When the node name is listed in an OutdoorAir:NodeList object,

the air temperature entering the condenser is based solely on the weather data. When the node name is listed in an OutdoorAir:Node object, the height of the node determines the air temperature entering the outdoor condenser (see description of Local Outdoor Air Temperature Calculation in the Atmospheric Variation section of this document for further details).

Supply Air Fan Control: Cycling vs. Continuous

One of the inputs to the DX coil model is the supply air fan operation mode: cycling fan, cycling compressor (CyclingFanAndCompressor) or continuous fan, cycling compressor (ContinuousFanWithCyclingCompressor). The first operation mode is frequently referred to as “AUTO fan”, where the compressor(s) and supply air fan operate in unison to meet the zone heating load, and cycle off together when the heating load has been met. The second operation mode is often referred to as “fan ON”, where the compressor(s) cycle on and off to meet the zone heating load but the supply air fan operates continuously regardless of compressor operation.

Since this DX heating coil can only be used in conjunction with a DX cooling coil (i.e. heat pumps), and these coils are used in AC equipment that specifies a fan operation mode schedule (e.g. AirLoopHVAC:UnitaryHeatPump:AirToAir), the selection made in the DX heating coil is currently ignored and the fan operation mode schedule value determines the fan operation mode for each time step throughout the simulation. A fan operation mode schedule value of 0 specifies AUTO fan mode operation while values other than 0 specify fan ON operation. The use of a schedule allows the fan operation mode to change based on time-of-day or with changes in season.

The EnergyPlus methodology for determining the impact that HVAC equipment has on an air stream is to calculate the mass flow rate and air properties (e.g., enthalpy, dry-bulb temperature, humidity ratio) exiting the equipment. These exiting conditions are passed along as inlet conditions to the next component model in the air stream. Eventually the flow rate and properties of the air being supplied to the conditioned zone are used in the zone energy balance to determine the resulting zone air temperature and humidity ratio.

With this methodology, the determination of the air mass flow rate and air properties for the two different supply air fan operation modes is slightly different. For the case of cycling fan/cycling compressor, the conditions of the air leaving the heating coil are the steady-state values calculated using equations (386), (387) and (388) above. However the air mass flow rate passed along to the next component (and eventually to the conditioned zone) is the average air mass flow rate for the system simulation time step (determined by the heating system; see AirLoopHVAC:UnitaryHeatPump:AirToAir). For this fan control type, the heating coil part-load fraction (Equation (365) or (366)) is also passed to Fan:OnOff (if used) to properly calculate the supply air fan power and associated fan heat.

For the case of continuous fan/cycling compressor, the air mass flow rate is constant. However, the air properties leaving the heating coil are calculated as the average conditions during the system simulation time step. The model assumes that the exiting air conditions are the steady-state values calculated using equations (386), (387) and (388) above when the compressor(s) operate. For the remainder of the system simulation time step, it is assumed that the air exiting the DX coil has the same properties as the air entering the coil. For this supply air fan operating strategy, the leaving air properties are calculated as follows:

$$h_{outlet,ContFanCycComp} = h_{outlet}(PLR) + h_{inlet}(1 - PLR) \quad (389)$$

$$\omega_{outlet,ContFanCycComp} = \omega_{outlet}(PLR) + \omega_{inlet}(1 - PLR) \quad (390)$$

$$T_{db,outlet,ContFanCycComp} = P_{sy}T_{dbFnHW}(h_{outlet,ContFanCycComp}, \omega_{outlet,ContFanCycComp}) \quad (391)$$

References

- DOE. 1982. *DOE-2 engineers manual*, version 2.1A. LBL-11353. Berkeley, CA: Lawrence Berkeley National Laboratory.
- ESTSC. 2001. DOE-2.1E Version 110 (source code). Oak Ridge, TN: Energy Science and Technology Software Center.
- Miller, R.L. and Jaster, H. 1985. Performance of Air-Source Heat Pumps. EM-4226. Palo Alto, CA: Electric Power Research Institute.

Multi-Speed Electric Heat Pump DX Air Heating Coil

Overview

This model (object name Coil:Heating:DX:MultiSpeed:) simulates the performance of an air-to-air direct expansion (DX) heating system. The main difference compared to the other heating coil model (Coil:Heating:DX:SingleSpeed) is that this heating coil allows modeling of two to four discrete compressor speeds. Each speed has a set of corresponding performance information at rated conditions along with curve fits for variations in total capacity, energy input ratio and part-load fraction to determine the performance of the unit at part-load conditions (DOE 1982). The full load supply airflow rate is dependent on the speed number and is set by its parent object (Ref: AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed). The part-load impact on coil energy use is automatically applied to the lowest speed. A choice is provided to determine whether part-load impacts on coil energy use are applied when the coil is operating at speeds greater than Speed 1. Adjustment factors applied to total capacity and input power to account for frost formation on the outdoor coil are calculated at each speed.

This model simulates the thermal performance of the indoor DX heating coil, and the power consumption of the outdoor unit (multispeed compressor, fans, crankcase heaters and defrost heaters). The performance of the indoor supply air fan varies widely from system to system depending on control strategy (e.g., constant fan vs. AUTO fan), fan type, fan motor efficiency and pressure losses through the air distribution system. Therefore, this DX system model does not account for the thermal effects or electric power consumption of the indoor supply air fan. EnergyPlus contains separate models for simulating the performance of various indoor fan configurations, and these models can be easily linked with the DX system model described here to simulate the entire DX system being considered. For the time being, this coil model can only be called by the parent object AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed.

When the model determines performance at Speed 1 (the lowest speed) or cycling between OFF and Speed 1, its performance is almost the same as the performance for the Coil:Heating:DX:SingleSpeed model. However, the outlet conditions are calculated slightly differently. Therefore, the Coil:Heating:DX:SingleSpeed model may be considered as a subset of the model described here. When the multispeed coil model determines performance at higher speeds (above 1), the model linearly interpolates the performance at two consecutive speeds (n-1 and n) as needed to meet the heating load, with the fraction of time at each speed established by the speed ratio.

Model Inputs

The model inputs are also very similar to the inputs of the Coil:Heating:DX:SingleSpeed object. The main difference is that this multispeed model requires a set of fields at each speed, such as rated capacity, rated COP, two capacity modifiers, two energy input ratio modifiers, and part-load correction. The inputs also include waste heat fraction and modifier as a function of temperature to calculate recoverable waste heat for heat recovery, which are not available in the similar Coil:Heating:DX:SingleSpeed object.

Speed 1 Operation

The calculation procedures in this model, including defrost and crankcase heater, are identical to the Coil:Heating:DX:SingleSpeed object (Ref: Coil:Heating:DX:SingleSpeed) with one exception: outlet node condition calculation when the supply air fan operation mode is ContinuousFanWithCyclingCompressor. The following procedure provides the detailed description of the exception.

- Total delivered heating capacity

The total delivered heating capacity for speed 1 operating at the cycling ratio needed to meet the requested heating load is:

$$Q_{coil,cycling} = \dot{m}_{Speed1} (CycRatio) (h_{inlet} - h_{outlet,full})$$

where,

$Q_{coil,cycling}$ = delivered sensible heating capacity for Speed 1 operating at a specific cycling ratio [W]

\dot{m}_{Speed1} = air mass flow rate through heating coil at Speed 1 as set by the parent object [kg/s]

$h_{outlet,full}$ = specific enthalpy of the coil outlet air during full-load operation at Speed 1 (no cycling) [J/kg]

h_{inlet} = specific enthalpy of the coil inlet air [J/kg]

$CycRatio$ = cycling ratio at Speed 1, ratio of requested heating load to the full-load capacity of the coil at Speed 1 [dimensionless]

It is assumed that the coil provides no heating capacity when the coil is OFF, even if the supply air fan continues to operate.

- Outlet air specific enthalpy

The average specific enthalpy of the coil outlet air is then calculated based on the delivered sensible heating capacity and the average air mass flow rate entering the coil:

$$h_{outlet,average} = h_{inlet} - \frac{Q_{coil,cycling}}{\dot{m}_{inlet}}$$

where,

$h_{outlet,average}$ = average specific enthalpy at the coil outlet [J/kg]

\dot{m}_{inlet} = mass flow rate at the inlet to the coil as established by the parent object (Ref. AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed, Mass Flow Rate Calculation). This flow rate is the average value determined by the parent object, accounting for the specified flow rate when the heating coil is ON and the specified flow rate when the heating coil is OFF for the time step being simulated.

- Outlet air temperature

The heating coil's outlet air humidity ratio equals the inlet air humidity ratio since the coil does not change the moisture content of the air. So the average outlet air temperature is calculated based on the inlet air humidity ratio and the average outlet air enthalpy using the psychrometric function PsyTdbFnHW.

The main reason for using the above approach is that outlet air conditions are calculated in the same way for all operating speeds.

The crankcase heater defined for this DX heating coil is enabled during the time that the compressor is not running for either heating or cooling. The crankcase heater power use from either heating or cooling is reported in the heating coil.

Higher Speed Operation

This section describes how higher speed operation is simulated. When the required sensible load is less than the full load sensible capacity at Speed n (Speed Number > 1), the following calculations are performed:

- Total delivered heating capacity at Speed n-1 and Speed n

$$TotCap_{n-1} = RatedCap_{n-1} (TotCapTempModFac_{n-1}) (TotCapFlowModFac_{n-1})$$

$$TotCap_n = RatedCap_n (TotCapTempModFac_n) (TotCapFlowModFac_n)$$

where,

$TotCap_i$ = total delivered heating capacity at given temperatures and flow rates at Speed i [W]

$RatedCap_i$ = heating capacity at the rated conditions at Speed i [W]

$TotCapTempModFac_i$ = total heating capacity modifier as a function of indoor and outdoor air dry-bulb temperature at Speed i

$TotCapFlowModFac_i$ = total heating capacity modifier as a function of the ratio of the actual flow rate across the heating coil to the rated airflow rate at Speed i

i = Speed n or Speed n-1

- EIR at Speed n-1 and Speed n

$$EIR_{n-1} = RatedEIR_{n-1} (EIRTempModFac_{n-1}) (EIRFlowModFac_{n-1})$$

$$EIR_n = RateEIR_n (EIRTempModFac_n) (EIRFlowModFac_n)$$

where,

EIR_i = energy input ratio at given temperatures and flow rates at Speed i [W]

$RatedEIR_i$ = energy input ratio at the rated conditions at Speed i [W]

$EIRTempModFac_i$ = energy input ratio modifier as a function of indoor and outdoor air dry-bulb temperature at Speed i

$EIRFlowModFac_i$ = energy input ratio modifier as a function of the ratio of the actual flow rate across the heating coil to the rated airflow rate at Speed i

i = Speed n or Speed n-1

- Full load outlet air specific enthalpy at Speed n-1 and Speed n

$$h_{outlet,full_Speed\ n} = h_{inlet} - \frac{(TotCap_n * HeatingCapacityMultiplier)}{\dot{m}_{inlet}}$$

$$h_{outlet,full_Speed\ n-1} = h_{inlet} - \frac{(TotCap_{n-1} * HeatingCapacityMultiplier)}{\dot{m}_{inlet}}$$

where,

HeatingCapacityMultiplier = frost adjustment factor for heating capacity (See Ref. Coil:Heating:DX:SingleSpeed)

$h_{outlet,full_Speed\ n}$ = specific enthalpy of the coil outlet air during full-load operation at Speed n (no cycling) [J/kg]

$h_{outlet,full_Speed\ n-1}$ = specific enthalpy of the coil outlet air during full-load operation at Speed n-1 (no cycling) [J/kg]

- Effective total heating capacity

$$Q_{coil,SpeedRatio} = (SpeedRatio) \dot{m}_{Speed\ n} (h_{inlet} - h_{outlet,full_Speed\ n}) + (1 - SpeedRatio) \dot{m}_{Speed\ n-1} (h_{inlet} - h_{outlet,full_Speed\ n-1})$$

where,

$Q_{coil,SpeedRatio}$ = delivered sensible heating capacity at a given speed ratio between two consecutive speeds [W]

$\dot{m}_{Speed\ n}$ = air mass flow rate through heating coil at Speed n as set by the parent object [kg/s]

$\dot{m}_{Speed\ n-1}$ = air mass flow rate through heating coil at Speed 1 as set by the parent object [kg/s]

- Average outlet air enthalpy

$$h_{outlet,average} = h_{inlet} - \frac{Q_{coil,SpeedRatio}}{\dot{m}_{inlet}}$$

where,

$h_{outlet,average}$ = average specific enthalpy at the coil outlet [J/kg]

h_{inlet} = specific enthalpy of the coil inlet air [J/kg]

\dot{m}_{inlet} = Mass flow rate at the inlet to the coil as established by the parent object (Ref. AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed, Mass Flow Rate Calculation). This flow rate is the average value determined by the parent object, accounting for the specified flow rate when the heating coil is at Speed n and the specified flow rate when the heating coil is at Speed n-1 for the time step being simulated.

- Average outlet air temperature

The heating coil's outlet air humidity ratio equals the inlet air humidity ratio since the coil does not change the moisture content of the air. So the average outlet air temperature is calculated based on the inlet air humidity ratio and the average outlet air enthalpy using the psychrometric function *PsyTdbFnHW*.

- Full load energy inputs at Speed n-1 and Speed n

$$HeatingPower_n = TotCap_n (EIR_n) (HeatingCapacityMultiplier) (InputPowerMultiplier)$$

$$HeatingPower_{n-1} = TotCap_{n-1} (EIR_{n-1}) (HeatingCapacityMultiplier) (InputPowerMultiplier)$$

where,

InputPowerMultiplier = Frost adjustment factor for heating power calculation (Ref. Coil:Heating:DX:SingleSpeed)

- Calculate combined energy input

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is No (equivalent to a single compressor), the combined energy output is calculated as follows:

$$HeatingPower = HeatingPower_n (SpeedRatio) + HeatingPower_{n-1} (1.0 - SpeedRatio)$$

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is Yes (equivalent to multiple compressors), the combined energy output is calculated as follows:

$$HeatingPower = HeatingPower_n (RTF) + HeatingPower_{n-1} (1.0 - RTF)$$

where,

HeatingPower = Power used in Watt

RTF = Run time fraction (SpeedRatio/Part-load Fraction) at Speed n

- Calculate defrost power

When the defrost strategy is resistive, the power calculation is the same as Speed 1 operation (Ref. Coil:Heating:DX:SingleSpeed). When the defrost strategy is reverse-cycle, the following calculations are performed:

$$Q_{defrost,n} = 0.01(t_{frac,defrost})(7.222 - T_{db,o}) \left(\frac{Q_{total,rated,n}}{1.01667} \right)$$

$$P_{defrost,n-1} = DefrostEIRTempModFac \left(\frac{Q_{total,rated,n-1}}{1.01667} \right) (t_{frac,defrost})$$

$$P_{defrost,n} = DefrostEIRTempModFac \left(\frac{Q_{total,rated,n}}{1.01667} \right) (t_{frac,defrost})$$

where:

$Q_{defrost,n}$ = additional indoor heating load due to reverse-cycle defrost at Speed n (W)

$Q_{total,rated,n}$ = total full-load heating capacity of the coil at rated conditions at Speed n (W)

$P_{defrost,n-1}$ = full load defrost power for the simulation time step at Speed n-1 (W)

$P_{defrost,n}$ = full load defrost power for the simulation time step at Speed n (W)

$Q_{total,rated,n-1}$ = capacity of the resistive defrost heating element at Speed n-1 (W)

$Q_{total,rated,n}$ = capacity of the resistive defrost heating element at Speed n (W)

DefrostEIRTempModFac = defrost energy input ratio (EIR) modifier curve (Ref. Coil:Heating:DX:SingleSpeed).

$T_{frac,defrost}$ = fractional defrost time (Ref. Coil:Heating:DX:SingleSpeed)

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is No (equivalent to a single compressor), the average defrost power is calculated as follows:

$$P_{defrost} = P_{defrost,n}(SpeedRatio) + P_{defrost,n-1}(1.0 - SpeedRatio)$$

When the input for the field 'Apply Part Load Fraction to Speeds Greater than 1' is Yes (equivalent to multiple compressors), the combined defrost energy is calculated as follows:

$$P_{defrost} = P_{defrost,n}(RTF) + P_{defrost,n-1}(1.0 - RTF)$$

where,

$P_{defrost}$ = average defrost power used in Watt

RTF = Run time fraction (SpeedRatio/Part-load Fraction) at Speed n

■ Crankcase heater

There is no power need at higher speed operation.

Waste heat calculation

The waste heat generated by this coil object is calculated as:

$$Q_{WasteHeat} = (Fraction)(TempModifier)(HeatingPowe)$$

where

Fraction = rated waste heat fraction of the energy input

TempModifier = waste heat modifier as a function of indoor and outdoor air dry-bulb temperature

Desuperheater-Recovery-Based Air Heating Coil

Overview

The input object Coil:Heating:Desuperheater provides a model that simulates the thermal performance of a refrigerant-to-air heating coil and the parasitic electric consumption of its control valves or other auxiliary devices. The model assumes that the heating energy provided by this coil is reclaimed from the superheated refrigerant gas leaving a compressor and does not impact the performance of the compressor. The objects from which this coil can obtain its heating energy are:

Coil:Cooling:DX:SingleSpeed

Coil:Cooling:DX:TwoSpeed

Coil:Cooling:DX:TwoStageWithHumidityControlMode

Refrigeration:CompressorRack

Refrigeration:Condenser* (multiple objects)

The heat reclaim recovery efficiency (specified by the user) defines the amount of heat available for use by this coil. Approximately 25-30% of the energy rejected by typical refrigeration system condensers is to reduce the superheated refrigerant vapor temperature to the condensing temperature. Recovery efficiencies higher than 30% may cause the refrigerant gas to condense which in turn impacts the performance of the refrigeration compressor rack. For this reason, the maximum heat reclaim recovery efficiency for this coil is 30%. For refrigeration condensers, the amount of waste energy available in the superheat region is explicitly calculated, so for these condensers the maximum heat recovery efficiency is 90%.

NOTE: When the heating source is a refrigeration compressor rack, the heat rejection location in the Refrigeration:CompressorRack object must be "Outdoors". If the compressor rack heat rejection location is

“Zone”, the total amount of heat rejection available for reclaim (e.g., by this desuperheater heating coil) is set to zero by the compressor rack object and the simulation proceeds.

The sources for the waste heat can also be used to supply other coils. To avoid double-counting this waste heat, the amount available is corrected to reflect these other uses. For the three DX Cooling Coil sources, this correction is made within the same time step loop. For the two refrigeration system sources, the correction is made using the value from the previous time step.

The desuperheater heating coil can be used in air loop simulations for various air heating applications. For example, it can be used as an air reheat coil for high humidity control in the compound objects `AirLoopHVAC:Unitary:Furnace:HeatCool` and `AirLoopHVAC:UnitaryHeatCool` (see High Humidity Control With HeatCool Configuration). For this application, the desuperheater coil is controlled based on the calculated heating load to maintain the zone temperature and humidity setpoints (load-based control). The source of reclaimed heat could be the direct expansion (DX) cooling coil itself (`Coil:Cooling:DX:SingleSpeed`) or a refrigerated case compressor rack (`Refrigeration:CompressorRack`).

The desuperheater heating coil can also be used with cooling/heating systems that maintain specific air loop (deck) temperatures. For example, Figure 141 shows a schematic diagram of the desuperheater heating coil used in conjunction with the `AirLoopHVAC:UnitaryCoolOnly` object. The desuperheater heating coil must be placed downstream of the DX cooling coil when reclaiming heat from that cooling coil’s compressor(s). Desuperheater heating coil placement is unrestricted when reclaiming heat from a refrigeration compressor rack or refrigeration condenser. The configuration in Figure 141 shows the heating coil being controlled via a temperature-based strategy. In this example, the DX cooling coil could be used to maintain its discharge air temperature at 11°C for zone dehumidification. The desuperheater heating coil could then raise the air dry-bulb temperature to 16°C providing a dry supply air stream at a temperature which does not require much additional heating by terminal units to meet the zone temperature setpoint.

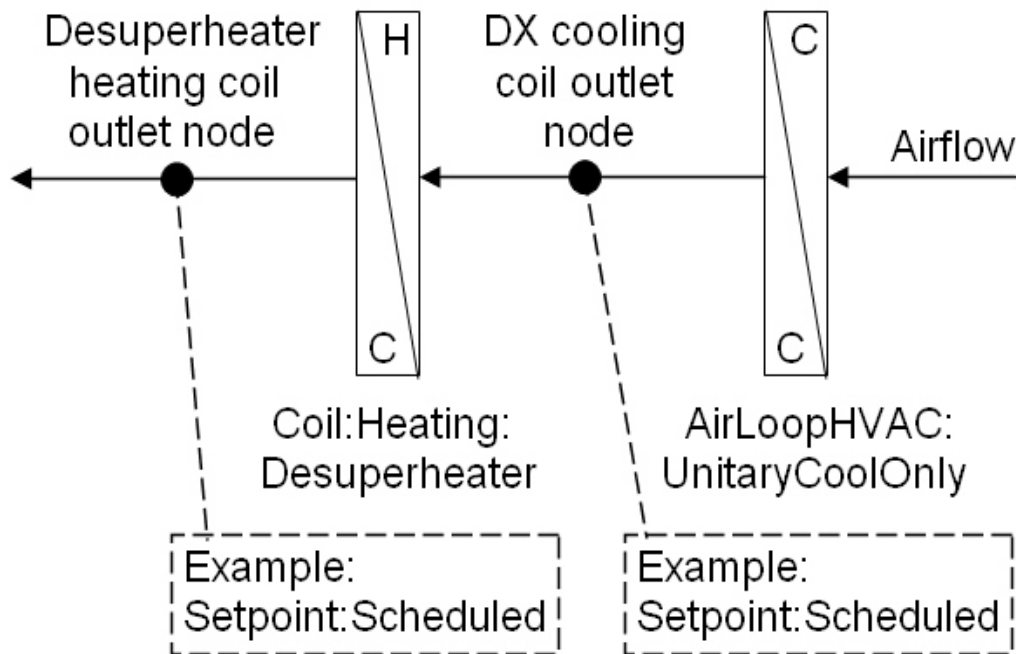


Figure 141. Desuperheater Heating Coil Used as a Reheat Coil with `AirLoopHVAC:UnitaryCoolOnly`

Since the heating provided by the desuperheater coil is based on available waste heat from another system, the selection of fan control for the air loop equipment is essential for proper modeling. When the coil's heating source is a direct expansion cooling coil (Coil:Cooling:DX:SingleSpeed, Coil:Cooling:DX:TwoSpeed, or Coil:Cooling:DX:TwoStageWithHumidityControlMode), the air loop's fan control mode may be auto fan (cycling fan cycling coil), constant fan, or variable volume since the desuperheater heating coil cycles on/off with the DX cooling coil. When the heating source is a compressor rack or condenser for refrigerated cases, the supply air fan control should be either variable volume or constant fan since the desuperheater heating coil will typically be available the entire simulation time step.

NOTE: Use of the desuperheater heating coil in variable air volume systems should be done with caution since the model assumption of a fixed heat reclaim recovery efficiency may not be valid if the air flow rate over the coil varies significantly.

The following sections describe the calculations used for both the load-based and temperature-based strategies that can control the desuperheater heating coil.

Model Inputs

A minimum of seven inputs must be defined for the desuperheater heating coil. The user must input the coil's name, an availability schedule name, and the heat reclaim recovery efficiency (default of 25%, with a range of 0% to 30% except for detailed refrigeration condensers which have a default of 80%, with a range of 0% to 90%). The next two inputs are the heating coil's inlet and outlet air node names. The user must also enter the desuperheater heat source type and name which are validated when the model inputs are read into the program.

The final two fields are optional. The first of these is the coil temperature setpoint node name. This field is used when the desuperheater heating coil is controlled based on an air loop temperature and a setpoint manager (Ref. SetpointManager:*) is used to place a temperature setpoint on this air node. The latter of these optional fields is the parasitic electric load. This field allows the user to define the electric energy use of parasitic equipment associated with the desuperheater heating coil (e.g. control valves).

Model Description

The model calculates the thermal performance of the heating coil based on the control strategy used. When a temperature setpoint node name is not entered, the control type is assumed to be load-based operation. This method should be used when this coil is specified as the air reheat coil for high humidity control with the compound object AirLoopHVAC:Unitary:Furnace:HeatCool or AirLoopHVAC:UnitaryHeatCool. If the temperature setpoint node name is entered, the control type is temperature-based operation. This method is used when the heating coil is controlled to maintain a dry-bulb temperature setpoint in an air loop.

Load-Based Operation

When load-based operation is selected, a load to be met by the heating coil is requested by the parent object (e.g., AirLoopHVAC:Unitary:Furnace:HeatCool). The model first calculates the heating coil capacity based on the total condenser waste heat rejected by the heating source.

$$\dot{Q}_{coil} = \dot{Q}_{cond} \times \epsilon_{reclaim}$$

where:

\dot{Q}_{coil} = heating capacity of the desuperheater coil (W)

\dot{Q}_{cond} = total amount of condenser waste heat rejected by the heating source (W)

$\mathcal{E}_{reclaim}$ = heat reclaim recovery efficiency

The model then compares the coil's heating capacity to the heating load being requested. If the heating capacity is greater than the heating load, then the amount of reclaimed heat (average heating rate over the simulation time step) is set equal to the requested load and the desuperheater heating coil will cycle off when the requested load is satisfied.

$$\dot{Q}_{reclaim} = \dot{Q}_{load}$$

Otherwise the amount of reclaimed heat is set equal to the coil's heating capacity and the desuperheater heating coil will operate the entire time that the waste heat source operates.

$$\dot{Q}_{reclaim} = \dot{Q}_{coil}$$

In either case, the coil's leaving air temperature is then calculated based on the amount of heat recovered and the air mass flow rate through the coil.

$$T_{out} = T_{in} + \frac{\dot{Q}_{reclaim}}{\dot{m}C_p}$$

where:

T_{out} = coil outlet air temperature (°C)

T_{in} = coil inlet air temperature (°C)

\dot{m} = air mass flow rate through the heating coil (kg/s)

C_p = specific heat of air entering the heating coil (J/kg-C)

A final calculation is made to determine the runtime fraction of the desuperheater heating coil. Since the maximum amount of reclaim heat available is dependent on the runtime fraction of the waste heat source, the runtime fraction of the desuperheater heating coil is the product of the waste heat source's runtime fraction and the ratio of the amount of heat reclaimed to the desuperheater coil's heating capacity.

$$RTF_{coil} = RTF_{source} \left(\frac{\dot{Q}_{reclaim}}{\dot{Q}_{coil}} \right)$$

where:

RTF_{coil} = runtime fraction of the desuperheater heating coil

RTF_{source} = runtime fraction of the desuperheater heating source

The heating coil's outlet air humidity ratio and air mass flow rate are simply set equal to the coil inlet air values. The outlet air enthalpy is calculated based on the outlet air dry-bulb temperature and the outlet air humidity ratio.

$$h_{out} = PsyHFnTdbW(T_{out}, \omega_{out})$$

where

h_{out} = enthalpy of the air leaving the heating coil, J/kg

ω_{out} = humidity ratio of the air leaving the heating coil, kg/kg

$PsyHFnTdbW$ = EnergyPlus psychrometric function, returns enthalpy given dry-bulb temperature and humidity ratio

Temperature-Based Operation

When temperature-based operation is selected, the model first calculates the heating coil capacity based on the total condenser waste heat rejected by the heating source (same as for load-based operation).

$$\dot{Q}_{coil} = \dot{Q}_{cond} \times \epsilon_{reclaim}$$

The model then calculates the heating load required to reach the desired setpoint temperature.

$$\dot{Q}_{load} = \dot{m}C_p (T_{setpoint} - T_{in})$$

where:

\dot{Q}_{load} = heating load to meet the desired dry-bulb temperature setpoint (W)

$T_{setpoint}$ = desired dry-bulb temperature setpoint (°C)

A comparison is then made to determine if the desuperheater heating coil should be energized. If the calculated load is less than or equal to 0, the coil's inlet air dry-bulb temperature is at or above the setpoint. In this case the desuperheater heating coil is not active and the outlet air dry-bulb temperature is set equal to the inlet air dry-bulb temperature.

$$\dot{Q}_{reclaim} = 0$$

$$T_{out} = T_{in}$$

If the amount of heat required to meet the setpoint is larger than the desuperheater coil's heating capacity, then the amount of reclaimed heat is set equal to the coil heating capacity and the outlet air dry-bulb temperature is calculated.

$$\dot{Q}_{reclaim} = \dot{Q}_{coil}$$

$$T_{out} = T_{in} + \frac{\dot{Q}_{reclaim}}{\dot{m}C_p}$$

If the amount of heat required to meet the setpoint is less than the desuperheater coil's heating capacity, the amount of reclaimed heat is set equal to the calculated heating load and the outlet air dry-bulb temperature is set equal to the setpoint temperature. In this case the desuperheater heating coil will cycle off when the requested load is satisfied.

$$\dot{Q}_{reclaim} = \dot{Q}_{load}$$

$$T_{out} = T_{setpoint}$$

A final calculation is made to determine the runtime fraction of the desuperheater heating coil. Since the maximum amount of reclaim heat available is dependent on the runtime fraction of the waste heat source, the runtime fraction of the desuperheater heating coil is the product of the waste heat source's runtime fraction and the ratio of the amount of heat reclaimed to the desuperheater coil's heating capacity.

$$RTF_{coil} = RTF_{source} \left(\frac{\dot{Q}_{reclaim}}{\dot{Q}_{coil}} \right)$$

The heating coil's outlet air humidity ratio and air mass flow rate are simply set equal to the coil inlet air values. The outlet air enthalpy is calculated based on the outlet air dry-bulb temperature and the outlet air humidity ratio.

$$h_{out} = P_{sy} H F n Tdb W (T_{out}, \omega_{out})$$

Parasitic Electric Load

The parasitic electric load attributed to the desuperheater heating coil is calculated using the user-supplied input value ($P_{parasitic}$) and the desuperheater heating coil runtime fraction calculated above.

$$P_{elec} = P_{parasitic} \times RTF_{coil}$$

The model assumes that this electric load is small and does not contribute to heating the air stream.

References

- Kosar, D. 2006. Dehumidification Enhancements, ASHRAE Journal, Vol. 48, No. 2, February 2006. <http://www.ashrae.org>
- Kosar, D. et al. 2006. Dehumidification Enhancement of Direct Expansion Systems Through Component Augmentation of the Cooling Coil. 15th Symposium on Improving Building Systems in Hot and Humid Climates, July 24-26, 2006. <http://www.hothumidsymposium.org/>
- Nimmo, B.G. et al. 1993. DEAC: Desiccant Enhancement of Cooling-Based Dehumidification. ASHRAE Transactions, Vol.99, Part 1, Paper number CH-93-4-4, pp. 842-848. <http://www.ashrae.org>

Desuperheater-Recovery-Based Water Heating Coil

Overview

The input object Coil:WaterHeating:Desuperheater provides a model that simulates the thermal performance of a refrigerant-to-water heating coil and the electric consumption of its water circulation pump, control valves, and other auxiliary devices. The model assumes that the heating energy provided by this coil is reclaimed from the superheated refrigerant gas leaving a compressor and does not impact the performance of the compressor. The objects from which this coil can obtain its heating energy are:

Coil:Cooling:DX:SingleSpeed
Coil:Cooling:DX:TwoSpeed
Coil:Cooling:DX:TwoStageWithHumidityControlMode
Refrigeration:CompressorRack
Refrigeration:Condenser* (multiple objects)

The amount of waste heat in the superheat region is explicitly calculated for the detailed refrigeration condenser. For the detailed condenser, the maximum heat recovery efficiency applied to the available superheat energy is 90%. For the other systems, including the refrigeration compressor rack, the amount of available superheat is simply a percentage of the total heat being rejected by the direct expansion (DX) system's condenser. Approximately 25-30% of the energy rejected by typical refrigeration system condensers is to reduce the superheated refrigerant vapor temperature to the condensing temperature. Recovery efficiencies higher than 30% may cause the refrigerant gas to condense which in turn impacts the performance of the refrigeration system. For this reason, the maximum heat reclaim recovery efficiency for this coil for these sources is 30%. The model includes the ability to modify the heat reclaim recovery efficiency based on variations in inlet water temperature and outdoor air dry-bulb temperature.

NOTE: When the heating source is a refrigerated case compressor rack, the heat rejection location in the Refrigeration:CompressorRack object must be "Outdoors". If the compressor rack heat rejection location is "Zone", the total amount of heat rejection available for reclaim (e.g., by this desuperheater heating coil) is set to zero by the compressor rack object and the simulation proceeds.

To model a desuperheater water heating coil, the input data file must include the following objects:

- Coil:WaterHeating:Desuperheater
- WaterHeater:Mixed
- Coil:Cooling:DX:* or Refrigeration:CompressorRack, or Refrigeration:Condenser:*

A schematic diagram showing the desuperheater water heating coil with its water heater and DX system condenser is shown below.

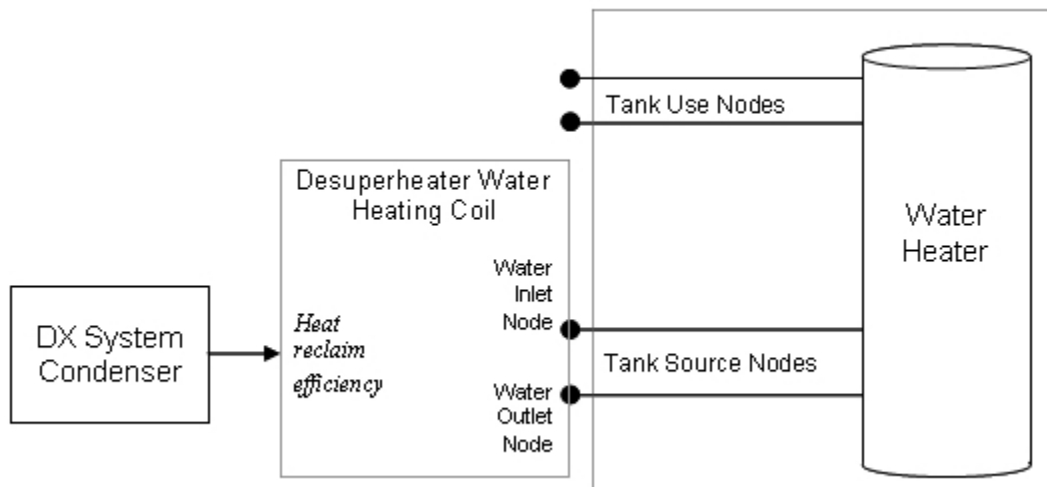


Figure 142. Schematic of Desuperheater Water Heating Coil

Model Description

The desuperheater heating coil input requires a setpoint temperature schedule and dead band temperature difference, which are independent from the setpoint temperature schedule and dead band temperature difference for the heater (element or burner) associated with the water heater tank. The cut-in temperature for the desuperheater coil is defined as the desuperheater coil's setpoint temperature minus its dead band temperature difference.

$$T_{cut-in} = T_{set\ point} - T_{dead\ band}$$

where:

T_{cut-in} = Cut-in temperature (°C)

$T_{set\ point}$ = Setpoint temperature (°C)

$T_{dead\ band}$ = Dead band temperature difference (°C)

Desuperheater water heating coils are typically used to offset energy consumption by the water tank's heater (element or burner). Therefore, the cut-in temperature of the desuperheater coil should always be higher than the water heater tank's setpoint temperature. At times when the water heater tank's setpoint temperature is greater than or equal to the cut-in temperature of the desuperheater coil, the desuperheater is disabled and the water heater tank heating element is used to heat the water. An additional input parameter sets the maximum inlet water temperature allowed for desuperheater heat reclaim. Any time the inlet water temperature to the desuperheater coil is above this maximum allowed temperature, heat reclaim is restricted so that the tank water does not exceed this temperature.

When the desuperheater coil is scheduled off by its availability schedule, the water heater tank's setpoint temperature is greater than or equal to the desuperheater coil cut-in temperature, or if the DX system's compressor (source of heat reclaim) is off, the water heating capacity and water mass flow rate for the desuperheater coil are set to zero and the water heater tank is simulated with the desuperheater heating coil disabled. Otherwise, the simulation proceeds by first determining the maximum amount of heat available for heat reclaim. If a heat reclaim efficiency modifier curve object is specified by the user, this bi-

quadratic curve is evaluated using the desuperheater coil inlet water temperature and the outdoor air dry-bulb temperature; otherwise, the output is assumed to be equal to 1.

$$\begin{aligned} & \text{IF } (HEffFTemp \text{ is specified}) \text{ THEN} \\ & \quad HEffFTemp = \text{MAX} \left(0.0, f \left(T_{\text{water,inlet}}, T_{\text{DB,outdoor}} \right) \right) \\ & \text{ELSE} \\ & \quad HEffFTemp = 1.0 \\ & \text{ENDIF} \end{aligned}$$

where:

$HEffFTemp$ = Heat reclaim efficiency modifier

$T_{\text{water,inlet}}$ = Desuperheater coil inlet water temperature (°C)

$T_{\text{DB,outdoor}}$ = Outdoor air dry-bulb temperature (°C)

Except for the detailed condenser source, heat reclaim recovery efficiencies higher than 30% may cause the refrigerant gas to condense which in turn impacts the performance of the refrigeration system. The model is unable to account for this refrigeration system impact, so the product of the rated heat reclaim recovery efficiency and heat reclaim efficiency modifier is not allowed to exceed 0.3. For the detailed refrigeration condenser, the limit for the heat reclaim efficiency has been set at 0.9 as follows:

$$\text{IF } (\varepsilon_{\text{reclaim,rated}})(HEffFTemp) > \text{Limit}, \text{ then } HEffFTemp = \frac{\text{Limit}}{\varepsilon_{\text{reclaim,rated}}}$$

where:

Limit = 0.3 for most sources, =0.9 for detailed condensers

$\varepsilon_{\text{reclaim,rated}}$ = rated heat reclaim recovery efficiency

The heating capacity of the desuperheater coil is then calculated based on the product of total amount of condenser waste heat rejected by the heating source, the desuperheater coil's rated heat reclaim recovery efficiency, and the heat reclaim efficiency modifier.

$$\dot{Q}_{\text{coil}} = \left(\dot{Q}_{\text{cond}} \right) (\varepsilon_{\text{reclaim,rated}}) HEffFTemp / DXSystemPLR$$

where:

\dot{Q}_{coil} = water heating capacity of the desuperheater coil (W)

\dot{Q}_{cond} = average rate of condenser waste heat rejected by the heating source (W)

$DXSystemPLR$ = part load ratio for the desuperheater coil heating source (1.0 for refrigerated case compressor rack and detailed refrigeration condenser)

The desuperheater is assumed to have a water circulation pump located downstream of the refrigerant-to-water coil. Pump heat is added to the desuperheater water heating capacity based on a user-defined fraction as follows:

$$\dot{Q}_{\text{coil}} = \dot{Q}_{\text{coil}} + P_{\text{pump}} \left(\text{Frac}_{\text{pumptowater}} \right)$$

where:

P_{pump} = water pump power (W)

$Frac_{pumptowater}$ = fraction of pump heat to water

The desuperheater is assumed to have a water circulation pump located downstream of the refrigerant-to-water coil. Pump heat is added to the desuperheater water heating capacity based on a user-defined fraction as follows:

$$\dot{Q}_{coil} = \dot{Q}_{coil} + P_{pump} (Frac_{pumptowater})$$

where:

$T_{water,outlet}$ = desuperheater outlet water temperature (°C)

\dot{m}_{water} = desuperheater water mass flow rate when coil is operating (kg/s)

C_p = specific heat of water entering the heating coil (J/kg-C)

Simulation of the desuperheater heating coil is based on its current mode of operation. This mode of operation is either floating (heating coil off and tank water temperature has not fallen below the heater cut-in temperature) or heating (tank water temperature dropped below the heater cut-in temperature on a previous time step but was unable to reach the heater setpoint temperature). Each mode is handled differently and they will be discussed separately.

Float Mode

When the water heater tank temperature is floating between the desuperheater coil's cut-in and cut-out temperatures at the end of the previous simulation time step, both the desuperheater coil and the water heater tank's heating element are disabled and a resulting tank temperature is calculated. If the resulting tank temperature is below the desuperheater coil cut-in temperature the desuperheater coil part-load ratio is estimated using a ratio of temperature differences as shown below. The part-load ratio can not be less than zero or greater than the part-load ratio of the DX system from which the desuperheater coil is reclaiming heat.

$$PLR = MIN \left(DXSystemPLR, MAX \left(0.0, \left(\frac{T_{cut-in} - T_{tank,float}}{T_{tank,initial} - T_{tank,float}} \right) \right) \right)$$

where:

PLR = part load ratio of desuperheater heating coil

$T_{tank,float}$ = tank temperature in float mode when heating capacity is set to zero (°C)

$T_{tank,initial}$ = tank water temperature at the beginning of a simulation time step (°C)

Since the water pump is assumed to cycle on and off with the desuperheater coil, the average water mass flow rate through the desuperheater coil is then set proportional to the PLR calculated above:

$$\dot{m}_{water,avg} = \dot{m}_{water} (PLR)$$

where:

$\dot{m}_{water,avg}$ = average desuperheater water mass flow rate for the time step (kg/s)

The water tank temperature is then calculated based on desuperheater heating coil operation (outlet water temperature and average water mass flow rate as calculated above) and with the water tank's heater element enabled. If the resulting water tank temperature is above the desuperheater coil's setpoint temperature, then the part-load ratio is reduced in the same manner described in the heating mode section below.

Heating Mode

When the desuperheater heating coil is in heating mode at the end of the previous simulation time step (i.e., desuperheater coil operated during the previous simulation time step but was unable to achieve the setpoint temperature), both the desuperheater and the water heater tank's heating element are enabled. The desuperheater coil outlet water temperature is calculated (as described above) and the water mass flow rate through the desuperheater coil is set to the maximum operating flow rate:

$$\dot{m}_{water, avg} = \dot{m}_{water}(DXSystemPLR)$$

If the resulting tank water temperature is above the desuperheater coil's setpoint (cut-out) temperature, the part load ratio of the desuperheater coil is reduced and the water heater tank is simulated again. The process is performed iteratively until the part load ratio of the desuperheater coil achieves the desired setpoint temperature.

Model Outputs

After completing the float mode or heating mode calculations and the final desuperheater part load ratio has been determined, the output (report) variables are calculated as follows:

$$Desuperheater\ Part\ Load\ Ratio = MIN\left(\frac{\dot{m}_{water, avg}}{\dot{m}_{water}}, DXSystemPLR\right)$$

$$Desuperheater\ Water\ Heating\ Rate\ (W) = \dot{Q}_{coil}(PLR)$$

$$Desuperheater\ Water\ Heating\ Energy\ (J) = \dot{Q}_{coil}(PLR)(3600)(t_{sys})$$

$$Desuperheater\ Pump\ Electric\ Power\ (W) = P_{pump}(PLR)$$

$$Desuperheater\ Pump\ Electric\ Consumption\ (J) = P_{pump}(PLR)(3600)(t_{sys})$$

$$Desuperheater\ Heat\ Reclaim\ Efficiency\ Modifier\ Curve\ Output = HEffFTemp$$

$$Desuperheater\ On-Cycle\ Parasitic\ Electric\ Power\ (W) = P_{parasiticload, on}(PLR)$$

$$Desuperheater\ On-Cycle\ Parasitic\ Electric\ Consumption\ (J) = P_{parasiticload, on}(PLR)(3600)t_{sys}$$

$$Desuperheater\ Off-Cycle\ Parasitic\ Electric\ Power\ (W) = P_{parasiticload, off}(1 - PLR)$$

$$Desuperheater\ Off-Cycle\ Parasitic\ Electric\ Consumption\ (J) = P_{parasiticload, off}(1 - PLR)(3600)(t_{sys})$$

where:

$P_{parasiticload,on}$ = on-cycle parasitic electric load (W)

$P_{parasiticload,off}$ = off-cycle parasitic electric load (W)

t_{sys} = HVAC system simulation time step (hours)

Note: All output variables, including off-cycle parasitic electrical power and consumption, equal 0 when the desuperheater heating coil availability schedule equals 0.

Heat Exchanger Assisted Air Cooling Coil Systems

An air-to-air heat exchanger can be used to enhance the dehumidification performance of a conventional cooling coil. EnergyPlus has two compound objects to model this scenario: `CoilSystem:Cooling:DX:HeatExchangerAssisted` and `CoilSystem:Cooling:Water:HeatExchangerAssisted`. The input syntax for these compound objects can be found in the EnergyPlus Input/Output Reference.

As shown in Figure 143, the air-to-air heat exchanger pre-conditions the air entering the cooling coil, and reuses this energy to post-condition the supply air leaving the cooling coil. This heat exchange process improves the latent removal performance of the cooling coil by allowing it to dedicate more of its cooling capacity toward dehumidification (lower sensible heat ratio).

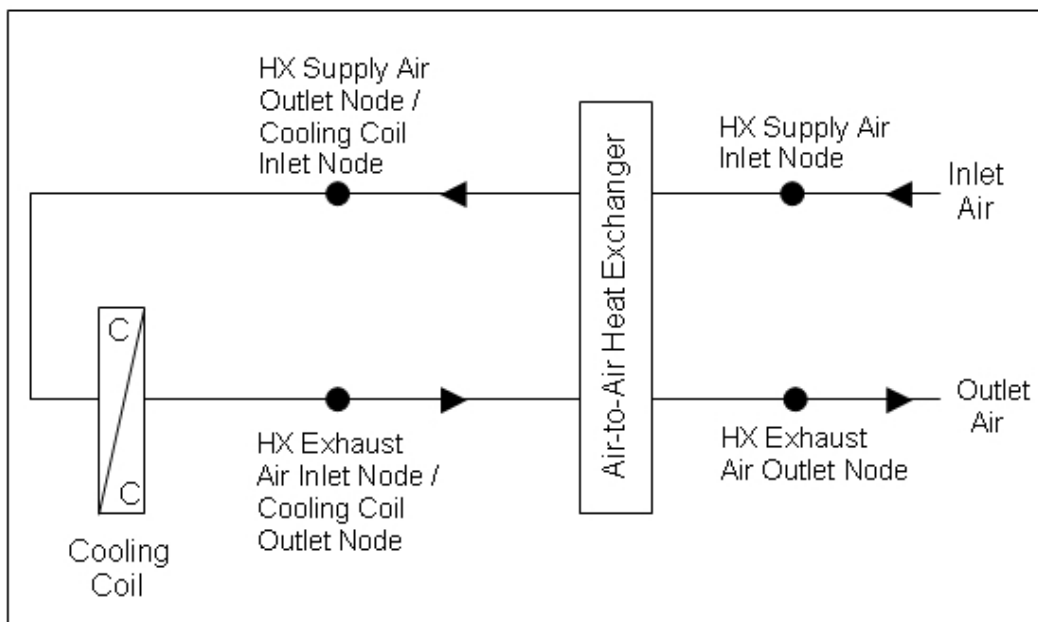


Figure 143. Schematic of a Heat Exchanger Assisted Cooling Coil

NOTE: Node naming shown in Figure 143 is representative for `HeatExchanger:AirToAir:SensibleAndLatent`. For `HeatExchanger:AirToAir:FlatPlate`, the exhaust air nodes are referred to as 'secondary air' nodes. For `HeatExchanger:Desiccant:BalancedFlow` (heat exchanger

assisted DX coil only), the supply air nodes are referred to as 'regeneration air' nodes and the exhaust air nodes as 'process air' nodes.

The dehumidification performance of cooling coils can be enhanced using a sensible-only heat exchanger (e.g., heat pipes, runaround coils, cross-flow heat exchangers). For example, a `CoilSystem:Cooling:DX:HeatExchangerAssisted` object could be used with a `HeatExchanger:AirToAir:FlatPlate` object or `HeatExchanger:AirToAir:SensibleAndLatent` object with only sensible effectiveness specified (see resulting psychrometric process in Figure 144). The dehumidification performance of cooling coils can also be enhanced using heat exchangers that transfer both sensible and latent energy (e.g., rotary desiccant heat exchanger). For example, a `CoilSystem:Cooling:DX:HeatExchangerAssisted` object could be used with a `HeatExchanger:Desiccant:BalancedFlow` object (see resulting psychrometric process in Figure 145).

Note that while the `HeatExchanger:AirToAir:SensibleAndLatent` and `HeatExchanger:Desiccant:BalancedFlow` objects can transfer both sensible and latent heat between two air streams, the `HeatExchanger:Desiccant:BalancedFlow` object with appropriate Performance Data Type object is the proper heat exchanger model to use for the heat exchanger assisted cooling coil described here. The `HeatExchanger:AirToAir:SensibleAndLatent` object should only be used to model the sensible-only heat exchange case (Figure 144).

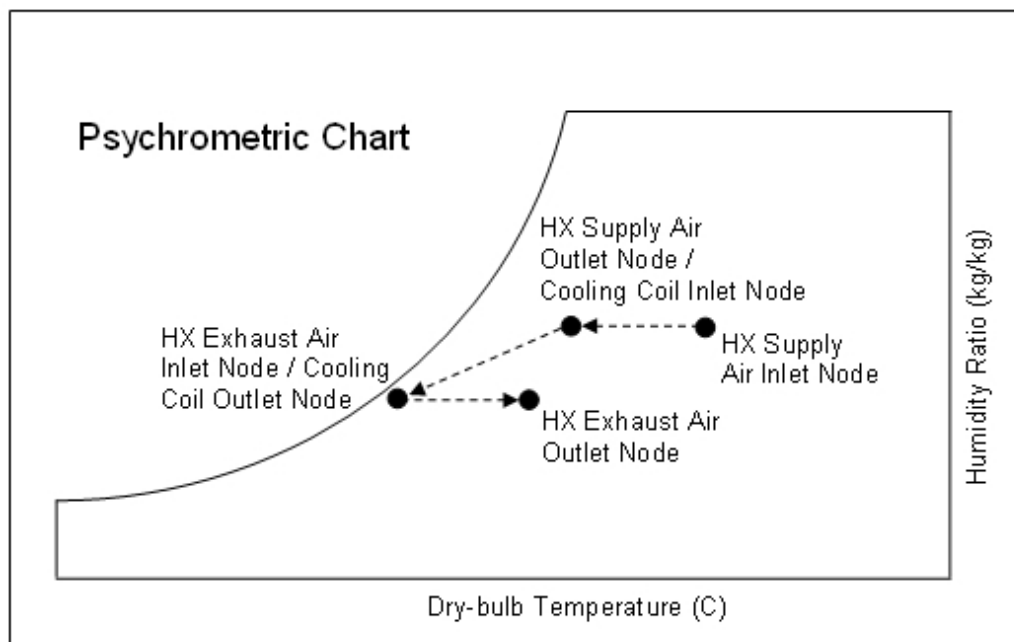


Figure 144. Psychrometric Process for Heat Exchanger Assisted Cooling Coil (Sensible HX Only)

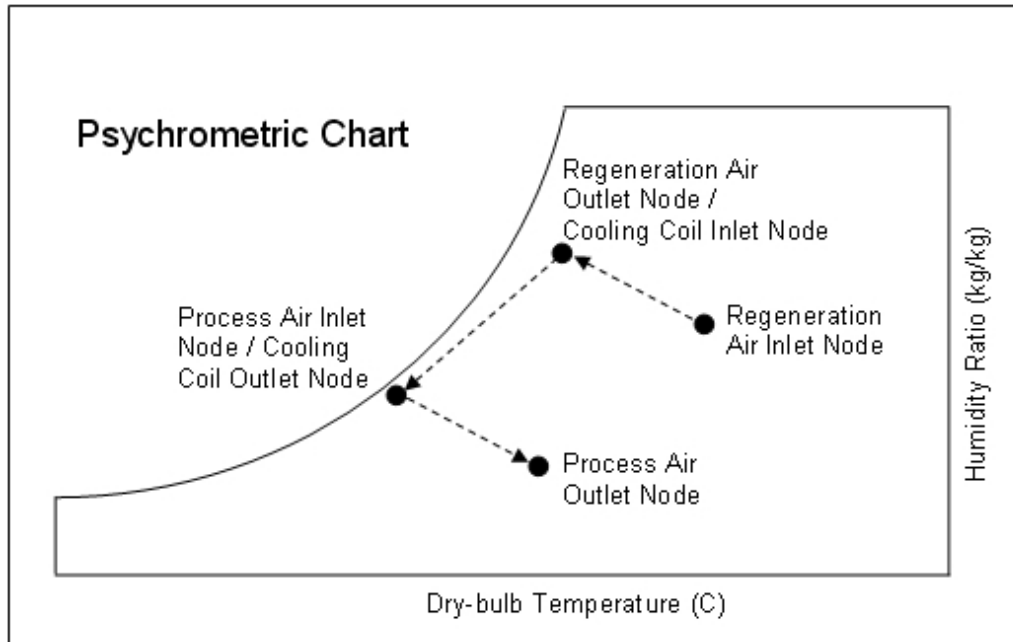


Figure 145. Psychrometric Process for Heat Exchanger Assisted Cooling Coil (Sensible+Latent HX)

Modeling of the heat exchanger assisted cooling coil is performed by consecutively modeling the air-to-air heat exchanger and the cooling coil until convergence on a solution is achieved. The detailed modeling calculations for the individual components (air-to-air heat exchangers and cooling coils) are described elsewhere in this document.

Modeling of the heat exchanger assisted cooling coil begins by initializing the air mass flow rate (based on the air mass flow rate placed on the compound object's inlet air node) and passing this value to the exhaust air inlet node of the air-to-air heat exchanger. The heat exchanger and cooling coil are then successively modeled using the calculation routines specific to the type of heat exchanger and cooling coil selected. The air temperature exiting the cooling coil is compared with the air temperature exiting the cooling coil on the previous modeling iteration for this simulation time step. Convergence is reached when the change in this air temperature for successive iterations is within a specified tolerance (0.0005°C). Consecutive modeling of the heat exchanger and cooling coil is terminated and a warning message is issued if the number of modeling iterations exceeds 50.

For the CoilSystem:Cooling:DX:HeatExchangerAssisted object, heat exchanger operation can be controlled based on high humidity levels in a zone. By default, the heat exchanger is assumed to always provide its heat transfer when the associated DX cooling coil is operating and no humidity control mechanism is specified. However, the heat exchanger's energy transfer may be controlled (i.e., turned on and off) based on a zone air humidity level using either a humidistat alone (Figure 146) or a humidistat and a maximum humidity setpoint manager (Figure 147) depending on the HVAC system that is utilizing the heat exchanger assisted cooling coil.

The heat exchanger assisted DX cooling coil may be used with furnaces or unitary systems located in an air loop (ref. AirLoopHVAC:Unitary:Furnace:HeatCool or AirLoopHVAC:UnitaryHeatCool). These system objects have three options for dehumidification control (None, Multimode, and CoolReheat). When no dehumidification control is specified in the furnace or unitary system object (None), the heat exchanger is always active when the cooling coil is operating. When multimode or coolreheat dehumidification control is specified, a humidistat is required as shown in Figure 146. For the case of multimode dehumidification control, the heat exchanger is only active when the zone air humidity level is above the humidistat setpoint (i.e., the system's cooling coil can't meet

the latent cooling load when operating without heat exchanger energy transfer) while the AC system operates to meet the sensible (dry-bulb cooling thermostat) load. For the case of coolreheat dehumidification control, the heat exchanger is always active when the cooling coil operates and this system tries to meet both the sensible (thermostat) and latent (humidistat) loads.

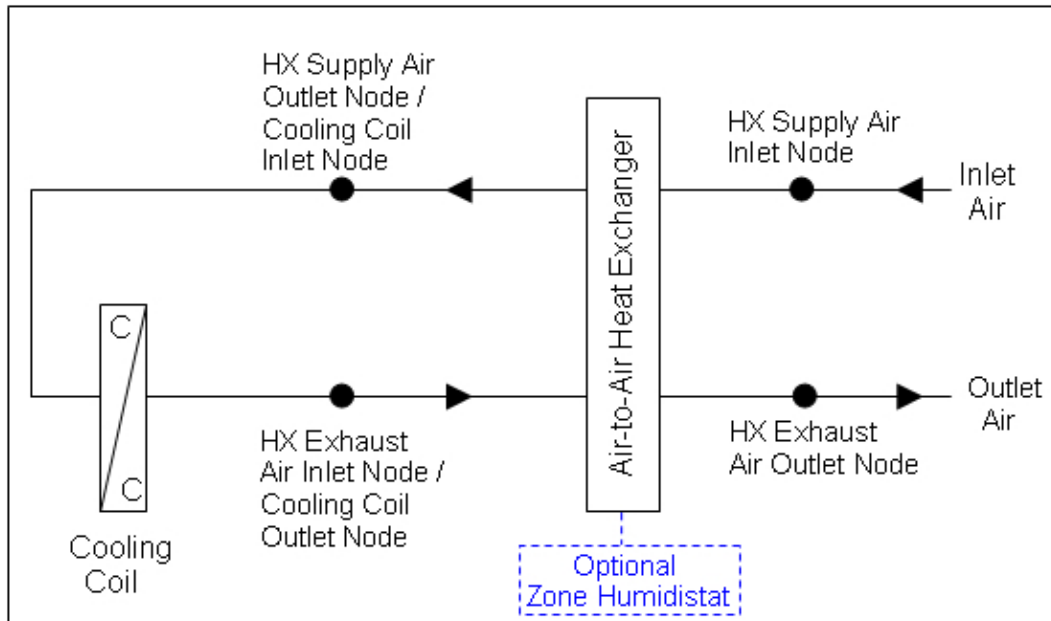


Figure 146. Schematic of a heat exchanger assisted DX cooling coil with optional humidistat

The heat exchanger assisted DX cooling coil may also be used with a DX system located in an air loop (ref. AirLoopHVAC:UnitaryCoolOnly). This system object also has three options for dehumidification control (None, Multimode, and CoolReheat). When no dehumidification control is specified (None), the heat exchanger is always active when the cooling coil is operating. When multimode or coolreheat dehumidification control is specified, a humidistat and a maximum humidity setpoint manager are required as shown in Figure 147 (setpoint needs to be placed on the DX system's control node). For multimode dehumidification control, the heat exchanger is only active when the zone humidity levels are above the humidistat setpoint (i.e., the system's cooling coil can't meet the maximum humidity ratio setpoint when operating without heat exchanger energy transfer) while the AC system operates to meet the sensible (dry-bulb cooling thermostat) load. For coolreheat dehumidification control, the heat exchanger is always active when the cooling coil operates and this system tries to meet both the sensible (thermostat) and latent (humidistat) loads.

When the heat exchanger assisted cooling coil is used with a furnace or unitary system (ref. AirLoopHVAC:Unitary:Furnace:HeatCool or AirLoopHVAC:UnitaryHeatCool) or DX system (ref. AirLoopHVAC:UnitaryCoolOnly) located in an air loop (or DX system used in an outside air system), an economizer function may be customized as necessary. For economizer control, an outdoor air controller (ref. Controller:OutdoorAir) is used to define the economizer control inputs and determine when economizer mode is active. The heat exchanger (ref. HeatExchanger:*) object provides an economizer lockout feature which disables heat recovery any time the economizer is active. This feature can be turned on and off using the heat exchanger lockout input. Heat exchanger assisted cooling coils used with the zone equipment described below disregard this economizer control feature.

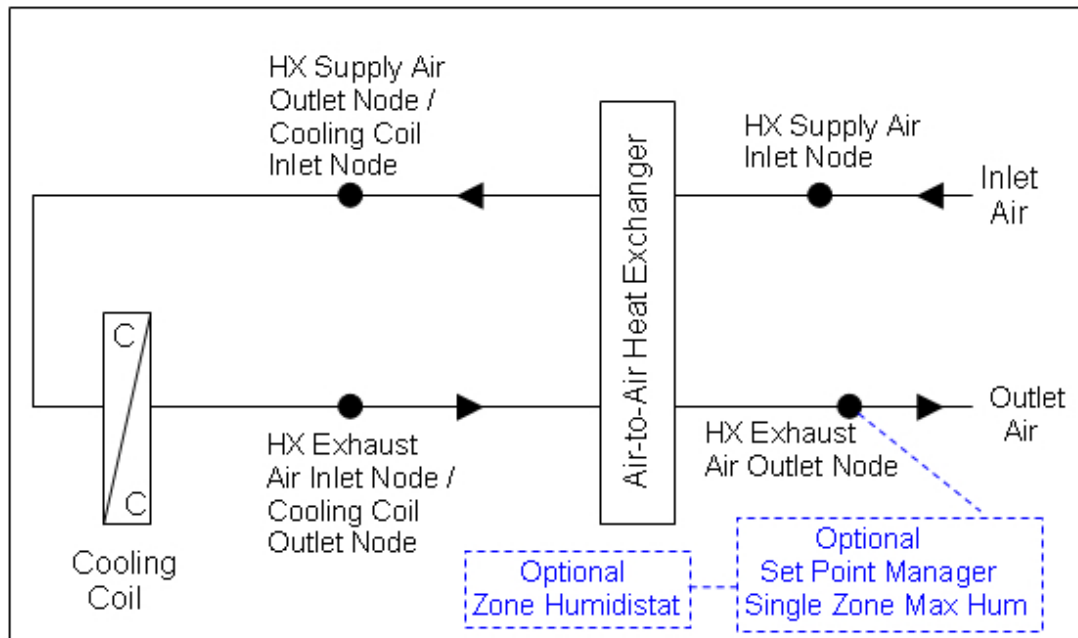


Figure 147. Schematic of Heat Exchanger Assisted DX Coil with Humidistat and Setpoint Manager

The heat exchanger assisted DX cooling coil may also be used with the unitary changeover bypass system and the unitary air-to-air heat pump system (ref. `AirLoopHVAC:UnitaryHeatCool:VAVChangeoverBypass` and `AirLoopHVAC:UnitaryHeatPump:AirToAir`); however, the heat exchanger is assumed to always provide its heat transfer when the cooling coil operates and can not be turned on and off based on a zone air humidity setpoint (ref. Figure 143). Two zone air conditioners may also use this heat exchanger/coil assembly model for improved dehumidification. The first type is the packaged terminal heat pump (ref. `ZoneHVAC:PackagedTerminalHeatPump`) where the heat exchanger's heat transfer is always active whenever the cooling coil operates (ref. Figure 143). The second type is the window air conditioner (ref. `ZoneHVAC:WindowAirConditioner`) where the heat exchanger's heat transfer is always active when the cooling coil operates and no humidity control mechanism is specified (ref. Figure 143), or the heat exchanger's heat transfer may be controlled based on zone air humidity level when a humidistat and high humidity setpoint manager are specified (maximum humidity ratio setpoint must be placed on the heat exchanger's exhaust air outlet node). For this case, the heat exchanger is only active when the zone air humidity level is above the humidistat setpoint (i.e., the system's cooling coil can't meet the maximum humidity ratio setpoint when operating without heat exchanger energy transfer) while only the sensible (dry-bulb cooling thermostat) load is met by the AC system (ref. Figure 147).

For the `CoilSystem:Cooling:Water:HeatExchangerAssisted` object, there is currently no method to enable or disable heat exchange based on zone air humidity level. Heat exchange will occur whenever the heat exchanger is available to operate (via its availability schedule) and a temperature difference exists between the two air streams.

References

- Kosar, D. 2006. Dehumidification Enhancements, ASHRAE Journal, Vol. 48, No. 2, February 2006. <http://www.ashrae.org>
- Kosar, D. et al. 2006. Dehumidification Enhancement of Direct Expansion Systems Through Component Augmentation of the Cooling Coil. 15th Symposium on Improving Building Systems in Hot and Humid Climates, July 24-26, 2006. <http://www.hothumidsymposium.org/>

Nimmo, B.G. et al. 1993. DEAC: Desiccant Enhancement of Cooling-Based Dehumidification. ASHRAE Transactions, Vol.99, Part 1, Paper number CH-93-4-4, pp. 842-848. <http://www.ashrae.org>

Single-Speed Electric Heat Pump DX Water Heating Coil

Overview

The input object Coil:WaterHeating:AirToWaterHeatPump provides a model used in a heat pump water heater (HPWH) consisting of a water heater tank (e.g., WaterHeater:Mixed), a direct expansion (DX) “coil” (i.e., an air-to-water DX compression system which includes a water heating coil, air coil, compressor, and water pump), and a fan to provide air flow across the air coil associated with the DX compression system. These objects work together to model a system which heats water using zone air, outdoor air, or a combination of zone and outdoor air as the primary heat source. The heat pump water heater (Ref. WaterHeater:HeatPump), water heater tank (Ref. WaterHeater:Mixed), and fan (Ref. Fan:*) objects are described elsewhere in this document.

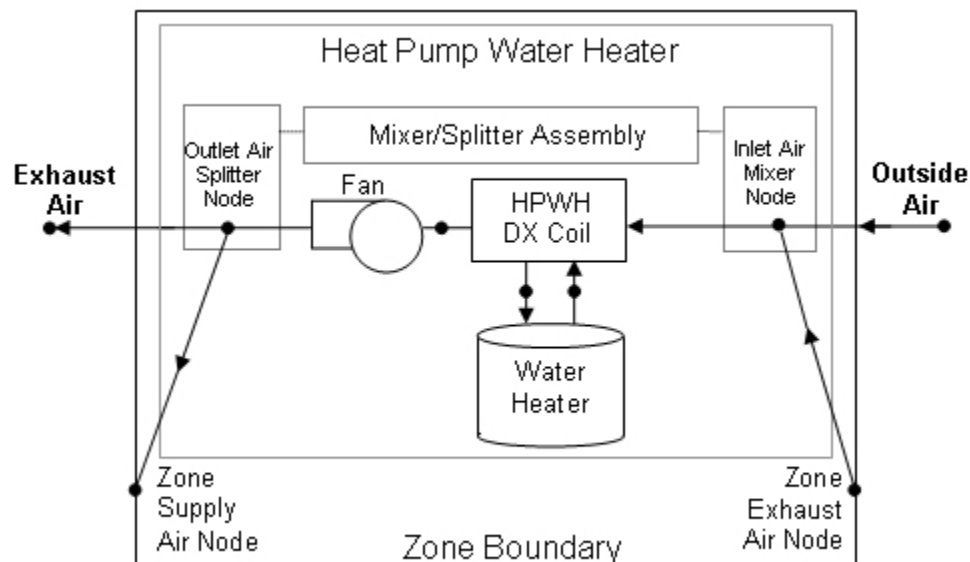


Figure 148. Schematic of a Heat Pump Water Heater using Optional Mixer/Splitter Nodes

The heat pump water heater DX coil model described here determines the thermal performance and energy consumption of the DX compression system, which includes a water heating coil (condenser), a coil used to extract heat from air (evaporator), and the compressor. This model also simulates the performance of a condenser water pump that is assumed to cycle on and off with the compressor.

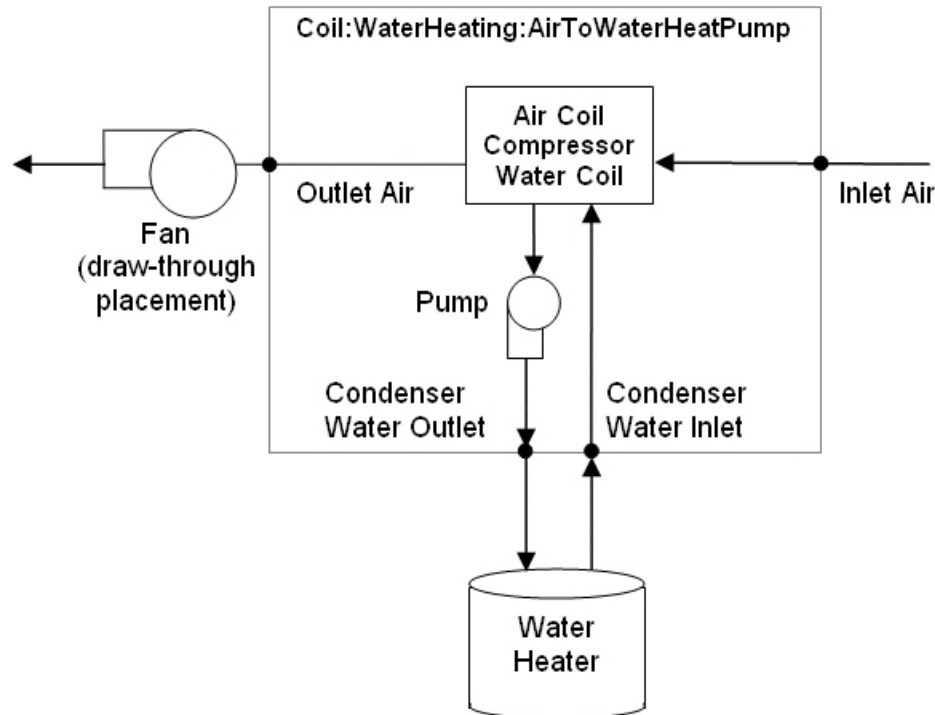


Figure 149. Schematic of the Heat Pump Water Heater DX Coil

Virtually all of the inputs to this DX coil model relate to its water heating performance and energy consumption. The air-side cooling capacity of this DX system is derived from user inputs and some model assumptions. The sensible/latent cooling capacity split is defined by the user at rated conditions, and the model adjusts this split at off-rated conditions.

Model Description

The user must input the heating capacity, coefficient of performance (COP), and the sensible heat ratio (SHR) of the evaporator coil at rated inlet fluid temperatures and flow rates that are also specified by the user. The evaporator SHR should be a “gross” value, excluding any thermal impacts due to the indoor supply air fan. The user may also input up to seven performance curves that describe the change in total water heating capacity and efficiency at off-rated and part-load conditions.

- 1)** Heating capacity modifier curve (function of temperature)
- 2)** Heating capacity modifier curve (function of air flow fraction)
- 3)** Heating capacity modifier curve (function of water flow fraction)
- 4)** Heating COP modifier curve (function of temperature)
- 5)** Heating COP modifier curve (function of air flow fraction)
- 6)** Heating COP modifier curve (function of water flow fraction)
- 7)** Part load fraction correlation (function of part load ratio)

Note: The air dry-bulb or wet-bulb temperature used in the following curve objects is the inlet air temperature to the HPWH evaporator coil/fan assembly. If the fan placement specified in the WaterHeater:HeatPump compound object is draw-through, then the inlet air temperature represents the temperature of the air entering the evaporator coil itself. If blow-through fan placement is specified, then the inlet air temperature represents air conditions entering the fan located immediately in front (upstream) of the evaporator coil.

- The heating capacity as a function of temperature modifier curve defines the variation in DX coil heating capacity as a function of inlet fluid (air and water) temperatures. The curve object may use either a bi-quadratic or cubic form. The bi-quadratic curve uses inlet air temperature (dry-bulb or wet-bulb temperature based on the input field Evaporator Air Temperature Type for Curve Objects) and condenser inlet water temperature as the independent variables. The cubic curve uses inlet air (dry-bulb or wet-bulb) temperature as the independent variable. The curve should be normalized to have the value of 1.0 at the rating point temperatures specified by the user.

$$HCapFTemp = a + b(T_{evap,in}) + c(T_{evap,in})^2 + d(T_{cond,in}) + e(T_{cond,in})^2 + f(T_{evap,in})(T_{cond,in})$$

or

$$HCapFTemp = a + b(T_{evap,in}) + c(T_{evap,in})^2 + d(T_{evap,in})^3$$

where:

$T_{evap,in}$ = dry-bulb or wet-bulb temperature of the air entering the evaporator coil/fan assembly based on the Evaporator Air Temperature Type for Curve Objects specified by the user (°C)

$T_{cond,in}$ = temperature of the water entering the DX coil condenser (°C)

- The heating capacity as a function of air flow fraction modifier curve is a quadratic or cubic curve that defines the variation in DX coil heating capacity as a function of the ratio of actual air flow rate across the evaporator coil to the rated evaporator air flow rate (i.e., fraction of full load air flow rate). When used, the output of this curve should be normalized to have the value of 1.0 at an air flow fraction of 1.

$$HCapFAirFlow = a + b(ff_{air}) + c(ff_{air})^2$$

or

$$HCapFAirFlow = a + b(ff_{air}) + c(ff_{air})^2 + d(ff_{air})^3$$

where:

$$ff_{air} = \text{air flow fraction} = \left(\frac{\text{Actual air mass flowrate}}{\text{Rated air mass flowrate}} \right)$$

Note: The actual volumetric air flow rate through the evaporator coil/fan assembly for any simulation time step where the compressor is operating must be between 0.00002684 m³/s and .00008056 m³/s per watt of rated total heating capacity. The simulation will issue a warning message if this air flow range is exceeded, but the simulation will continue.

- The heating capacity as a function of water flow fraction modifier curve is a quadratic or cubic curve that defines the variation in DX coil heating capacity as a function of the ratio of actual water flow rate through the condenser to the rated condenser water flow rate (i.e., fraction of full load water flow rate). When used, the output of this curve should be normalized to have the value of 1.0 at a water flow fraction of 1.

$$HCapFWaterFlow = a + b(ff_{water}) + c(ff_{water})^2$$

or

$$HCapFWaterFlow = a + b(ff_{water}) + c(ff_{water})^2 + d(ff_{water})^3$$

where:

$$ff_{water} = \text{water flow fraction} = \left(\frac{\text{Actual water mass flow rate}}{\text{Rated water mass flow rate}} \right)$$

- The heating COP as a function of temperature modifier curve defines the variation in DX coil heating COP as a function of inlet fluid (air and water) temperatures. The curve object may use either a bi-quadratic or cubic form. The bi-quadratic curve uses inlet air temperature (dry-bulb or wet-bulb temperature based on the input field Evaporator Air Temperature Type for Curve Objects) and condenser inlet water temperature as the independent variables. The cubic curve uses inlet air (dry-bulb or wet-bulb) temperature as the independent variable. The curve should be normalized to have the value of 1.0 at the rating point temperatures specified by the user.

$$HCOPFTemp = a + b(T_{evap,in}) + c(T_{evap,in})^2 + d(T_{cond,in}) + e(T_{cond,in})^2 + f(T_{evap,in})(T_{cond,in})$$

or

$$HCOPFTemp = a + b(T_{evap,in}) + c(T_{evap,in})^2 + d(T_{evap,in})^3$$

- The heating COP as a function of air flow fraction modifier curve is a quadratic or cubic curve that defines the variation in DX coil heating COP as a function of the ratio of actual air flow rate across the evaporator coil to the rated evaporator air flow rate (i.e., fraction of full load air flow rate). When used, the output of this curve should be normalized to have the value of 1.0 at an air flow fraction of 1.

$$HCOPFAirFlow = a + b(ff_{air}) + c(ff_{air})^2$$

or

$$HCOPFAirFlow = a + b(ff_{air}) + c(ff_{air})^2 + d(ff_{air})^3$$

- The heating COP as a function of water flow fraction modifier curve is a quadratic or cubic curve that defines the variation in DX coil heating COP as a function of the ratio of actual water flow rate through the condenser to the rated condenser water flow rate (i.e., fraction of full load water flow rate). When used, the output of this curve should be normalized to have the value of 1.0 at a water flow fraction of 1.

$$HCOPFWaterFlow = a + b(ff_{water}) + c(ff_{water})^2$$

or

$$HCOPFWaterFlow = a + b(ff_{water}) + c(ff_{water})^2 + d(ff_{water})^3$$

- The part load fraction as a function of part load ratio correlation curve parameterizes the variation of electrical power input to the heat pump DX coil as a function of the part load ratio (PLR, heating delivered/steady-state heating capacity). The part load ratio divided by the part load fraction yields the runtime fraction of the DX heating coil for a given simulation time step. The part load fraction correlation accounts for efficiency losses due to compressor cycling.

$$PartLoadFrac = PLF = a + b(PLR) + c(PLR)^2$$

or

$$PartLoadFrac = a + b(PLR) + c(PLR)^2 + d(PLR)^3$$

where:

$$PLR = \text{part load ratio} = \left(\frac{\text{water heating delivered}}{\text{steady-state water heating capacity}} \right)$$

The part load fraction correlation should be normalized to a value of 1.0 when the part load ratio equals 1.0 (i.e., no efficiency losses when the compressor runs continuously for the simulation time step). For PLR values between 0 and 1 ($0 \leq PLR < 1$), the following rules apply:

$$PLF \geq 0.7 \quad \text{and} \quad PLF \geq PLR$$

If $PLF < 0.7$ a warning message is issued, the program resets the PLF value to 0.7, and the simulation proceeds. The runtime fraction of the coil is defined as PLR/PLF . If $PLF < PLR$, then a warning message is issued and the runtime fraction of the coil is limited to 1.0.

When the part load fraction correlation for a heat pump water heater DX coil is unknown, the typical part load fraction correlation for a conventional, single-speed DX cooling coil (e.g., residential or small commercial unit) may be used:

$$PLF = 0.85 + 0.15(PLR)$$

All seven curves are accessed through EnergyPlus' built-in performance curve equation manager (curve:quadratic, curve:cubic and curve:biquadratic). It is not imperative that the user utilize all coefficients shown in the equations above if their performance equation has fewer terms (e.g., if the user's part load fraction correlation curve is linear instead of quadratic, simply enter the values for a and b, and set coefficient c equal to zero).

Note: If any of the above performance curves are not specified by the user, the output of that curve is assumed to be 1 for the entire simulation.

For any simulation time step, the heating capacity of the heat pump water heater DX coil is calculated as follows:

$$\dot{Q}_{heating} = \dot{Q}_{heating,rated} (HCapFTemp)(HCapFAirFlow)(HCapFWaterFlow)$$

In a similar fashion, the coefficient of performance for the DX coil for any simulation time step is calculated using the following equation:

$$COP_{heating} = COP_{heating,rated} (HCOPFTemp)(HCOPFAirFlow)(HCOPFWaterFlow)$$

The electric power for the heat pump water heater DX coil is then calculated using the water heating capacity and coefficient of performance calculated above.

$$P_{heating} = \left(\frac{\dot{Q}_{heating}}{COP_{heating}} \right)$$

where:

$\dot{Q}_{heating}$ = water heating capacity at the current operating conditions (W)

$\dot{Q}_{heating,rated}$ = water heating capacity at rated conditions, user input (W)

$COP_{heating}$ = coefficient of performance at the current operating conditions (W/W)

$COP_{heating,rated}$ = coefficient of performance at rated conditions, user input (W/W)

$P_{heating}$ = DX heating coil power (electricity consumption rate) at the current operating conditions (W)

The heating capacity calculated above may or may not include the impact of pump heat. For this reason, the user input Condenser Pump Heat Included in Rated Heating Capacity and Rated COP is used to determine the total water heating capacity including pump heat.

IF (Condenser Pump Heat Included In Rated Heating Capacity and Rated COP)

$$\dot{Q}_{heating,total} = \dot{Q}_{heating}$$

ELSE

$$\dot{Q}_{heating,total} = \dot{Q}_{heating} + P_{pump} (Frac_{pumptowater})$$

END IF

where:

$\dot{Q}_{heating,total}$ = total water heating capacity including condenser pump heat (W)

P_{pump} = condenser water pump power, user input (W)

$Frac_{pumptowater}$ = fraction of condenser pump heat to water, user input

Compressor power (electricity consumption rate) is then calculated based on two additional inputs provided by the user. The first input specifies if the condenser pump heat is included in the rated heating capacity and rated COP. If the condenser pump heat is included in the

rated heating capacity and COP, then condenser pump power must be subtracted from the DX heating coil power calculated above to determine the compressor power.

The second of these inputs specifies if the evaporator fan power is included in the rated heating COP. If evaporator fan power is included in the rated COP, then fan power must also be subtracted from the DX heating coil power to determine the compressor power as follows:

IF (Evap Fan Power In COP) THEN
IF (Cond Pump Power In COP) THEN

$$P_{comp} = P_{heating} - P_{pump} - P_{fan}$$

ELSE

$$P_{comp} = P_{heating} - P_{fan}$$

END IF

where:

P_{comp} = Compressor power (electric consumption rate) at the current operating conditions (W)

P_{fan} = Fan power, specified in Fan:OnOff object (W)

If fan power is not included in the rated heating COP, then the calculation of compressor power only includes condenser pump power and does not involve the fan:

IF (.NOT. Evap Fan Power In COP) THEN
IF (Cond Pump Power In COP) THEN

$$P_{comp} = P_{heating} - P_{pump}$$

ELSE

$$P_{comp} = P_{heating}$$

END IF

The model assumes that all compressor power is rejected as heat via the DX heating coil. Therefore, the evaporator total cooling capacity at the current operating conditions is determined depending on the user input for pump heat:

IF (Condenser Pump Heat Included In Rated Heating Capacity and Rated COP)

$$\dot{Q}_{evap} = \dot{Q}_{heating} - P_{comp} - (P_{pump} Frac_{pumptowater})$$

ELSE

$$\dot{Q}_{evap} = \dot{Q}_{heating} - P_{comp}$$

END IF

where:

\dot{Q}_{evap} = Total evaporator cooling capacity at the current operating conditions (W)

This evaporator cooling capacity is used to calculate the air-side performance of the heat pump water heater DX coil. The sensible heat ratio of the cooling capacity at rated conditions is specified by the user in the input for this DX coil object. The calculation of sensible heat ratio at off-rated conditions uses the ADP/BF approach described for the DX cooling coil model (Ref. Coil Model – DX Cooling Coil Model). The exiting air conditions for the HPWH DX coil are calculated the same way as they are for the DX cooling coil model (cycling fan, cycling coil). The crankcase heater power and consumption are also calculated using the same equations used for the DX cooling coil model.

The runtime fraction of the DX coil compressor is calculated as the ratio of the compressor part load ratio to the part load fraction correlation entered by the user. The part load ratio of the DX coil is determined by the heat pump water heater compound object (Ref. WaterHeater:HeatPump) and is used by the DX coil to determine the run time fraction of the compressor.

$$RTF = \left(\frac{PLR}{PartLoadFrac} \right) = \text{runtime fraction of the compressor}$$

Finally, the condenser water outlet temperature is calculated based on the total water heating capacity of the DX coil and the actual condenser water mass flow rate.

$$T_{cond,out} = T_{cond,in} + \left(\frac{\dot{Q}_{heating,total}}{\dot{m}_{cond} C_p} \right)$$

where:

$T_{cond,out}$ = condenser outlet water temperature when the DX coil is operating (°C)

$T_{cond,in}$ = condenser inlet water temperature (°C)

C_p = specific heat of water entering the condenser coil (J/kg-C)

\dot{m}_{cond} = actual condenser water mass flow rate when the coil is operating, defined in the WaterHeater:HeatPump parent object (kg/s)

Model Outputs

After completing the calculations for heating capacity and power and the final compressor part load ratio has been determined, the output (report) variables are calculated as follows:

$$DX \text{ Coil Total Water Heating Rate } (W) = \dot{Q}_{heating,total} (PLR)$$

$$DX \text{ Coil Total Water Heating Energy } (J) = \dot{Q}_{heating,total} (PLR)(3600)(t_{sys})$$

$$DX \text{ Coil Water Heating Electric Power } (W) = (P_{comp} + P_{pump})(RTF)$$

$$DX \text{ Coil Water Heating Electric Consumption } (J) = (P_{comp} + P_{pump})(RTF)(3600)(t_{sys})$$

$$DX \text{ Coil Total Cooling Rate } (W) = \dot{Q}_{evap} (PLR)$$

$$DX \text{ Coil Total Cooling Energy } (J) = \dot{Q}_{evap} (PLR)(3600)(t_{sys})$$

$$DX \text{ Coil Sensible Cooling Rate } (W) = \dot{Q}_{evap} (PLR)(SHR)$$

$$DX \text{ Coil Sensible Cooling Energy } (J) = \dot{Q}_{evap} (PLR)(SHR)(3600)(t_{sys})$$

$$DX \text{ Coil Latent Cooling Rate } (W) = \dot{Q}_{evap} (PLR)(1.0 - SHR)$$

$$DX \text{ Coil Latent Cooling Energy } (J) = \dot{Q}_{evap} (PLR)(1.0 - SHR)(3600)(t_{sys})$$

$$DX \text{ Coil Runtime Fraction} = \left(\frac{PLR}{PartLoadFrac} \right)$$

$$DX \text{ Coil Crankcase Heater Power } (W) = (P_{crankcase})(1.0 - RTF)$$

$$DX \text{ Coil Crankcase Heater Consumption } (J) = (P_{crankcase})(1.0 - RTF)(3600)(t_{sys})$$

where:

t_{sys} = HVAC system simulation time step, hr

SHR = sensible heat ratio at the current inlet air conditions and air flow rate
(Ref. Coil Model – DX Cooling Coil Model, ADP/BF approach)

$P_{crankcase}$ = Crankcase heater capacity, user input (W)

Water Source Electric DX Air Cooling Coil

There are two objects for water-to-air heat pumps in cooling mode:
Coil:Cooling:WaterToAirHeatPump:EquationFit and
Coil:Cooling:WaterToAirHeatPump:ParameterEstimation. These are described elsewhere.

Water Source Electric Heat Pump DX Air Heating Coil

There are two objects for water-to-air heat pumps in heating mode:
Coil:Heating:WaterToAirHeatPump:EquationFit and
Coil:Heating:WaterToAirHeatPump:ParameterEstimation. These are described elsewhere.

Steam-Based Air Heating Coil

The steam coils are included in the discussion on steam loops, see: Steam To Air Heat Exchanger.

HVAC Controllers

Control Valve for Water-Based Air System Coils

The input object Controller:WaterCoil provides a simple controller model for adjusting water flow rates to meet the desired air side conditions. It is really a solution inverter. For a water coil the simulation cannot be inverted where the mass flow rate of the water through the coil can be solved directly given an air temperature. Thus, this “controller” will numerically step through all of the water flow possibilities by a root finding technique until the mass flow rate is determined to meet the specified outlet air temperature within a specified user tolerance.

Model Description

The figure below illustrates the use of a simple controller used with a central chilled water coil (control variable Temperature). The controller reads the desired temperature setpoint from the control node (established by a SetpointManager) and modulates the chilled water flow rate at the actuator node in order to meet the desired supply (coil outlet) air temperature.

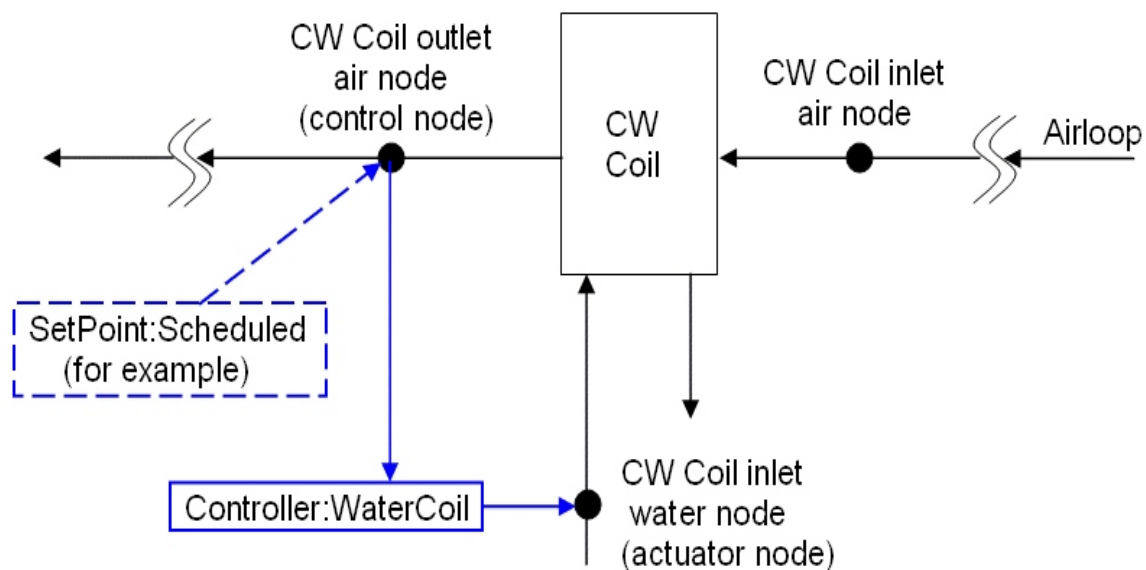


Figure 150. Controller:WaterCoil used with Central Chilled Water Coil

In this case, the controller simply senses the temperature at the control node and compares this value with the desired temperature setpoint. If the sensed temperature is above the desired setpoint temperature, the chilled water flow rate is increased. This controller may be used for both chilled water and hot water coils. The action parameter input is set to REVERSE for chilled water cooling coils and NORMAL for hot water heating coils.

The simple controller may also be used to control both high temperature and high humidity levels by controlling the water flow rate through a chilled water coil. Setting the controller's control variable to TemperatureAndHumidityRatio enables this feature. In this case, the controller monitors two setpoint values, one for temperature control and the other for high humidity control. Note that two setpoint managers must be used to establish these setpoints as shown in the figure below. The limiting case for either temperature or high humidity control (i.e., the minimum supply air temperature required to meet both setpoints) is used for controlling the water flow rate through the chilled water coil. If high humidity control is the limiting case then colder supply air will be delivered by the cooling coil to achieve proper dehumidification and some form of air reheat may be required to avoid overcooling of the zones being served by this air loop.

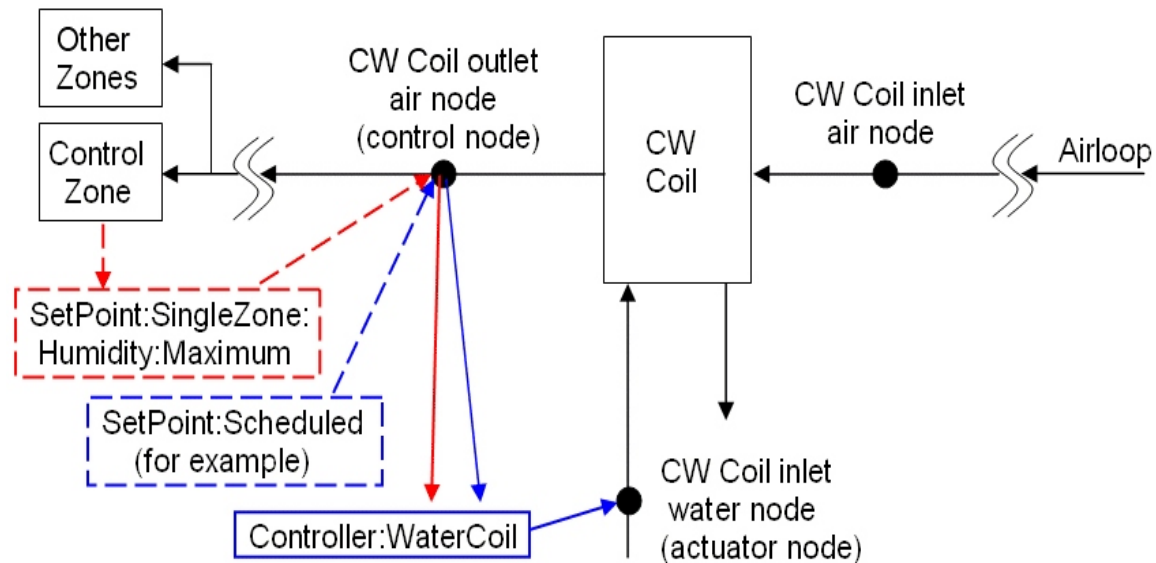


Figure 151. Two Setpoint managers used in Controller:WaterCoil

When the control variable TemperatureAndHumidityRatio is used, the controller modulates water flow through a chilled water coil to meet both a temperature and a humidity ratio setpoint. These two setpoints are placed on the control node by setpoint managers.

The model first calculates the approach temperature using the dry-bulb temperature and dewpoint temperature of the air leaving the water coil:

$$T_{dp} = \text{PsyTdpFnWPb}(\omega_{SA}, P)$$

$$T_{approach} = T_{SA} - T_{dp}$$

where:

$T_{approach}$ = approach temperature, °C

ω_{SA} = supply (outlet) air humidity ratio, kg/kg

P = outdoor barometric pressure, Pa

T_{dp} = supply (outlet) air dewpoint temperature, °C

T_{SA} = supply (outlet) air dry-bulb temperature, °C

PsyTdpFnWPb = EnergyPlus psychrometric function, returns dewpoint temperature given humidity ratio and barometric pressure

The supply air dewpoint temperature required to meet the humidity ratio setpoint (placed on the control node by SetpointManager:SingleZone:Humidity:Maximum) is then calculated as a function of the humidity ratio setpoint and barometric pressure as follows:

$$T_{dp,SP} = \text{PsyTdpFnWPb}(\omega_{SP}, P)$$

where:

$T_{dp,SP}$ = dewpoint temperature corresponding to ω_{SP} , °C

ω_{SP} = humidity ratio setpoint, kg/kg

In order for the dewpoint temperature of the coil's supply (outlet) air to reach $T_{dp,SP}$ the dry-bulb temperature of air leaving the cooling coil must be at $T_{dp,SP} + T_{approach}$:

$$T_{HR,SP} = T_{dp,SP} + T_{approach}$$

where:

$T_{HR,SP}$ = supply air dry-bulb temperature setpoint required to achieve the specified humidity ratio setpoint, °C

The supply air temperature setpoint required to achieve the specified humidity ratio setpoint is then compared to the setpoint temperature required for zone temperature control, and the minimum of these two setpoint values is used as the setpoint temperature for controlling the chilled water coil.

$$T_{SP} = MIN(T_{Temp,SP}, T_{HR,SP})$$

where:

T_{SP} = chilled water coil supply air temperature setpoint, °C

$T_{Temp,SP}$ = supply air temperature setpoint required for zone temperature control, °C

As described previously, the controller varies the chilled water flow rate through the coil using a root finding technique until the actual supply air temperature reaches T_{sp} within the specified tolerance:

$$T_{Actual} - T_{SP} \leq \textit{Controller Convergence Tolerance}$$

where:

T_{Actual} = actual air temperature leaving the cooling coil, °C

Outdoor Air Damper Controller for Air Systems

When the heat exchanger assisted cooling coil is used with a furnace or unitary system (ref. AirLoopHVAC:Unitary:Furnace:HeatCool or AirLoopHVAC:UnitaryHeatCool) or DX system (ref. AirLoopHVAC:UnitaryCoolOnly) located in an air loop (or DX system used in an outside air system), an economizer function may be customized as necessary. For economizer control, an outdoor air controller (ref. Controller:OutdoorAir) is used to define the economizer control inputs and determine when economizer mode is active. The heat exchanger (ref. HeatExchanger:*) object provides an economizer lockout feature which disables heat recovery any time the economizer is active. This feature can be turned on and off using the heat exchanger lockout input. Heat exchanger assisted cooling coils used with the zone equipment described below disregard this economizer control feature.

Inputs

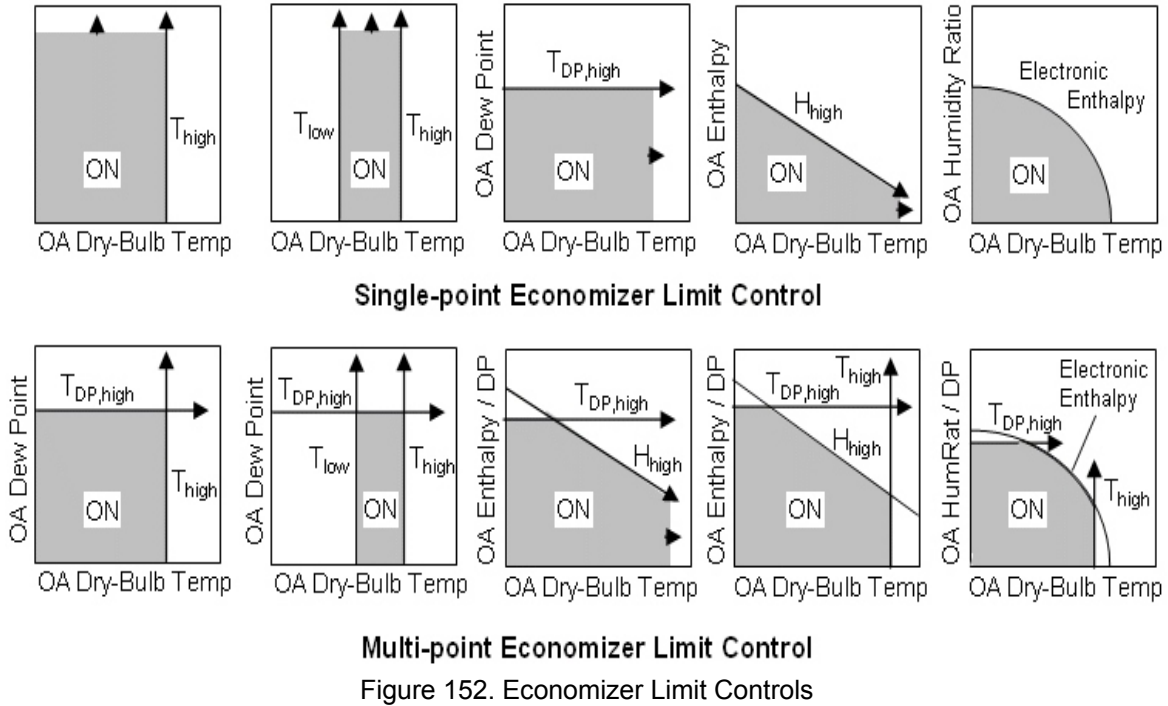
- Controller Name
- Relief air node
- Return air node

- Control node (the mixed air node)
- Actuated node (the outside air inlet node)
- Minimum outside air flow rate (at design) [m³/s]
- Maximum outside air flow rate ($\dot{V}_{oa,max}$) [m³/s]
- EconomizerChoice: *FixedDryBulbr* | *FixedEnthalpy* | *DifferentialDryBulb* | *DifferentialEnthalpy* | *ElectronicEnthalpy* | *FixedDewPointAndDryBulb* | *DifferentialDryBulbAndEnthalpy* | *NoEconomizer*
- Bypasschoice: *MinimumFlowWithBypass* | *ModulateFlow*
- High temperature limit (T_{high}) [C]
- High specific enthalpy limit (h_{high}) [J/kg]
- High dewpoint temperature limit ($T_{DP,high}$) [C]
- Electronic Enthalpy limit (quadratic or cubic curve object)
- Low temperature limit (T_{low}) [C]
- Lockout: *LockoutWithHeating* | *LockoutWithCompressor*
- MinimumLimit: *FixedMinimum* | *ProportionalMinimum*
- Minimum outside air schedule (schedule values multiply the minimum outside air flow rate)
- Minimum Outside air flow Schedule(schedule values sets the limit on minimum outside air)
- Maximum Outside air flow Schedule(schedule values sets the maximum amount of outside air possible in the system)
- Name of Controller:MechanicalVentilation object
- Time-of-day economizer control schedule (economizer active when schedule values are greater than 0)
- High humidity control flag: *Yes* | *No*
- Humidstat control zone name (zone name where humidistat is located)
- Control high indoor humidity based on outdoor humidity ratio: *Yes* | *No*
- High humidity outside air flow ratio (this flow ratio is multiplied by the maximum outside air flow rate during periods of high indoor relative humidity)

Economizer Limits

Economizer limits may be used to create single-point or multi-point controllers. The figure below shows several examples of each type of controller. Single-point economizer controllers use a single independent variable to specify the operation of the economizer. In all cases, the economizer is disabled (OFF) when the outdoor weather condition exceeds the limit. However, when a low temperature limit is used the economizer is disabled when the outdoor temperature is below the low dry-bulb temperature limit.

Economizer limits may also be used to create multi-point controllers where two or more limits specify the economizer operating region. The economizer is disabled (OFF) if any single multi-point economizer limit is exceeded.



Simulation

The simulation contains 4 steps.

Step 1: calculate the minimum outside air flow rate

The user inputs the minimum outside air flow rate (or it is automatically calculated by the program from the user's design inputs). This is converted to a mass flow rate $\dot{m}_{oa,min,des}$ using the density of dry air at 20 degrees C. If the economizer is on an air loop then the minimum outside air flow fraction is initialized to:

$$f_{oa,min} = \dot{m}_{oa,min,des} / \dot{m}_{s,des}$$

Where $\dot{m}_{s,des}$ is the design supply air mass flow rate. If the economizer is not on an air loop the minimum outside air flow fraction is initialized to:

$$f_{oa,min} = \dot{m}_{oa,min,des} / \dot{m}_{oa,max,des}$$

where $\dot{m}_{oa,max,des}$ is the user input maximum outside air volumetric flow rate converted to mass flow rate in the same manner as for the minimum outside air flow rate. If a minimum outside air schedule has been input, $f_{oa,min}$ is multiplied by the current schedule value.

Step 2: calculate an initial outside air signal

We now define an initial outside air signal $S_{oa,init}$.

If $|T_r - T_{in}| > \Delta T_{small}$ then:

$$S_{oa,init} = (T_r - T_{mix,set}) / (T_r - T_i)$$

Otherwise:

if $T_r < T_{mix,set}$ and $T_r \geq T_i$, $S_{oa,init}$ is set to -1;

if $T_r < T_{mix,set}$ and $T_r < T_i$, $S_{oa,init}$ is set to 1;

if $T_r \geq T_{mix,set}$ and $T_r \geq T_i$, $S_{oa,init}$ is set to 1;

if $T_r \geq T_{mix,set}$ and $T_r < T_i$, $S_{oa,init}$ is set to -1.

Finally, $S_{oa,init}$ is constrained to be: $f_{oa,min} \leq S_{oa,init} \leq 1$. Basically, if the outside air can not accomplish cooling, $S_{oa,init}$ is set to $f_{oa,min}$. If it can do cooling, $S_{oa,init}$ is greater than the minimum outside air flow fraction but not greater than 1. Here

T_r is the return air temperature [C];

T_i is the outside air inlet temperature [C];

$T_{mix,set}$ is the mixed air setpoint [C];

ΔT_{small} is a small temperature difference [delta C], equal to .00001.

Step 3: do the on/off and limit checks

- If *EconomizerChoice* = *No Economizer*, $S_{oa,init} = f_{oa,min}$.
- If $\dot{V}_{oa,max} < \dot{V}_{small}$, $S_{oa,init} = f_{oa,min}$; \dot{V}_{small} is .001 m³/s.
- If the economizer is locked out, $S_{oa,init} = f_{oa,min}$.

Note: the above three checks also set the *EconomizerOperationFlag* and *HighHumidityControlFlag* to *false* (economizer and high humidity control not operating).

- If $T_i > T_{mix,set}$, $S_{oa,init} = 1$.
- If Differential dry-bulb was input as Economizer choice and $T_i > T_r$, then $S_{oa,init} = f_{oa,min}$.
- If Differential Enthalpy was input as Economizer choice and $h_i > h_r$, then $S_{oa,init} = f_{oa,min}$, where h_i and h_r are the outside air inlet and return air enthalpies.

Setpoints are checked after this which include check for Fixed dry-bulb temperature limit, Enthalpy Limit, Dewpoint Limit and Humidity ratio limit if specified.

- If Differential Enthalpy was input as Economizer choice and $h_i > h_r$, then $S_{oa,init} = f_{oa,min}$, where h_i and h_r are the outside air inlet and return air enthalpies.

Setpoints are checked after this which include check for Fixed dry-bulb temperature limit, Enthalpy Limit, Dewpoint Limit and Humidity ratio limit if specified.

- If a FixedDryBulb / FixedEnthalpy / FixedDewPointAndDryBulb / ElectronicEnthalpy was input as the economizer choice then the limits are checked as before.
- Limits are checked as follows
 - If a high temperature limit was input and $T_{oa} > T_{high}$, $S_{oa,init} = f_{oa,min}$; where T_{oa} is the outside air temperature.
 - If an enthalpy limit was input and $h_{oa} > h_{high}$, $S_{oa,init} = f_{oa,min}$; where h_{oa} is the outside air specific enthalpy.

- If a dewpoint temperature limit was input and $T_{DP,OA} > T_{DP,high}$, $S_{oa,init} = f_{oa,min}$; where $T_{DP,OA}$ is the outside air dewpoint temperature and $T_{DP,high}$ is dewpoint temperature limit.
- If an electronic enthalpy curve was input and $w_{OA} > w_{CurveOutput}$, $S_{oa,init} = f_{oa,min}$; where w_{OA} is the outside air humidity ratio and $w_{CurveOutput}$ is the curve's humidity ratio output as a function of outdoor air dry-bulb temperature.
- Another Economizer choice called DifferentialDryBulbAndEnthalpy checks the outside air temperature against return temperature and outside air enthalpy against return air enthalpy and also checks the setpoints.
- If a low temperature limit was input and $T_{oa} < T_{low}$, $S_{oa,init} = f_{oa,min}$.

Note: the above nine cases set the *EconomizerOperationFlag* to *false* (economizer not operating), otherwise the economizer is active.

- If high humidity control is specified and the zone humidistat indicates a moisture load (i.e. zone relative humidity greater than the relative humidity setpoint), the *HighHumidityOperationFlag* is set to *true*. If high humidity control is based on the outdoor humidity ratio then the *HighHumidityOperationFlag* is set to *true* only when the outdoor air humidity ratio is less than the humidstat's zone humidity ratio. A *true* *HighHumidityOperationFlag* also enables economizer operation in the heat exchangers as if the economizer flag used here was also set to *true* (Ref. HeatExchanger:* - field Economizer Lockout).

The economizer schedule is then checked to determine if a “push-button” type economizer control is used. When schedule values are greater than 0, the economizer is active (*EconomizerOperationFlag* = *true*). This test overrides the economizer limit checks described above in Step 3.

Step 4: calculate the final outside air signal

If $S_{oa,init}$ is greater than $f_{oa,min}$ and less than 1 and the mixed air mass flow rate is greater than $\dot{m}_{verysmall} (10^{-30})$ and night venting is not occurring and *HighHumidityOperationFlag* is false then

- we calculate a final outside air signal S_{oa} by using the regula falsi method routine *SolveRegulaFalsi* to zero the residual $T_{mix,set} - T_{mix}$ by varying the outside air mass flow rate $S_{oa} \cdot \dot{m}_{mix}$. Mass and energy balance are used to obtain the mixed air humidity ratio and enthalpy from the recirculated air and outside air inlet conditions. The psychrometric function *PsyTdbFnHW* is used to obtain T_{mix} from the mixed air enthalpy and humidity ratio.

Otherwise

- $S_{oa} = S_{oa,init}$.

Step 5: calculate the outside air flow rate and apply final constraints

- If *BypassChoice* = *True* and *HighHumidityOperationFlag* = *false*, $S_{oa} = f_{oa,min}$.
- If the *HighHumidityOperationFlag* is *true* then

$$S_{oa} = \text{MAX} \left[f_{oa,min}, \text{OAF} \text{FlowRatio}_{\text{HighRH}} \left(\frac{\dot{m}_{oa,max,des}}{\dot{m}_{mix}} \right) \right]$$

- If night ventilation is occurring, $S_{oa} = 1$. (Night Ventilation has priority over the above constraints)
- If the Minimum Outside air flow Schedule and the Maximum outside air flow schedule is available, Out Air signal $S_{oa,init}$ is checked against those fractions from the schedule.

$$S_{oa} = \text{Maximum}(\text{Minimum OA fraction}, S_{oa})$$

$$S_{oa} = \text{Minimum}(\text{Maximum OA fraction}, S_{oa})$$

Now the outside air flow rate is calculated:

$$m_{oa} = S_{oa} * m_{mix}$$

Calculate the outside air mass flow rate:

If EconomizerOperationFlag is true and HighHumidityOperationFlag is false then

$$\dot{m}_{oa} = S_{oa} \cdot \dot{m}_{oa,max,des}$$

Otherwise

$$\dot{m}_{oa} = S_{oa} \cdot \dot{m}_{mix}$$

Then the following constraints are applied.

- \dot{m}_{oa} must be greater than or equal to the air primary loop total exhaust air mass flow rate ($\dot{m}_{exhaust}$).
- If *MinimumLimit = Fixed Minimum*, \dot{m}_{oa} must be greater than or equal to the minimum outside air mass flow rate.
- \dot{m}_{oa} must be $\leq \dot{m}_{mix}$.
- \dot{m}_{oa} must be \leq the maximum outside air mass flow rate.

Step 6: set the relief air mass flow rate

$$\dot{m}_{relief} = \max(0, \dot{m}_{oa} - \dot{m}_{exhaust})$$

Outdoor Air Damper Controller for Zone Energy Recovery Ventilator

The stand alone energy recovery ventilator (ERV) controller is used solely in conjunction with a stand alone energy recovery ventilator (see figure below).

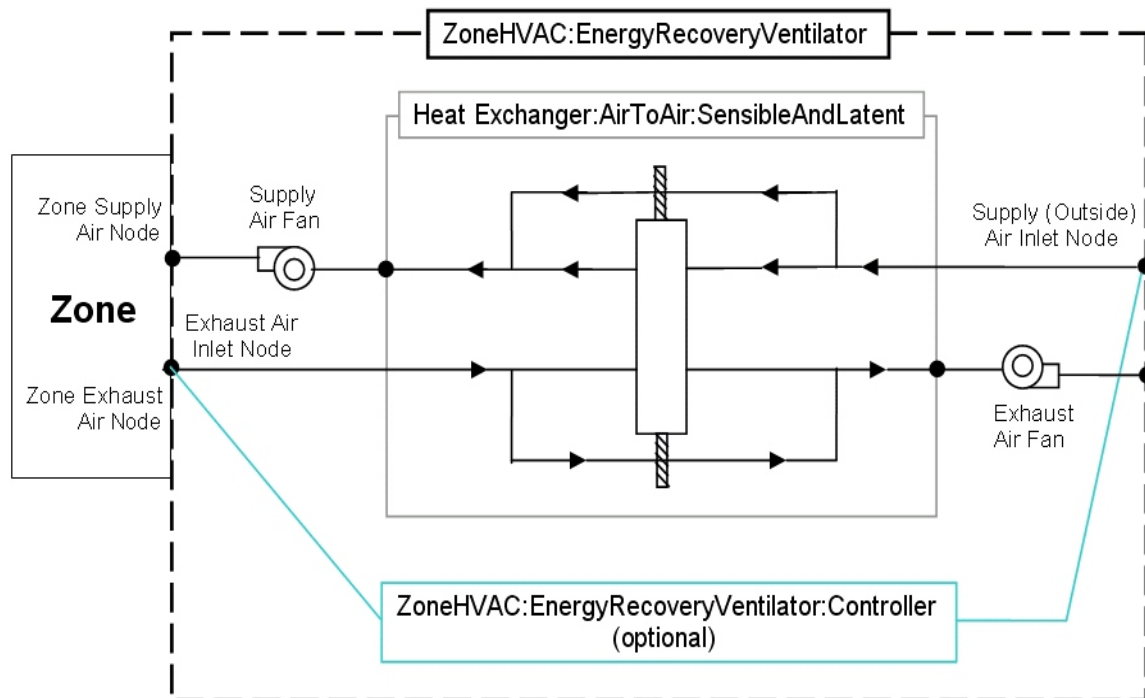


Figure 153. Schematic of the ZoneHVAC:EnergyRecoveryVentilator Compound Object

This controller object mimics virtually all of the control logic available for a conventional air-side economizer as embodied in the object Controller:OutdoorAir. However, this controller is only used with the stand alone energy recovery ventilator object (dedicated to serving a single zone, without a traditional air distribution system) while the Controller:OutdoorAir is used with systems that utilize an air loop to provide conditioned air to one or more zones. The purpose of this controller is to signal the object HeatExchanger:AirToAir:SensibleAndLatent that favorable conditions are available for free cooling and heat exchange should be suspended (i.e., air flow is fully bypassed around a fixed-plate heat exchanger or the rotation of a rotary heat exchanger is stopped). A time-of-day schedule may also be used to simulate a “push-button” type economizer. The air flow rate through the stand alone ERV remains the same regardless of whether the controller is signaling for economizer (free cooling) operation or not. In this way, this controller is very similar to Controller:OutdoorAir with the field Economizer Control Action Type set to “MinimumFlowWithBypass”. However, the supply and exhaust air flow rates may be modified in response to a high indoor humidity condition (i.e., when the zone relative humidity is “predicted” to exceed the setpoint, at times the actual zone relative humidity may be slightly below the setpoint. Ref. Zone/Sys Moisture Load Rate Predicted.) and works in a similar fashion to the outside air controller where the air flow rates are adjusted based on a user specified air flow *ratio*.

Controller Logic

In many ways, the logic for this controller follows that established for the object Controller:OutdoorAir. Nearly the same computations (source code) are used for this controller as for Controller:OutdoorAir, except the addition of a few calculations that are unique for this stand alone ERV controller. Refer to the *Simulation* discussion for the outdoor air controller to review the calculation procedure. In some instances local variables used in the Controller:OutdoorAir computations are set to specific values for ZoneHVAC:EnergyRecoveryVentilator:Controller to allow the same computations and logic to be used for both controllers. The logic that is being applied for ZoneHVAC:EnergyRecoveryVentilator:Controller is presented below.

As explained above the controller senses when air conditions are such that heat exchange by the air-to-air heat exchanger should be suspended to provide free cooling to the zone, thereby reducing the amount of mechanical cooling that needs to be provided by other equipment. The inputs for this controller specify temperature and/or enthalpy conditions that are used to trigger economizer operation. An economizer lockout input is provided in the heat exchanger object and is used for customizing heat exchanger performance during economizer operation. Heat exchange is suspended only if the heat exchanger's economizer lockout input is set to Yes.

The user can enter a high and low temperature limit for economizer operation. When the supply inlet (outdoor) air is between these two values, heat exchange is suspended while air flow rates remain unchanged. This logic represents a conventional single-point temperature economizer control. If the user wishes to model differential temperature control, EXHAUST AIR TEMP LIMIT should be specified in the proper input field. In this case, heat exchange is suspended whenever the temperature of the exhaust air is greater than the temperature of the outdoor air. The user still needs to set the low temperature limit to restart the heat exchange process when the outdoor temperature falls too low.

A high dewpoint temperature limit may also be specified. When the supply inlet (outdoor) air is below this limit, heat exchange is suspended while air flow rates remains unchanged. The user still needs to set the low temperature limit to restart the heat exchange process when the outdoor temperature falls too low.

Similar logic can be used with air enthalpy. The user can enter a high enthalpy limit, and heat exchange between the supply and exhaust air streams will be suspended when the outdoor air enthalpy falls below this value. This logic represents single-point enthalpy economizer control. If the user wishes to model differential enthalpy control, EXHAUST AIR ENTHALPY LIMIT should be specified in the proper input field. Regardless of modeling single-point enthalpy or differential enthalpy control, the user still needs to set the low temperature limit to restart the heat exchange process when the outdoor temperature falls too low.

The user may also specify an electronic enthalpy curve object to represent a variable enthalpy controller. A quadratic or cubic curve is used in this case. The output of this curve would represent the maximum outdoor humidity ratio allowed as a function of outdoor dry-bulb temperature.

The air flow rates through the stand alone ERV may also be modified based on a high indoor relative humidity condition. The change in air flow rates may occur at any time the indoor relative humidity is high or when the indoor relative humidity is high and the outdoor humidity ratio is lower than the indoor humidity ratio. This control mode modifies the air flow rates according to the user specified high humidity outside air flow ratio. When high humidity control is active, heat exchange is suspended in the same manner as if an economizer were used and, as with economizer operation, heat exchange is suspended only if the heat exchanger's economizer lockout input is set to Yes.

The model is flexible, and checks all limits established by the user in the object's input data. The model initially assumes that heat exchange should be suspended, and then checks each one of the limits that the user has set (single-point temperature, differential temperature, single-point enthalpy and differential point enthalpy, single-point dewpoint or electronic (variable) enthalpy). If any of the limits entered by the user is exceeded, then economizer operation is terminated and heat exchange between the supply and exhaust air streams is modeled.

Baseboard Heaters

Hot Water Baseboard Heater with Only Convection

Overview

The convective water baseboard heater (Ref. ZoneHVAC:Baseboard:Convective:Water) is a simple model for establishing a convective-only heat addition to a space through a baseboard

heater. In most situations, the baseboard heater does give a significant amount of heat off via natural convection, but some heat is given off via radiation. In this model, the radiant component is ignored and all heat is assumed to be delivered to the space via convection. The baseboard heater is supplied with hot water from the primary system which is circulated through the inside of a finned tube within the space. Heat is transferred from the water inside the pipe, through the tube and fins, and eventually convected to the surrounding air within the space. EnergyPlus then assumes that this heat is evenly spread throughout the space thus having an immediate impact on the zone air heat balance which is used to calculate the mean air temperature (MAT) within the space.

The model requests an effective UA value to handle the heat exchange between the water loop and the zone air. From there, it employs an effectiveness-NTU heat exchanger model to determine the heat transfer between the water and the zone air. This is necessary because during the simulation only the inlet water and “inlet air” (assumed to be zone air) is known. As a result, the effectiveness-NTU heat exchanger methodology is better suited to determine the performance of the system.

Model Description

Convective Water Baseboard Heater Inputs

Like many other HVAC components, the convective-only water baseboard model requires a unique identifying name, an availability schedule, and water side inlet and outlet nodes. These define the availability of the unit for providing condition to the space and the node connections that relate to the primary system. In addition, a convergence tolerance is requested of the user to help define the ability of the local controls to tightly control the system output. In almost all cases, the user should simply accept the default value for the convergence tolerance unless engaged in an expert study of controls logic in EnergyPlus.

The input also requires a UA value and a maximum design flow rate for the unit. Both of these parameters can be chosen to be auto-sized by EnergyPlus. The UA value corresponds to the convective heat transfer from the water to the tube, the conduction through the tube and fin material, and the natural convection from the tube/fins to the zone air. The maximum flow rate and UA value define the performance and maximum output of the baseboard unit.

Simulation and Control

The simulation of the convective water baseboard unit follows standard effectiveness-NTU methodology. It begins by calculating the product of the specific heat and mass flow rate for both the air and water sides of the unit (heat exchanger). In the initialization of the model, it is assumed that the air mass flow rate is equal to 0.42 times the water mass flow rate. The purpose of this calculation is to provide some reasonable estimate of the air mass flow rate so that the model can calculate system performance correctly. This calculation is kept constant throughout the rest of the simulation.

Each system time step, the baseboard attempts to meet any remaining heating requirement of the zone through the following calculations:

$$C_{water} = c_{p,water} \dot{m}_{water}$$

$$C_{air} = c_{p,air} \dot{m}_{air}$$

$$C_{max} = MAX(C_{air}, C_{water})$$

$$C_{min} = MIN(C_{air}, C_{water})$$

$$C_{ratio} = C_{min} / C_{max}$$

$$\varepsilon = 1 - e^{-\left(\frac{NTU^{0.22} e^{-C_{ratio} NTU^{0.78}}}{C_{ratio}} - 1\right)}$$

Once the effectiveness is determined, the outlet conditions for and output from the unit are determined using the following equations:

$$T_{air,outlet} = T_{air,inlet} + \varepsilon (T_{water,inlet} - T_{air,inlet}) C_{min} / C_{air}$$

$$T_{water,outlet} = T_{water,inlet} - (T_{air,outlet} - T_{air,inlet}) C_{air} / C_{water}$$

$$Output(Convection) = C_{water} (T_{water,inlet} - T_{water,outlet})$$

If the unit was scheduled off or there is no water flow rate through the baseboard unit, then there will be no convection from the unit. The model assumes no heat storage in the baseboard unit itself and thus no residual heat transfer in future system time steps due to heat storage in the water or metal of the baseboard unit.

References

The effectiveness-NTU method is taken from Incropera and DeWitt, Fundamentals of Heat and Mass Transfer, Chapter 11.4, p. 523, eq. 11.33. The user can always refer to the ASHRAE Handbook series for general information on different system types as needed.

Electric Baseboard Heater with Only Convection

Overview

The input object ZoneHVAC:Baseboard:Convective:Electric provides a model for electric baseboard heaters that assumes only convective heat addition to a space from a baseboard heater. In most situations, the baseboard heater does give a significant amount of heat off via natural convection, but some heat is given off via radiation. In this model, the radiant component is ignored and all heat is assumed to be delivered to the space via convection. The baseboard heater is transferred to the unit via electric resistance heating. EnergyPlus then assumes that this heat is evenly spread throughout the space thus having an immediate impact on the zone air heat balance which is used to calculate the mean air temperature (MAT) within the space.

Model Description

Convective Electric Baseboard Heater Inputs

Like many other HVAC components, the convective-only electric baseboard model requires a unique identifying name and an availability schedule. The availability schedule defines the availability of the unit for providing condition to the space.

The input also requires a capacity and efficiency for the unit. While the efficiency is a required input that defaults to unity, the capacity can be chosen to be auto-sized by EnergyPlus.

Simulation and Control

When the unit is available and there is a heating load within a space, the electric baseboard unit will meet the entire remaining provided that it has enough capacity to do so. The energy consumption of the baseboard heat is calculated using the user-supplied efficiency and the current load on the baseboard unit as follows:

$$Energy_{electric} = Heating_{baseboard} / Efficiency$$

If the unit was scheduled off or there is no heating load for the zone, then there will be no convection from the unit. The model assumes no heat storage in the baseboard unit itself and thus no residual heat transfer in future system time steps due to heat storage in the metal of the baseboard unit.

References

No specific reference. Refer to the ASHRAE Handbook series for general information on different system types as needed.

Hot Water Baseboard Heater with Radiation and Convection

Overview

The input object ZoneHVAC:Baseboard:RadiantConvective:Water provides a model for water baseboard heaters that is intended to represent a simple convective baseboard heater model that provides both convective and radiant heat addition to a space through a baseboard heater. In this model, the radiant components are calculated to interact with the building surfaces through the radiant distribution fractions specified by the user. Heat is also delivered to the space via convection. The baseboard heater is supplied with hot water from the primary system which is circulated through the inside of a finned tube within the space. This could also be used to model a hot water radiator (convector in the UK). Heat is transferred from the water inside the pipe, through the tube and fins, and not only convected to the surrounding air but also radiated to the surfaces within the space. The user is allowed to specify the percentage of radiant heat from the heater to the surfaces which the user designated. In addition, the user is allowed the ability to define what fraction of radiation leaving the heater is incident directly on a person within the zone for thermal comfort purposes. This amount of heat is then used in the thermal comfort models as in the high temperature radiant heater model.

The model calculates a UA value using the log mean temperature difference (LMTD) method and is based on information in the I=B=R rating document and overall energy balances to the hot and cold fluids to handle the heat exchange between the water loop and the zone air. Once the UA for the baseboard heat has been determined, the model employs an effectiveness-NTU heat exchanger method to determine the heat transfer between the water and the zone air. This is necessary because during the simulation only the inlet water and "inlet air" (assumed to be zone air) temperatures are known. As a result, the effectiveness-NTU heat exchanger methodology is better suited to determine the performance of the system during the simulation.

Model Description

Water Baseboard Heater Inputs

Like many other HVAC components, the water baseboard model requires a unique identifying name, an availability schedule, and water side inlet and outlet nodes. These define the availability of the unit for providing conditions to the space and the node connections that relate to the primary system. For the calculation of the UA value, standardized information is required. In the I=B=R rating document, the user specifies the same baseboard heater or a similar one and provides the rated average temperature of the water, rated capacity, rated mass flow rate, and standard inlet air temperature. The UA value corresponds to the convective heat transfer from the water to the tube, the conduction through the tube and fin material, the natural convection from the tube/fins to the zone air, and radiation to the surfaces. In addition, a convergence tolerance is requested of the user to help define the ability of the local controls to tightly control the system output. In almost all cases, the user should simply accept the default value for the convergence tolerance unless engaged in an expert study of controls logic in EnergyPlus.

The input also requires a maximum design flow rate for the unit. This parameters can be chosen to be auto-sized by EnergyPlus. The maximum flow rate and the UA value define the performance and maximum output of the baseboard unit.

Many of the inputs for the radiant heat calculation are the same as the high temperature radiant system model. User input of the fraction radiant and of the fraction of radiant energy incident both on people and on surfaces is required to calculate radiant energy distribution from the baseboard heater. The sum of radiant energy of these distribution fractions must sum to unity, and each water baseboard heater is allowed to distribute energy to up to 20 surfaces.

Simulation and Control

The simulation of this water baseboard unit is similar to the water convective only baseboard unit and thus the model follows standard effectiveness-NTU methodology. In initialization, the standard conditions informed by the manufacturer determine the UA value of each baseboard heater. Knowing those parameters, both the standard water inlet and outlet temperature and the standard air temperature are calculated and then the UA value is determined by LMTD Method using the following equations:

$$T_{water,inlet} = \frac{q}{2\dot{m}_{water}c_{p,water}} + T_{water,average}$$

$$T_{water,outlet} = 2T_{water,average} - T_{water,inlet}$$

$$T_{air,outlet} = \frac{q}{\dot{m}_{air}c_{p,air}} + T_{air,inlet}$$

$$\Delta T1 = T_{water,inlet} - T_{air,outlet}$$

$$\Delta T2 = T_{water,outlet} - T_{air,inlet}$$

$$LMTD = \frac{\Delta T1 - \Delta T2}{\log\left(\frac{\Delta T1}{\Delta T2}\right)}$$

$$UA = \frac{LMTD}{q}$$

Once the UA value is obtained, it keeps the same value throughout the simulation for the unit. Also, inlet conditions for both the air and water sides of the unit are calculated in the initializing subroutine.

In this model, the actual system impact on the zone is the sum of the additional convective heat transfer from the surfaces to the zone air after they have been heated by the baseboard unit as well as radiant heat transferred to people and the convective heat transfer to the zone from the baseboard heater. This actual convective power tries to meet any remaining heating requirement in the zone and determines the outlet conditions in each system time step.

The radiant heat simulation in the water baseboard heater is very similar to the high temperature radiant system model. The controls are the same as shown in Figure 207. Also, the surface heat balances are recalculated because of the direct impact on the surfaces within a zone. After all system time steps have been simulated, an “average” zone heat balance calculation is done (similar to the radiant systems). If the unit was scheduled off or there is no water flow rate through the baseboard unit, then, there will be no heat transfer from the unit. The model assumes no heat storage in the baseboard unit itself and thus no

residual heat transfer in future system time steps due to heat storage in the water or metal of the baseboard unit.

References

I=B=R Ratings for Boilers, Baseboard Radiation, Finned Tube (Commercial) Radiation, and Indirect Fired Water Heaters, January 2007 Edition
 Incropera and DeWitt, Fundamentals of Heat and Mass Transfer, Chapter 11.3 and 11.4, eq. 11.15, 11.17, and 11.33.

Cooling Towers

One and Two Speed Cooling Towers

Overview

The input objects CoolingTower:SingleSpeed and CoolingTower:TwoSpeed provide models for single-speed and two-speed cooling towers that are based on Merkel's theory (Merkel 1925), which is also the basis for the tower model included in ASHRAE's HVAC1 Toolkit for primary HVAC system energy calculations (ASHRAE 1999, Bourdouxhe et al. 1994). Cooling tower performance is modeled using effectiveness-NTU relationships for counterflow heat exchangers. The model can be used to simulate the performance of both single speed and two speed mechanical-draft cooling towers. The model will also account for tower performance in the "free convection" regime, when the tower fan is off but the water pump remains on. For part-load operation, the model assumes a simple linear interpolation between two steady-state regimes without accounting for any cycling losses.

For single speed cooling towers, the capacity control can be fan cycling or fluid bypass. In fluid bypass mode, portion of the water goes through the tower media and gets cooled while the remaining water flow gets bypassed, two water flows then mix together trying to meet the tower exiting water setpoint temperature. In both the free convection cooling when fan is off and normal cooling when fan is on for the entire time step, if the tower exiting water temperature is lower than the setpoint, the tower operates in fluid bypass mode. The model determines the fluid bypass fraction by iterations until the mixed water meets the tower exiting water temperature setpoint. In the fluid bypass mode, except the free convection, the tower fan runs at full speed for the entire time step. The maximum amount of tower water that can be bypassed is bounded by the freezing point of the tower water – the tower exiting water temperature cannot be lower than the freezing setpoint.

Model Description

Based on Merkel's theory, the steady-state total heat transfer between the air and water entering the tower can be defined by the following equation:

$$d\dot{Q}_{total} = \frac{UdA}{c_p}(h_s - h_a) \quad (392)$$

where

h_s = enthalpy of saturated air at the wetted-surface temperature, J/kg

h_a = enthalpy of air in the free stream, J/kg

c_p = specific heat of moist air, J/kg-°C

U = cooling tower overall heat transfer coefficient, W/m²-°C

A = heat transfer surface area, m²

Equation (392) is based on several assumptions:

- air and water vapor behave as ideal gases

- the effect of water evaporation is neglected
- fan heat is neglected
- the interfacial air film is assumed to be saturated
- the Lewis number is equal to 1

In this model, it is also assumed that the moist air enthalpy is solely a function of the wet-bulb temperature and that the moist air can be treated as an equivalent ideal gas with its mean specific heat defined by the following equation:

$$\bar{c}_{pe} = \frac{\Delta h}{\Delta T_{wb}} \quad (393)$$

where

Δh = enthalpy difference between the air entering and leaving the tower, J/kg

ΔT_{wb} = wet-bulb temperature difference between the air entering and leaving the tower, °C

Since the liquid side conductance is much greater than the gas side conductance, the wetted-surface temperature is assumed to be equal to the water temperature. Based on this assumption and equations (392) and (393), the expression for total heat transfer becomes:

$$d\dot{Q}_{total} = U_e dA (T_w - T_{wb}) \quad (394)$$

where

$$U_e = \frac{U \bar{c}_{pe}}{c_p}$$

T_{wb} = wet-bulb temperature of the air, °C

T_w = temperature of the water, °C

An energy balance on the water and air sides of the air/water interface yields the following equations:

$$d\dot{Q}_{total} = \dot{m}_w c_{pw} dT_w \quad (395)$$

$$d\dot{Q}_{total} = \dot{m}_a \bar{c}_{pe} dT_{wb} \quad (396)$$

where

\dot{m}_w = mass flow rate of water, kg/s

\dot{m}_a = mass flow rate of air, kg/s

Assuming that the heat capacity rate ($\dot{m}c_p$) for the cooling tower water is less than that for the air, the effectiveness of the cooling tower can be defined by analogy to the effectiveness of a simple heat exchanger:

$$\varepsilon = \frac{T_{win} - T_{wout}}{T_{win} - T_{wbin}} \quad (397)$$

where

ε = heat exchanger effectiveness

T_{win} = inlet water temperature, °C

T_{wout} = outlet water temperature, °C

T_{wbin} = wet-bulb temperature of the inlet air, °C

Combining equations (394), (395), and (396) and integrating over the entire heat transfer surface area, and combining the result with equation (397) provides the following expression for cooling tower effectiveness:

$$\varepsilon = \frac{1 - \exp \left\{ -NTU \left[1 - \left(\frac{\dot{C}_w}{\dot{C}_a} \right) \right] \right\}}{1 - \left(\frac{\dot{C}_w}{\dot{C}_a} \right) \exp \left\{ -NTU \left[1 - \left(\frac{\dot{C}_w}{\dot{C}_a} \right) \right] \right\}} \quad (398)$$

where

$$\dot{C}_w = \dot{m}_w c_{p_w} \quad \text{and} \quad \dot{C}_a = \dot{m}_a \bar{c}_{p_e}$$

$$NTU = \text{Number of Transfer Units} = \frac{UA_e}{\dot{C}_w}$$

This equation is identical to the expression for effectiveness of an indirect contact (i.e., fluids separated by a solid wall) counterflow heat exchanger (Incropera and DeWitt 1981). Therefore, the cooling tower can be modeled, in the steady-state regime, by an equivalent counterflow heat exchanger as shown in the following figure.

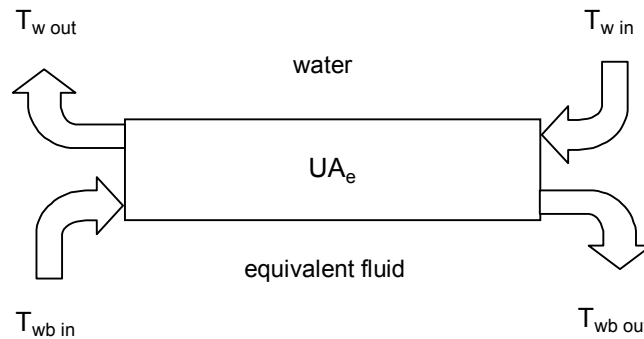


Figure 154. Cooling Tower Schematic

The first fluid is water and the second fluid is an equivalent fluid entering the heat exchanger at temperature T_{wbin} and specific heat \bar{c}_{pe} . The heat exchanger is characterized by a single parameter, its overall heat transfer coefficient-area product UA_e . The actual cooling tower heat transfer coefficient-area product is related to UA_e by the following expression:

$$UA = UA_e \frac{c_p}{c_{pe}} \quad (399)$$

This heat transfer coefficient-area product is assumed to be a function of the air mass flow rate only and can be estimated from laboratory test results or manufacturers' catalog data.

Method for Calculating Steady-State Exiting Water Temperature

The objective of the cooling tower model is to predict the exiting water temperature and the fan power required to meet the exiting water setpoint temperature. Since only the inlet air and inlet water temperatures are known at any simulation time step, an iterative procedure is required to determine the exiting fluid temperatures using the equations defined in the previous section. In the case of the EnergyPlus model, the iterations are performed to determine the exiting wet-bulb temperature of the air. The exiting water temperature is then calculated based on an energy balance that assumes that the energy absorbed by the air is equivalent to the energy removed from the water. The procedure for calculating the steady-state, exiting air wet-bulb temperature is outlined below.

As explained previously, it is assumed that the moist air enthalpy can be defined by the wet-bulb temperature alone. Therefore, the first step in the procedure is to calculate the enthalpy of moist air entering the cooling tower based on the ambient wet-bulb temperature from the weather file. Since an iterative solution is required, a first guess of the outlet air wet-bulb temperature is then made and the enthalpy of this estimated outlet air wet-bulb temperature is calculated. Based on these inlet and outlet air conditions, the mean specific heat of the air is calculated based on equation (393), repeated here:

$$\bar{c}_{pe} = \frac{\Delta h}{\Delta T_{wb}}$$

With the overall heat transfer coefficient-area product for the cooling tower entered by the user, the effective heat transfer coefficient-area product is calculated by rearranging equation (399):

$$UA_e = UA \frac{\bar{c}_{pe}}{c_p}$$

With \bar{c}_{pe} and UA_e known, the effectiveness of the heat exchanger is then calculated:

$$\varepsilon = \frac{1 - \exp\left\{-NTU \left[1 - \left(\frac{\dot{C}_{\min}}{\dot{C}_{\max}}\right)\right]\right\}}{1 - \left(\frac{\dot{C}_{\min}}{\dot{C}_{\max}}\right) \exp\left\{-NTU \left[1 - \left(\frac{\dot{C}_{\min}}{\dot{C}_{\max}}\right)\right]\right\}}$$

where

$$\dot{C}_{\min} = \text{Minimum}(\dot{C}_w, \dot{C}_a) \text{ and } \dot{C}_{\max} = \text{Maximum}(\dot{C}_w, \dot{C}_a)$$

$$\dot{C}_w = \dot{m}_w c_{p_w} \text{ and } \dot{C}_a = \dot{m}_a \bar{c}_{pe}$$

$$NTU = \text{Number of Transfer Units} = \frac{UA_e}{\dot{C}_{\min}}$$

The heat transfer rate is then calculated as follows:

$$\dot{Q}_{total} = \varepsilon \dot{C}_{\min} (T_{win} - T_{wbin})$$

The outlet air wet-bulb temperature is then recalculated:

$$T_{wbout} = T_{wbin} + \frac{\dot{Q}_{total}}{\dot{C}_a}$$

The iterative process of calculating T_{wbout} continues until convergence is reached.

Finally, the outlet water temperature is calculated as follows:

$$T_{wout} = T_{win} + \frac{\dot{Q}_{total}}{\dot{C}_w}$$

Calculating the Actual Exiting Water Temperature and Fan Power

The previous section describes the methodology used for calculating the steady-state temperature of the water leaving the cooling tower. This methodology is used to calculate the exiting water temperature in the free convection regime (water pump on, tower fan off) and with the tower fan operating (including low and high fan speed for the two-speed tower). The exiting water temperature calculations use the fluid flow rates (water and air) and the UA-values entered by the user for each regime.

The cooling tower model seeks to maintain the temperature of the water exiting the cooling tower at (or below) a setpoint. The setpoint schedule is defined by the field “Loop Temperature Setpoint Schedule Name” for the CondenserLoop object. The model first checks to determine the impact of “free convection”, if specified by the user, on the tower exiting water temperature. If free convection is not specified by the user, then the exiting water temperature is initially set equal to the entering tower water temperature. If the user specifies “free convection” and the steady-state exiting water temperature based on “free convection” is at or below the setpoint, then the tower fan is not turned on.

If the exiting water temperature remains above the setpoint after “free convection” is modeled, then the tower fan is turned on to reduce the exiting water temperature to the setpoint. The model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (e.g., tower fan on for the entire simulation time step and tower fan off for the entire simulation time step). Cyclic losses are not taken into account.

The fraction of time that the tower fan must operate is calculated based on the following equation:

$$\omega = \frac{T_{set} - T_{wout,off}}{T_{wout,on} - T_{wout,off}} \quad (400)$$

where

T_{set} = exiting water setpoint temperature, °C

$T_{wout,off}$ = exiting water temperature with tower fan off, °C

$T_{wout,on}$ = exiting water temperature with tower fan on, °C

The average fan power for the simulation time step is calculated by multiplying ω by the steady-state fan power specified by the user.

The calculation method for the two-speed tower is similar to that for the single-speed tower example described above. The model first checks to see if “free convection” is specified and if the resulting exiting water temperature is below the setpoint temperature. If not, then the model calculates the steady-state exiting water temperature with the tower fan at low speed. If the exiting water temperature at low fan speed is below the setpoint temperature, then the

average fan power is calculated based on the result of equation (400) and the steady-state, low speed fan power specified by the user. If low-speed fan operation is unable to reduce the exiting water temperature below the setpoint, then the tower fan is increased to its high speed and the steady-state exiting water temperature is calculated. If this temperature is below the setpoint, then a modified version of equation (400) is used to calculate runtime at high fan speed:

$$\omega = \frac{T_{set} - T_{wout,low}}{T_{wout,high} - T_{wout,low}} \quad (401)$$

where

T_{set} = exiting water setpoint temperature, °C

$T_{wout,low}$ = exiting water temperature with tower fan at low speed, °C

$T_{wout,high}$ = exiting water temperature with tower fan at high speed, °C

The average fan power for the simulation time step is calculated for the two-speed cooling tower as follows:

$$P_{fan,avg} = \omega(P_{fan,high}) + (1 - \omega)(P_{fan,low}) \quad (402)$$

Variable Speed Cooling Towers

Overview

The input object CoolingTower:VariableSpeed provides models for variable speed towers that is based on empirical curve fits of manufacturer's performance data or field measurements. The user specifies tower performance at design conditions, and empirical curves are used to determine the approach temperature and fan power at off-design conditions. The user defines tower performance by entering the inlet air wet-bulb temperature, tower range, and tower approach temperature at the design conditions. The corresponding water flow rate, air flow rate, and fan power must also be specified. The model will account for tower performance in the "free convection" regime, when the tower fan is off but the water pump remains on and heat transfer still occurs (albeit at a low level). Basin heater operation and makeup water usage (due to evaporation, drift, and blowdown) are also modeled.

The cooling tower seeks to maintain the temperature of the water exiting the cooling tower at (or below) a setpoint. The setpoint temperature is defined by the field "Condenser Loop Temperature Setpoint schedule or reference" for the CondenserLoop object. The model simulates the outlet water temperature in four successive steps:

- The model first determines the tower outlet water temperature with the tower fan operating at maximum speed. If the outlet water temperature is above the setpoint temperature, the fan runs at maximum speed.
- If the outlet water temperature with maximum fan speed is below the setpoint temperature, then the model next determines the impact of "free convection" (water flowing through tower with fan off). If the exiting water temperature based on "free convection" is at or below the setpoint, then the tower fan is not turned on.
- If the outlet water temperature remains above the setpoint after "free convection" is modeled, then the tower fan is turned on at the minimum fan speed (minimum air flow rate ratio) to reduce the leaving water temperature. If the outlet water temperature is below the setpoint at minimum fan speed, the tower fan is cycled on and off to maintain the outlet water setpoint temperature.

- If the outlet water temperature remains above the setpoint after minimum fan speed is modeled, then the tower fan is turned on and the model determines the required air flow rate and corresponding fan speed to meet the desired setpoint temperature.

Model Description

The variable speed tower model utilizes user-defined tower performance at design conditions along with empirical curves to determine tower heat rejection and fan power at off-design conditions. Basin heater operation and makeup water usage are also modeled based on user inputs, tower entering air conditions, and tower operation. The following sections describe how each of these tower performance areas is modeled.

Tower Heat Rejection

Heat rejection by the variable speed cooling tower is modeled based on the CoolTools correlation, YorkCalc correlation, or user-defined coefficients for either the CoolTools or YorkCalc correlations. These purely-empirical correlations model the tower approach temperature using a polynomial curve fit with a large number of terms and either three or four independent variables.

The CoolTools correlation has 35 terms with four independent variables:

$$\begin{aligned} \text{Approach} = & \text{Coeff}(1) + \text{Coeff}(2) \cdot \text{FRair} + \text{Coeff}(3) \cdot (\text{FRair})^2 + \\ & \text{Coeff}(4) \cdot (\text{FRair})^3 + \text{Coeff}(5) \cdot \text{FRwater} + \\ & \text{Coeff}(6) \cdot \text{FRair} \cdot \text{FRwater} + \text{Coeff}(7) \cdot (\text{FRair})^2 \cdot \text{FRwater} + \\ & \text{Coeff}(8) \cdot (\text{FRwater})^2 + \text{Coeff}(9) \cdot \text{FRair} \cdot (\text{FRwater})^2 + \\ & \text{Coeff}(10) \cdot (\text{FRwater})^3 + \text{Coeff}(11) \cdot \text{Twb} + \text{Coeff}(12) \cdot \text{FRair} \cdot \text{Twb} + \\ & \text{Coeff}(13) \cdot (\text{FRair})^2 \cdot \text{Twb} + \text{Coeff}(14) \cdot \text{FRwater} \cdot \text{Twb} + \\ & \text{Coeff}(15) \cdot \text{FRair} \cdot \text{FRwater} \cdot \text{Twb} + \text{Coeff}(16) \cdot (\text{FRwater})^2 \cdot \text{Twb} + \\ & \text{Coeff}(17) \cdot (\text{Twb})^2 + \text{Coeff}(18) \cdot \text{FRair} \cdot (\text{Twb})^2 + \\ & \text{Coeff}(19) \cdot \text{FRwater} \cdot (\text{Twb})^2 + \text{Coeff}(20) \cdot (\text{Twb})^3 + \text{Coeff}(21) \cdot \text{Tr} + \\ & \text{Coeff}(22) \cdot \text{FRair} \cdot \text{Tr} + \text{Coeff}(23) \cdot (\text{FRair})^2 \cdot \text{Tr} + \\ & \text{Coeff}(24) \cdot \text{FRwater} \cdot \text{Tr} + \text{Coeff}(25) \cdot \text{FRair} \cdot \text{FRwater} \cdot \text{Tr} + \\ & \text{Coeff}(26) \cdot (\text{FRwater})^2 \cdot \text{Tr} + \text{Coeff}(27) \cdot \text{Twb} \cdot \text{Tr} + \\ & \text{Coeff}(28) \cdot \text{FRair} \cdot \text{Twb} \cdot \text{Tr} + \text{Coeff}(29) \cdot \text{FRwater} \cdot \text{Twb} \cdot \text{Tr} + \\ & \text{Coeff}(30) \cdot (\text{Twb})^2 \cdot \text{Tr} + \text{Coeff}(31) \cdot (\text{Tr})^2 + \text{Coeff}(32) \cdot \text{FRair} \cdot (\text{Tr})^2 + \\ & \text{Coeff}(33) \cdot \text{FRwater} \cdot (\text{Tr})^2 + \text{Coeff}(34) \cdot \text{Twb} \cdot (\text{Tr})^2 + \text{Coeff}(35) \cdot (\text{Tr})^3 \end{aligned}$$

where:

Approach = approach temperature (°C) = outlet water temperature minus inlet air wet-bulb temperature

FRair = air flow rate ratio (actual air flow rate divided by design air flow rate)

FRwater = water flow rate ratio (actual water flow rate divided by design water flow rate)

Tr = range temperature (°C) = inlet water temperature minus outlet water temperature

Twb = inlet air wet-bulb temperature (°C)

Coeff(#) = correlation coefficients

If the user selects Tower Model Type = CoolToolsCrossFlow, then the 35 coefficients derived for the CoolTools simulation model (Benton et al. 2002) are used and these coefficients are already defined within EnergyPlus as shown in Table 44. If the user specifies Tower Model Type = CoolToolsUserDefined, then the user must enter a CoolingTowerPerformance.CoolTools object to define the 35 coefficients that will be used by the CoolTools approach temperature correlation.

Table 44. Approach Temperature Correlation Coefficients

Coefficient Number	Coefficient Value	
	CoolTools	YorkCalc
Coeff(1)	0.52049709836241	-0.359741205
Coeff(2)	-10.617046395344	-0.055053608
Coeff(3)	10.7292974722538	0.0023850432
Coeff(4)	-2.74988377158227	0.173926877
Coeff(5)	4.73629943913743	-0.0248473764
Coeff(6)	-8.25759700874711	0.00048430224
Coeff(7)	1.57640938114136	-0.005589849456
Coeff(8)	6.51119643791324	0.0005770079712
Coeff(9)	1.50433525206692	-0.00001342427256
Coeff(10)	-3.2888529287801	2.84765801111111
Coeff(11)	0.0257786145353773	-0.121765149
Coeff(12)	0.182464289315254	0.0014599242
Coeff(13)	-0.0818947291400898	1.680428651
Coeff(14)	-0.215010003996285	-0.0166920786
Coeff(15)	0.0186741309635284	-0.0007190532
Coeff(16)	0.0536824177590012	-0.025485194448
Coeff(17)	-0.00270968955115031	0.0000487491696
Coeff(18)	0.00112277498589279	0.00002719234152
Coeff(19)	-0.00127758497497718	-0.0653766255555556
Coeff(20)	0.0000760420796601607	-0.002278167
Coeff(21)	1.43600088336017	0.0002500254
Coeff(22)	-0.5198695909109	-0.0910565458
Coeff(23)	0.117339576910507	0.00318176316
Coeff(24)	1.50492810819924	0.000038621772
Coeff(25)	-0.135898905926974	-0.0034285382352
Coeff(26)	-0.152577581866506	0.00000856589904
Coeff(27)	-0.0533843828114562	-0.000001516821552
Coeff(28)	0.00493294869565511	N/A
Coeff(29)	-0.00796260394174197	N/A
Coeff(30)	0.000222619828621544	N/A
Coeff(31)	-0.0543952001568055	N/A
Coeff(32)	0.00474266879161693	N/A
Coeff(33)	-0.0185854671815598	N/A
Coeff(34)	0.00115667701293848	N/A

Coeff(35)	0.000807370664460284	N/A
-----------	----------------------	-----

Similarly, the YorkCalc correlation has 27 terms with three independent variables:

$$\begin{aligned} \text{Approach} = & \text{Coeff}(1) + \text{Coeff}(2) \cdot \text{Twb} + \text{Coeff}(3) \cdot \text{Twb}^2 + \text{Coeff}(4) \cdot \text{Tr} + \\ & \text{Coeff}(5) \cdot \text{Twb} \cdot \text{Tr} + \text{Coeff}(6) \cdot \text{Twb}^2 \cdot \text{Tr} + \text{Coeff}(7) \cdot \text{Tr}^2 + \\ & \text{Coeff}(8) \cdot \text{Twb} \cdot \text{Tr}^2 + \text{Coeff}(9) \cdot \text{Twb}^2 \cdot \text{Tr}^2 + \text{Coeff}(10) \cdot \text{LGRatio} + \\ & \text{Coeff}(11) \cdot \text{Twb} \cdot \text{LGRatio} + \text{Coeff}(12) \cdot \text{Twb}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(13) \cdot \text{Tr} \cdot \text{LGRatio} + \text{Coeff}(14) \cdot \text{Twb} \cdot \text{Tr} \cdot \text{LGRatio} + \\ & \text{Coeff}(15) \cdot \text{Twb}^2 \cdot \text{Tr} \cdot \text{LGRatio} + \text{Coeff}(16) \cdot \text{Tr}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(17) \cdot \text{Twb} \cdot \text{Tr}^2 \cdot \text{LGRatio} + \text{Coeff}(18) \cdot \text{Twb}^2 \cdot \text{Tr}^2 \cdot \text{LGRatio} + \\ & \text{Coeff}(19) \cdot \text{LGRatio}^2 + \text{Coeff}(20) \cdot \text{Twb} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(21) \cdot \text{Twb}^2 \cdot \text{LGRatio}^2 + \text{Coeff}(22) \cdot \text{Tr} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(23) \cdot \text{Twb} \cdot \text{Tr} \cdot \text{LGRatio}^2 + \text{Coeff}(24) \cdot \text{Twb}^2 \cdot \text{Tr} \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(25) \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 + \text{Coeff}(26) \cdot \text{Twb} \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 + \\ & \text{Coeff}(27) \cdot \text{Twb}^2 \cdot \text{Tr}^2 \cdot \text{LGRatio}^2 \end{aligned}$$

where:

Approach = approach temperature (°C) = outlet water temperature minus inlet air wet-bulb temperature

Tr = range temperature (°C) = inlet water temperature minus outlet water temperature

Twb = inlet air wet-bulb temperature (°C)

LGRatio = liquid-to-gas ratio = ratio of water flow rate ratio (FR_{water}) to air flow rate ratio (FR_{air})

Coeff(#) = correlation coefficients

If the user selects Tower Model Type = YorkCalc, then the 27 coefficients derived for the YorkCalc simulation model (York International Corp. 2002) are used and these coefficients are already defined within EnergyPlus as shown in Table 44. If the user specifies Tower Model Type = YorkCalcUserDefined, then the user must enter a CoolingTowerPerformance:YorkCalc object to define the 27 coefficients that will be used by the YorkCalc approach temperature correlation.

The approach temperature correlations for the CoolTools and YorkCalc simulation models are valid for a range of conditions defined in Table 45. If the user defines their own model coefficients (CoolingTowerPerformance:CoolTools or CoolingTowerPerformance:YorkCalc), then they must also define in that same object the range of conditions for which the model is valid. For all of these correlation variables, the program issues warnings if the actual values are beyond the minimum/maximum values specified for the correlation being used. For inlet air wet-bulb temperature and water mass flow rate ratio, the values of these variables used in the calculation of approach temperature are limited to be within the valid minimum/maximum range. For approach, range, and liquid-to-gas ratio the warnings are issued if the values are beyond the specified minimum/maximum range but the actual values are still used. The warnings issued do not necessarily indicate a poor estimate of tower performance at the condition(s) which caused the warning, but are provided to identify conditions outside the defined correlation limits. Exceeding the defined limits by a small amount may not introduce significant errors, but large deviations may be problematic. It is for this reason that we recommend using a very broad range of cooling tower performance data (i.e., data covering the entire range expected during the simulation) when generating user-defined coefficients for the variable speed tower model.

Table 45. Minimum and Maximum Limits for Approach Temperature Correlation Variables

Independent Variable Limit	CoolTools	YorkCalc
Minimum Inlet Air Wet-Bulb Temperature	-1.0°C	-34.4°C
Maximum Inlet Air Wet-Bulb Temperature	26.7°C	26.7°C
Minimum Tower Range Temperature	1.1°C	1.1°C
Maximum Tower Range Temperature	11.1°C	22.2°C
Minimum Tower Approach Temperature	1.1°C	1.1°C
Maximum Tower Approach Temperature	11.1°C	40°C
Minimum Water Flow Rate Ratio	0.75	0.75
Maximum Water Flow Rate Ratio	1.25	1.25
Maximum Liquid-to-Gas Ratio	N/A	8.0

The approach temperature correlation(s) used to simulate cooling tower heat rejection are based on water and air flow rate “ratios” and are not directly dependent on the size of the tower or the actual air and water flow rates through the tower. However, the model correlations are developed based on a reference condition. For Model Types “CoolToolsCrossFlow” and “YorkCalc”, the reference condition is a water flow rate of 0.000043 m³/s per kW of heat rejected (2.4 gal/min per ton of heat rejected) with 25.6°C (78°F) enter air wet-bulb temperature, 35°C (95°F) hot water inlet temperature, and 29.4°C (85°F) cold water outlet temperature. The reference condition may be different if the user defines tower model coefficients using CoolingTowerPerformance:CoolTools or CoolingTowerPerformance:YorkCalc.

Due to the inherent reference condition used to generate the tower performance curves, the water flow rate at the reference condition must be determined using the design performance information specified by the user and the tower model’s approach temperature correlation. This is done by using the model’s approach temperature correlation (described earlier in this section) to calculate the water flow rate ratio which yields the user-defined design approach temperature based on an air flow rate ratio of 1.0 ($FR_{air} = 1.0$), the design inlet air wet-bulb temperature, and the design range temperature. The calculated approach temperature (using the model correlation) must satisfy the following two equations:

$$T_{water, outlet, design} = T_{wb, air, design} + T_{approach, design} \{FR_{air} = 1.0\}$$

$$T_{water, inlet, design} = T_{wb, air, design} + T_{approach, design} + T_{range, design} \{FR_{air} = 1.0\}$$

where:

$$T_{water, outlet, design} = \text{design outlet water temperature (}^{\circ}\text{C)}$$

$$T_{water, inlet, design} = \text{design inlet water temperature (}^{\circ}\text{C)}$$

$$T_{wb, air, design} = \text{design inlet air wet-bulb temperature (}^{\circ}\text{C)}$$

$$T_{approach, design} = \text{design approach temperature (}^{\circ}\text{C)}$$

$$T_{range, design} = \text{design range temperature (}^{\circ}\text{C)}$$

FR_{air} = air flow rate ratio (actual air flow rate divided by design air flow rate)

The water flow rate ratio used in the approach temperature correlation which satisfies these two equations is the ratio of the design water flow rate (specified by the user) to the water flow rate at the reference condition. This ratio is used to calculate the reference water volumetric flow rate, which is then used throughout the simulation to determine the actual water flow rate ratio used in the approach temperature correlation for each simulation time step.

$$\dot{V}_{reference} = \frac{\dot{V}_{design}}{FR_{water,reference}}$$

where:

$\dot{V}_{reference}$ = water volumetric flow rate at the reference condition (m³/s)

\dot{V}_{design} = design water volumetric flow rate specified by the user (m³/s)

$FR_{water,reference}$ = design water flow rate divided by the reference water flow rate

The cooling tower seeks to maintain the temperature of the water exiting the cooling tower at (or below) a setpoint. The setpoint temperature is defined by the field “Condenser Loop Temperature Setpoint schedule or reference” for the CondenserLoop object. The model simulates the outlet water temperature in four successive steps:

- The model first determines the tower outlet water temperature with the tower fan operating at maximum speed. If the outlet water temperature is above the setpoint temperature, the fan runs at maximum speed.
- If the outlet water temperature with maximum fan speed is below the setpoint temperature, then the model next determines the impact of “free convection” (water flowing through tower with fan off). If the exiting water temperature based on “free convection” is at or below the setpoint, then the tower fan is not turned on.
- If the outlet water temperature remains above the setpoint after “free convection” is modeled, then the tower fan is turned on at the minimum fan speed (minimum air flow rate ratio) to reduce the leaving water temperature. If the outlet water temperature is below the setpoint at minimum fan speed, the tower fan is cycled on and off to maintain the outlet water setpoint temperature.
- If the outlet water temperature remains above the setpoint after minimum fan speed is modeled, then the tower fan is turned on and the model determines the required air flow rate and corresponding fan speed to meet the desired setpoint temperature.

For each simulation time step, the model first calculates the outlet water temperature with the tower fan operating at maximum speed ($FR_{air} = 1.0$). The calculated approach temperature (using the correlations described above), inlet air wet-bulb temperature (weather data), and range temperature are used to determine the tower outlet water temperature as follows:

$$\begin{aligned} T_{water,outlet,fanMAX} &= T_{wb,air} + T_{approach} \quad \{FR_{air} = 1.0\} \\ &= T_{water,inlet} + T_{range} \end{aligned}$$

where:

$T_{water,outlet,fanMAX}$ = tower outlet water temperature at maximum fan speed (°C)

$T_{wb,air}$ = tower inlet air wet-bulb temperature (°C)

$T_{approach}$ = approach temperature at current operating conditions (°C)

T_{range} = range temperature at current operating conditions (°C)

Note that the approach temperature correlation as described previously is a function of range temperature, so the equations above must be solved iteratively to converge on a solution. If the resulting outlet water temperature is above the desired setpoint temperature, then the fan runs at maximum speed and does not cycle on/off (fan part-load ratio = FanPLR = 1.0 and $FR_{air} = 1.0$).

If the outlet water temperature with maximum fan speed is below the setpoint temperature, then the model next determines the impact of “free convection” (water flowing through tower with fan off). In the free convection regime, the outlet water temperature is calculated using a fraction of the water temperature difference through the tower when the fan is at its maximum speed. This fraction is defined by the user (Fraction of Tower Capacity in Free Convection Regime).

$$T_{water,outlet,fanOFF} = T_{water,inlet} - \left[Frac_{freeconv} \left(T_{water,inlet} - T_{water,outlet,fanMAX} \right) \right]$$

where:

$T_{water,outlet,fanOFF}$ = tower outlet water temperature in free convection regime (°C)

$T_{water,inlet}$ = tower inlet water temperature (°C)

$Frac_{freeconv}$ = fraction of tower capacity in free convection regime (user specified)

If the outlet water temperature in the free convection regime is below the setpoint temperature, the tower fan is not turned on and the fan part-load ratio is set equal to 0. In addition, the air flow rate ratio through the tower is assumed to be equal to the fraction of tower capacity in the free convection regime.

$$FanPLR = FanPLR_{freeconv} = 0.0$$

$$FR_{air} = FR_{air,freeconv} = Frac_{freeconv}$$

where:

$FanPLR$ = fan part-load ratio

$FanPLR_{freeconv}$ = fan part-load ratio in free convection regime

$FR_{air,freeconv}$ = air flow rate ratio in free convection regime

If the outlet water temperature in the free convection regime is above the setpoint temperature, then the fan is turned on at the minimum fan speed (minimum air flow rate ratio, $FR_{air,min}$, entered by the user) and the outlet water temperature is calculated as the inlet air wet-bulb temperature plus the calculated approach temperature:

$$FR_{air} = FR_{air,min}$$

$$T_{water,outlet,FanMIN} = T_{wb,air} + T_{approach} \quad \{FR_{air} = FR_{air,min}\}$$

where:

$T_{water,outlet,FanMIN}$ = outlet water temperature at minimum fan speed (°C)

$FR_{air,min}$ = air flow rate ratio at the minimum fan speed

If the outlet water temperature at minimum fan speed is below the setpoint temperature, the cooling tower fan cycles on and off at the minimum air flow rate ratio in order to meet the setpoint temperature.

$$FanPLR = \left(\frac{T_{water,outlet,fanOFF} - T_{setpoint}}{T_{water,outlet,fanOFF} - T_{water,outlet,fanMIN}} \right)$$

where:

$$T_{setpoint} = \text{outlet water setpoint temperature (}^{\circ}\text{C)}$$

If the outlet water temperature at minimum fan speed is above the outlet water temperature setpoint, then the cooling tower fan speed (FR_{air}) is increased until the calculated approach temperature produces the required outlet water temperature to meet the setpoint.

$$T_{approach} = f(FR_{air})$$

$$T_{water,outlet} = T_{setpoint} = T_{wb,air} + T_{approach}$$

$$FanPLR = 1.0 \text{ (i.e., fan does not cycle on/off)}$$

Fan Power

When the cooling tower fan is operating, fan electric power is calculated based on the air flow rate ratio required to meet the above conditions. If the user has entered a fan power curve object (cubic curve), the output of that curve is multiplied by the design fan power. Otherwise, tower fan power is assumed to be directly proportional to the cube of the air flow rate ratio. In either case, the fan part-load ratio is applied to account for times when the tower fan cycles on/off to meet the setpoint temperature. Fan energy consumption is calculated each simulation time step.

IF (FanPowerCurveObject is Defined) THEN

$$P_{fan} = (CurveOutput)(FanPLR)P_{fan,design}$$

ELSE

$$P_{fan} = (P_{fan,design})(FanPLR)(FR_{air})^3$$

ENDIF

$$E_{fan} = (P_{fan})(TimeStepSys)3600.$$

where:

FanPowerCurveObject = name of fan power ratio as a function of air flow rate ratio curve

P_{fan} = tower fan electric power (W)

E_{fan} = tower fan electric consumption (J)

CurveOutput = output of FanPowerCurveObject evaluated at the operating air flow rate ratio (FR_{air})

$P_{fan,design}$ = design fan power at design (maximum) air flow through the tower (W)

TimeStepSys = HVAC system simulation time step (hr)

Tower Basin Heater

Calculations are also made to estimate the electric power input to the tower basin heater. A schedule may be used to disable the basin heater during regular maintenance periods or other time periods (e.g., during summer). If a schedule is not provided, the basin heater is assumed to be available the entire simulation time period. The basin heater operates when it is scheduled on, the outdoor air dry-bulb temperature is below the basin heater setpoint temperature, and the cooling tower is not active (i.e., water is not flowing through the tower). The user is required to enter a basin heater capacity (watts per degree Kelvin) and a heater setpoint temperature (°C) if they want to model basin heater electric power.

$$P_{\text{heater,basin}} = 0.0$$

IF (WaterNotFlowingThroughTower) THEN

IF (Schedule_{heater,basin} is Defined) THEN

IF (CAP_{heater,basin} > 0 .AND. Schedule_{heater,basin} = ON) THEN

$$P_{\text{heater,basin}} = \text{MAX} \left(0.0, \text{CAP}_{\text{heater,basin}} \left(T_{\text{setpoint,basin}} - T_{\text{db,outdoor}} \right) \right)$$

ENDIF

ELSE

IF (CAP_{heater,basin} > 0) THEN

$$P_{\text{heater,basin}} = \text{MAX} \left(0.0, \text{CAP}_{\text{heater,basin}} \left(T_{\text{setpoint,basin}} - T_{\text{db,outdoor}} \right) \right)$$

ENDIF

ENDIF

ENDIF

$$E_{\text{heater,basin}} = \left(P_{\text{heater,basin}} \right) (\text{TimeStepSys}) 3600.$$

where:

$P_{\text{heater,basin}}$ = tower basin heater electric power (W)

$E_{\text{heater,basin}}$ = tower basin heater electric consumption (J)

$T_{\text{setpoint,basin}}$ = basin heater setpoint temperature (°C)

$T_{\text{db,outdoor}}$ = outdoor air dry-bulb temperature (°C)

$\text{CAP}_{\text{heater,basin}}$ = basin heater capacity (W/K)

$\text{Schedule}_{\text{heater,basin}}$ = basin heater schedule (schedule value > 0 means ON)

References

ASHRAE 1999. HVAC1 Toolkit: A Toolkit for Primary HVAC System Energy Calculations. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Benton, D.J., Bowman, C.F., Hydeman, M., Miller, P. 2002. An Improved Cooling Tower Algorithm for the CoolTools™ Simulation Model. *ASHRAE Transactions*, Vol. 108, Part 1, pp.760-768.

Bourdouxhe, J.P., M. Grodent, J. Lebrun and C. Silva. 1994. Cooling tower model developed in a toolkit for primary HVAC system energy calculation: part 1. Proceedings of the fourth

international conference on system simulation in buildings, Liege (Belgium), December 5-7, 1994.

Incropera, F.P. and D.P. DeWitt. 1981. Fundamentals of Heat Transfer. New York: John Wiley & Sons.

Merkel, F. 1925. Verdüftungskühlung. VDI Forschungsarbeiten, No 275, Berlin.

York International Corporation, 2002. "YORKcalc™ Software, Chiller-Plant Energy-Estimating Program", Form 160.00-SG2 (0502).

Cooling Tower Makeup Water Usage

Makeup water use for all types of cooling towers is made up of three components: evaporation, drift, and blowdown. The first is the amount of water evaporated to reduce the water's temperature as it passes through the cooling tower. There are two methods that evaporation makeup water can be modeled in EnergyPlus. The first method assumes that the tower outlet air conditions are saturated (which may not always be the case for certain operating conditions). For this "Saturated Exit" mode, the enthalpy of the tower's outlet air is calculated as the inlet air enthalpy plus the water side heat transfer divided by the air mass flow rate through the tower.

$$\dot{Q}_{water} = \dot{m}_{water} C_p (T_{water,inlet} - T_{water,outlet})$$

$$h_{sat\ air,outlet} = h_{air,inlet} + \left(\frac{\dot{Q}_{water}}{\dot{m}_{air}} \right)$$

where:

\dot{Q}_{water} = water-side heat transfer (W)

\dot{m}_{water} = mass flow rate of water through the tower (kg/s)

C_p = specific heat of water (W/kg-K)

$h_{sat\ air,outlet}$ = saturated outlet air enthalpy (J/kg)

$h_{air,inlet}$ = inlet air enthalpy (J/kg)

\dot{m}_{air} = mass flow rate of air through the tower (kg/s)

The saturation temperature and humidity ratio are then calculated for the tower's outlet air.

$$T_{sat\ air,outlet} = P_{sy}T_{satFnHPb}(h_{sat\ air,outlet}, OutBaroPress)$$

$$\omega_{sat\ air,outlet} = P_{sy}WFnTdbH(T_{sat\ air,outlet}, h_{sat\ air,outlet})$$

where:

$T_{sat\ air,outlet}$ = saturated outlet air temperature (°C)

$P_{sy}T_{satFnHPb}$ = EnergyPlus psychrometric function, returns saturation temperature given enthalpy and barometric pressure

OutBaroPress = outdoor barometric pressure (Pa)

$\omega_{sat\ air,outlet}$ = saturated outlet air humidity ratio (kg_{water}/kg_{dry air})

$PsyWFnTdbH$ = EnergyPlus psychrometric function, returns humidity ratio given dry-bulb temperature and enthalpy

The makeup water quantity required to replenish the water lost due to evaporation is then calculated as the product of the air mass flow rate and the difference between the entering and leaving air humidity ratio divided by the density of water.

$$\dot{V}_{evaporation,makeup} = \frac{\dot{m}_{air} (\omega_{sat,air,outlet} - \omega_{air,inlet})}{\rho_{water}}$$

where:

$\dot{V}_{evaporation,makeup}$ = makeup water usage due to evaporation (m³/s)

\dot{m}_{air} = mass flow rate of air through tower (kg/s)

$\omega_{air,inlet}$ = humidity ratio of tower inlet air (kg_{water}/kg_{dry air})

ρ_{water} = density of water evaluated at the tower inlet air temperature (kg/m³)

The second method available for calculating water makeup for evaporation is for the user to provide a value for a loss factor. The evaporation loss is then calculated as a fraction of the circulating condenser water flow and varies with the temperature change in the condenser water. The value provided by the user is in units of percent-per-degree Kelvin. The evaporation rate will equal this value times each degree Kelvin of temperature drop in the condenser water. Typical values are from 0.15 to 0.27 [percent/K]. The default is 0.2. The rate of water makeup for evaporation is then calculated by multiplying this factor times the condenser water flow rate and the temperature decrease in the condenser water flow rate.

Additional makeup water usage is modeled as a percentage of design water flow rate through the tower to account for drift, and as a scheduled flow rate to model blowdown. Drift is water loss due to the entrainment of small water droplets in the air stream passing through the tower. Drift is defined by the model user as a percentage of the tower's design water flow rate, and is assumed to vary with tower air flow rate ratio as follows:

$$\dot{V}_{drift,makeup} = \dot{m}_{water,design} \left(\frac{PercentWatertoDrift}{100.0} \right) FR_{air}$$

where:

$\dot{V}_{drift,makeup}$ = makeup water usage due to drift (m³/s)

$\dot{m}_{water,design}$ = design (volumetric) water flow rate (m³/s)

$PercentWatertoDrift$ = percent of design water flow rate lost to drift at the tower design air flow rate

FR_{air} = ratio of actual air flow rate to tower design air flow rate

Blowdown is water flushed from the basin on a periodic basis to purge the concentration of mineral scale or other contaminants. There are two ways that blowdown is calculated in EnergyPlus. Blowdown water rates can be scheduled so that we have..

$IF (Schedule_{blowdown} \text{ is defined}) THEN$
 $\dot{V}_{blowdown,makeup} = ScheduleValue_{blowdown}$
 $ELSE$
 $\dot{V}_{blowdown,makeup} = 0.0$
 $ENDIF$

where:

$\dot{V}_{blowdown,makeup}$ = makeup water usage due to blowdown (m³/s)

$ScheduleValue_{blowdown}$ = blowdown schedule value for the time step being simulated (m³/s)

The second (and default) way that blowdown can be calculated is to assume that blowdown water is continually introduced at a rate that will provide a constant concentration ratio. As water evaporates it leaves behind minerals and the like causing the concentration of water impurities to be higher in the tower than in the makeup water. Acceptable concentration ratios are in the range of 3 to 5 depending on the purity of the make up water. Water lost as drift does not evaporate and decrease the water needed for blowdown. Using the "Concentration Ratio" method, the rate of blowdown can be calculated using:

$$\dot{V}_{blowdown,makeup} = \frac{\dot{V}_{evaporation,makeup}}{R_{concentration} - 1} - \dot{V}_{drift,makeup}$$

where,

$R_{concentration}$ is the concentration ratio or the ratio of solids in the blowdown water to solids in the makeup water.

The tower makeup water consumption (m³) for each simulation time step is calculated as the sum of the individual components of makeup water usage multiplied by the simulation time step in hours and the conversion for hours to seconds (3600 sec/hr). Makeup water usage is only calculated when the cooling tower is active and water is flowing through the cooling tower.

$$V_{water,makeup} = (\dot{V}_{evaporation,makeup} + \dot{V}_{drift,makeup} + \dot{V}_{blowdown,makeup}) \cdot TimeStepSys \cdot 3600.$$

where:

$V_{water,makeup}$ = tower makeup water consumption (m³)

References

Robert C. Rosaler. 1995. Standard Handbook of Plant Engineering, 2nd Ed. McGraw-Hill, New York, NY, pp 6-36-37.

One and Two Speed Fluidcoolers

Overview

The input objects Fluidcooler:SingleSpeed and Fluidcooler:TwoSpeed provide models for fluidcoolers that are similar to the cooling tower model with the difference that the 'nominal capacity' method is used instead of the 'UA and design water flow rate' method. Fluid Cooler performance is modeled using effectiveness-NTU relationships for cross flow heat exchanger with both streams unmixed. The model can be used to simulate the performance of both

single speed and two speed mechanical-draft fluid coolers. For part-load operation, the model assumes a simple linear interpolation between two steady-state regimes without accounting for any cycling losses.

Model Description

The expression for Fluid Cooler effectiveness is as follows:

$$\varepsilon = 1 - \exp\left(\frac{e^{-NTU \cdot Z \cdot \eta} - 1}{Z \cdot \eta}\right) \quad (403)$$

Where

ε = heat exchanger effectiveness

$$\eta = NTU^{-0.22}$$

$$Z = \dot{C}_{\min} / \dot{C}_{\max}$$

$$\dot{C}_w = \dot{m}_w c_{pw} \text{ and } \dot{C}_a = \dot{m}_a \bar{c}_{pe}$$

$$\dot{C}_{\min} = \text{Minimum}(\dot{C}_w, \dot{C}_a) \text{ and } \dot{C}_{\max} = \text{Maximum}(\dot{C}_w, \dot{C}_a)$$

$$NTU = \text{Number of Transfer Units} = \frac{UA_e}{\dot{C}_{\min}}$$

The first fluid is water and the second fluid is air entering the heat exchanger at temperature T_{dbin} and specific heat \bar{c}_{pe} . The heat exchanger is characterized by a single parameter, its overall heat transfer coefficient-area product UA_e . If the user selects the nominal capacity method, the UA is calculated as follows:

We specify the model inputs (other than the UA) at design conditions and the design Fluid Cooler load that the Fluid Cooler must meet. Then we numerically invert the Fluid Cooler model to solve for the UA that will enable the Fluid Cooler to meet the design Fluid Cooler load given the specified inputs.

For Fluid Cooler *Performance Input Method = Nominal Capacity*

$$Q_{\text{fluidcooler, des}} = Q_{\text{fluidcooler, nom}}$$

Then we assign the inputs needed for the model.

T_{win} = inlet water temperature, °C

T_{wout} = outlet water temperature, °C

$T_{\text{in,air}}$ = design air inlet temperature

$T_{\text{in,air,wb}}$ = design air inlet wetbulb temperature

W_{in} is calculated from the entering air drybulb and wetbulb temperature.

The inlet water mass flow rate is just the design water volumetric flow rate times the density of water.

The actual Fluid Cooler heat transfer coefficient-area product is related to UA_e by the following expression:

$$UA = UA_e \frac{C_p}{\bar{c}_{pe}} \quad (404)$$

And the mean specific heat of air is defined by the following equation:

$$\bar{c}_{pe} = \frac{\Delta h}{\Delta T_{db}} \quad (405)$$

where

Δh = Enthalpy difference between the air entering and leaving the fluid cooler, J/kg

ΔT_{db} = Temperature difference between the air entering and leaving the fluid cooler, °C

This heat transfer coefficient-area product is assumed to be a function of the air mass flow rate only and can be estimated from laboratory test results or manufacturers' catalog data.

Method for Calculating Steady-State Exiting Water Temperature

The objective of the Fluid Cooler model is to predict the exiting water temperature and the fan power required to meet the exiting water setpoint temperature. Since only the inlet air and inlet water temperatures are known at any simulation time step, an iterative procedure is required to determine the exiting fluid temperatures using the equations defined in the previous section. In the case of the EnergyPlus model, the iterations are performed to determine the exiting dry-bulb temperature of the air. The exiting water temperature is then calculated based on an energy balance that assumes that the energy absorbed by the air is equivalent to the energy removed from the water. The procedure for calculating the steady-state, exiting air dry-bulb temperature is outlined below.

The first step in the procedure is to calculate the enthalpy of air entering the Fluid Cooler based on the ambient dry-bulb temperature from the weather file. Since an iterative solution is required, a first guess of the outlet air dry-bulb temperature is then made and the enthalpy of this estimated outlet air dry-bulb temperature is calculated. Based on these inlet and outlet air conditions, the mean specific heat of the air is calculated based on equation (405), repeated here:

$$\bar{c}_{pe} = \frac{\Delta h}{\Delta T_{db}}$$

With the overall heat transfer coefficient-area product for the Fluid Cooler is calculated by the nominal capacity information entered by the user, the effective heat transfer coefficient-area product is calculated by rearranging equation (404):

$$UA_e = UA \frac{\bar{c}_{pe}}{c_p}$$

With \bar{c}_{pe} and UA_e known, the effectiveness of the heat exchanger is then calculated:

$$\varepsilon = 1 - \exp\left(\frac{e^{-NTU \cdot Z \cdot \eta} - 1}{Z \cdot \eta}\right)$$

The heat transfer rate is then calculated as follows:

$$Q_{total} = \varepsilon * C_{\min}(T_{win} - T_{dbin}) \quad (406)$$

The outlet air dry-bulb temperature is then recalculated:

$$T_{dbout} = T_{dbin} + \frac{Q_{total}}{C_a} \quad (407)$$

The iterative process of calculating T_{wbout} continues until convergence is reached.

Finally, the outlet water temperature is calculated as follows:

$$T_{wout} = T_{win} + \frac{Q_{total}}{\dot{C}_w} \quad (408)$$

T_{win} = inlet water temperature, °C

T_{wout} = outlet water temperature, °C

T_{dbin} = dry-bulb temperature of the inlet air, °C

T_{dbout} = dry-bulb temperature of the outlet air, °C

Calculating the Actual Exiting Water Temperature and Fan Power

The previous section describes the methodology used for calculating the steady-state temperature of the water leaving the fluid cooler. This methodology is used to calculate the exiting water temperature with the Fluid Cooler fans operating (including low and high fan speed for the two-speed fluid cooler). The exiting water temperature calculations use the fluid flow rates (water and air) and the Nominal capacity information entered by the user for each regime.

The Fluid Cooler model seeks to maintain the temperature of the water exiting the Fluid Cooler at (or below) a setpoint. The setpoint schedule is defined by the field “Loop Temperature Setpoint Node or reference” for the CondenserLoop object.

The Fluid Cooler fans are turned on to reduce the exiting water temperature to the setpoint. The model assumes that part-load operation is represented by a simple linear interpolation between two steady-state regimes (e.g., Fluid Cooler fans on for the entire simulation time step and Fluid Cooler fans off for the entire simulation time step). Cyclic losses are not taken into account.

The fraction of time that the Fluid Cooler fans must operate is calculated based on the following equation:

$$\omega = \frac{T_{set} - T_{wout,off}}{T_{wout,on} - T_{wout,off}} \quad (409)$$

Where

T_{set} = exiting water setpoint temperature, °C

$T_{wout,off}$ = exiting water temperature with all Fluid Cooler fans off, °C

$T_{wout,on}$ = exiting water temperature with all Fluid Cooler fans on, °C

The average fan power for the simulation time step is calculated by multiplying ω by the steady-state fan power specified by the user.

The calculation method for the two-speed Fluid Cooler is similar to that for the single-speed Fluid Cooler example described above. The model first calculates the steady-state exiting water temperature with the Fluid Cooler fans at low speed. If the exiting water temperature at low fan speed is below the setpoint temperature, then the average fan power is calculated based on the result of equation (409) and the steady-state, low speed fan power specified by the user. If low-speed fan operation is unable to reduce the exiting water temperature below the setpoint, then the Fluid Cooler fans’ speed is increased to high speed and the steady-state exiting water temperature is calculated. If this temperature is below the setpoint, then a modified version of equation (409) is used to calculate runtime at high fan speed:

$$\omega = \frac{T_{set} - T_{wout,low}}{T_{wout,high} - T_{wout,low}} \quad (410)$$

where

T_{set} = exiting water setpoint temperature, °C

$T_{wout,low}$ = exiting water temperature with Fluid Cooler fans at low speed, °C

$T_{wout,high}$ = exiting water temperature with Fluid Cooler fans at high speed, °C

The average fan power for the simulation time step is calculated for the two-speed Fluid Cooler as follows

$$P_{fan,avg} = \omega(P_{fan,high}) + (1 - \omega)(P_{fan,low})$$

Fluid Cooler Sizing

The quantities needed to autosize a Fluid Cooler include the nominal fan power and air flow rate, and the Fluid Cooler UA. This data may be needed to be given at more than one operating point, for instance – high speed fan and low speed fan. EnergyPlus provides two input choices: the user can input the design water flow rate and Fluid Cooler UA at each operating point or the Fluid Cooler nominal capacity and the design water flow rate (and let the program calculate the UA). Choice of input method will affect the sizing calculations in ways noted below.

Fan Power at Design Air Flow Rate

The nominal fan power is sized to be 0.0105 times the design load.

If Fluid Cooler *Performance Input Method* = UA and Design Water Flow Rate then

$$Q_{fluidcooler, nom} = C_{p,w} * \rho_w * V_{fluidcooler, w} * \Delta T_{loop, des}$$

where

$C_{p,w}$ is the specific heat of water at the condenser loop design exit temperature;

ρ_w is the density of water at standard conditions (5.05 °C);

$\Delta T_{loop, des}$ is the condenser water loop design temperature rise;

Finally

$$Q_{fan, nom} = 0.0105 * Q_{fluidcooler, nom}$$

Design Air Flow Rate

We assume a fan efficiency of 0.5 and a fan pressure rise of 190 Pascals. Then

$$V_{fluidcooler, air, des} = Q_{fan, nom} * 0.5 * \rho_{air} / 190$$

where

ρ_{air} is the density of air at standard conditions.

Fluid Cooler UA Value at Design Air Flow Rate

To obtain the UA of the fluid cooler, we specify the model inputs (other than the UA) at design conditions and the design Fluid Cooler load that the Fluid Cooler must meet. Then we numerically invert the Fluid Cooler model to solve for the UA that will enable the Fluid Cooler to meet the design Fluid Cooler load given the specified inputs.

The design Fluid Cooler load is:

For Fluid Cooler *Performance Input Method = UA and Design Water Flow Rate*

$$Q_{fluidcooler, des} = C_{p, w} * \rho_w * V_{fluidcooler, w, des} * \Delta T_{loop, des}$$

For Fluid Cooler *Performance Input Method = Nominal Capacity*

$$Q_{fluidcooler, des} = Q_{fluidcooler, nom}$$

Then we assign the inputs needed for the model.

$T_{in,air}$ = design air inlet temperature

$T_{in,air,wb}$ = design air inlet wetbulb temperature

W_{in} is calculated from the entering air drybulb and wetbulb.

The inlet water mass flow rate is just the design water volumetric flow rate times the density of water.

The inlet water temperature is set slightly differently for the 2 input methods. For

UA and Design Water Flow Rate

$$T_{in,water} = T_{loop,exit,des} + \Delta T_{loop,des}$$

Nominal Capacity

$T_{in,water}$ = design inlet water temperature

We now have all the data needed to obtain UA. The numerical inversion is carried out by calling subroutine *SolveRegulaFalsi*. This is a general utility routine for finding the zero of a function. In this case it finds the UA that will zero the residual function – the difference between the design Fluid Cooler load and the Fluid Cooler output divided by the design Fluid Cooler load. The residual is calculated in the function *SimpleFluidcoolerUAResidual*.

Air Flow Rate at Low Fan Speed

The nominal air flow rate at low fan speed is set to one-half of the full speed air flow rate.

Fan Power at Low Fan Speed

The fan power at low fan speed is set to 0.16 times the fan power at full speed.

Fluid Cooler UA Value at Low Fan Speed

For Fluid Cooler *Performance Input Method = UA and Design Water Flow Rate* the low speed UA is set to 0.6 times the full speed UA. For Fluid Cooler *Performance Input Method = Nominal Capacity* the low speed UA is calculated in the same manner as the full speed UA using $Q_{fluidcooler, nom, lowspeed}$ instead of $Q_{fluidcooler, nom}$.

Demand Controlled Ventilation

ASHRAE Standard 62, Ventilation for Acceptable Indoor Air Quality, contains provisions that allow building ventilation systems to vary the amount of outside ventilation air delivered to occupied zones based on feedback from sensors that monitor various indoor air contaminants (ASHRAE 2008). Although not a contaminant of concern in most buildings, carbon dioxide (CO₂) levels can be monitored as an indicator of building occupancy and the associated human bioeffluent concentration. CO₂-based demand controlled ventilation (DCV) is being increasingly used to modulate outside ventilation air based on real-time occupancy (Emmerich and Persily 1997, Schell et al. 1998, Schell and Int-Hout 2001). Modulating the outside ventilation air while maintaining proper indoor air quality has the potential for large energy savings compared to constant rate ventilation systems that are typically designed to provide outside ventilation air based on maximum anticipated occupancy.

EnergyPlus can model CO₂-based DCV by the ventilation rate procedure (VRP) defined in ASHRAE Standard 62.1-2007 for single path systems. The VRP first calculates the breathing-zone outdoor air flow rate based on two components – the zone occupant component and the zone floor area component, then it calculates the zone supply outdoor air flow rate considering the zone air distribution effectiveness, and finally calculates the system outdoor air flow rate considering the zone diversity and system ventilation efficiency. The user must include the following three objects in their input data file in order to model CO₂-based DCV:

- AirLoopHVAC:OutdoorAirSystem to simulate the mixed air box of the air loop
- Controller:MechanicalVentilation to determine the minimum outside air flow rate to be provided by the mixed air box
- Controller:OutdoorAir to control the outside air flow rate introduced via the mixed air box

The outside air system is a subsystem of an AirLoopHVAC which handles the mixed air portion of the primary air system: the system relief air, the outside air inlet, and any components and controllers associated with the system relief air and outside air streams. The inputs for this object are fully described in the EnergyPlus Input Output Reference. Determining the outside air ventilation rate and introducing this ventilation via the mixed air box are accomplished by the mechanical ventilation and outside air controller objects.

The VRP currently requires outdoor air ventilation rates to be determined based on the floor area of each occupied zone plus the number of people in each zone. The outside air ventilation rate can be reset dynamically as operating conditions change (e.g., variations in occupancy). The Controller:MechanicalVentilation object simplifies the procedure for calculating these outside air ventilation requirements and resetting them based on varying occupancy levels. This is particularly useful for large air distribution systems that serve a number of different zone types with varying occupancy levels.

Ventilation Rate Procedure

Calculation of zone minimum outdoor air flow

For the *i*-th zone, first the breathing-zone outdoor air flow is calculated,

$$V_{bz,i} = R_{p,i} \times P_i + R_{a,i} \times A_i$$

Where:

$V_{bz,i}$ = the breathing-zone outdoor air flow, m³/s

$R_{p,i}$ = the ventilation rate per person, m³/s-person

P_i = the number of occupants for the zone

$R_{a,i}$ = the ventilation rate per floor area, m³/s-m²

A_i = the floor area of the zone, m²

i = the index of the zone or zone list

Next the *i*-th zone outdoor air flow is calculated,

$$V_{oz,i} = V_{bz,i} / E_i$$

Where:

$V_{oz,i}$ = the zone outdoor air flow, m³/s

E_i = the zone air distribution effectiveness, see following table for ASHRAE recommended values.

Zone Air Distribution Effectiveness

Air Distribution Configuration	E_z
Ceiling supply of cool air	1.0
Ceiling supply of warm air and floor return	1.0
Ceiling supply of warm air 15°F (8°C) or more above space temperature and ceiling return.	0.8
Ceiling supply of warm air less than 15°F (8°C) above space temperature and ceiling return provided that the 150 fpm (0.8 m/s) supply air jet reaches to within 4.5 ft (1.4 m) of floor level. Note: For lower velocity supply air, $E_z = 0.8$.	1.0
Floor supply of cool air and ceiling return provided that the 150 fpm (0.8 m/s) supply jet reaches 4.5 ft (1.4 m) or more above the floor. Note: Most underfloor air distribution systems comply with this proviso.	1.0
Floor supply of cool air and ceiling return, provided low-velocity displacement ventilation achieves unidirectional flow and thermal stratification	1.2
Floor supply of warm air and floor return	1.0
Floor supply of warm air and ceiling return	0.7

Figure 155. Controller:MechanicalVentilation Zone Air Distribution Effectiveness Typical Values (Source: ASHRAE Standard 62.1-2007)

Calculation of system minimum outdoor air flow

For single zone systems, the system outdoor air flow,

$$V_{OZ} = V_{OZ}$$

For 100% outdoor air multi-zone systems, the system outdoor air flow,

$$V_{OZ} = \sum_{i=1}^{\text{all zones}} V_{OZi}$$

For non 100% outdoor air multi-zone systems, the system outdoor air flow,

$$V_{OZ} = V_{OZu} / E_v$$

Where:

V_{OZu} = the uncorrected system outdoor air flow, m³/s

$$V_{OZu} = \sum_{i=1}^{\text{all zones}} (R_{p,i} \times P_i) + \sum_{i=1}^{\text{all zones}} (R_{a,i} \times A_i)$$

E_v = the system ventilation efficiency

$$E_v = 1 + X_s - \max(Z_{d,t})$$

X_s = the uncorrected system outdoor air fraction

$$X_s = V_{ow}/V_{ps}$$

V_{ps} = the system supply air flow, m³/s

$Z_{d,t}$ = the zone outdoor air fraction

$$Z_{d,t} = V_{oz,t}/V_{iz,t}$$

$V_{oz,t}$ = the zone supply air flow, m³/s

The program then calculates the minimum outside air flow fraction based on the information provided in the object Controller:MechanicalVentilation and the maximum branch air flow rate as follows:

$$\text{MechVentOutsideAirMinFrac} = V_{out} * \text{RhoStd} / \dot{m}_{\text{max branch}}$$

where:

MechVentOutsideAirMinFrac = outside air minimum fraction based on all zones specified in the Controller:MechanicalVentilation object

Controller:OutdoorAir controls the amount of outside ventilation air introduced via the mixed air box based on several user inputs. The user can define the minimum outside air flow rate as a percentage of the system's supply air flow rate (e.g., for a variable-air volume system) or a fixed minimum outside air flow rate (not as a percentage but a fixed value) (field MinimumLimit). CO₂-based DCV, using the Controller:MechanicalVentilation object in conjunction with the Controller:OutdoorAir object, allows a third option for setting the minimum outside air flow. Economizer operation can also be specified to increase the outside air flow above the minimum flow rate to provide free cooling when conditions permit (Controller:OutdoorAir, field Economizer Control Type).

EnergyPlus uses the largest outside air flow rate calculated by the various methods described above when modeling system performance (as long this rate doesn't exceed the maximum flow rate specified for the main air loop branch or for the outside air controller itself).

The method used to calculate the outside ventilation air flow rate for each system simulation time step is described in more detail below. The figure below schematically illustrates air flow paths used in the calculation of outside air flow rate.

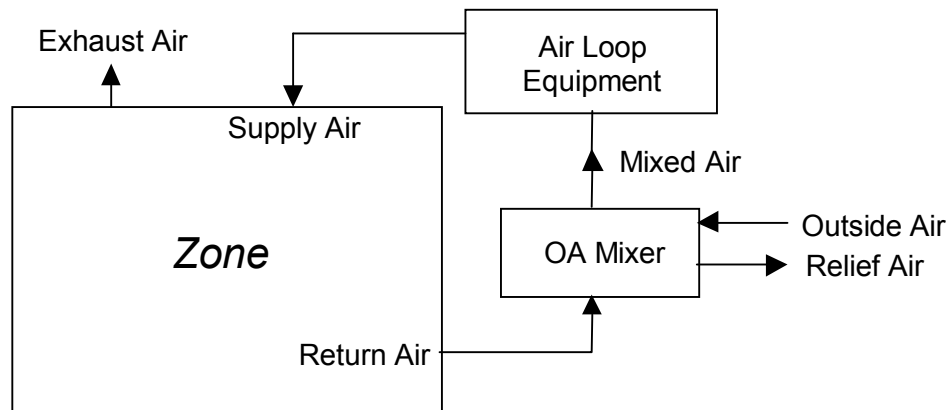


Figure 156. Demand Control Ventilation -- Air Flow Paths

The minimum outside air flow rate is first calculated based on the minimum outside air flow rate and the minimum outside air schedule value as defined by the user inputs for the object Controller:OutdoorAir:

$$\dot{m}_{\min OA} = \dot{V}_{\min OA} * MinOAScheduleValue * RhoStd$$

where:

$\dot{m}_{\min OA}$ = minimum outside air flow rate for this time step, kg/s

$\dot{V}_{\min OA}$ = minimum outside air flow rate defined in Controller:OutdoorAir, m³/s

MinOAScheduleValue = minimum outside air schedule value defined by the schedule identified in Controller:OutdoorAir

RhoStd = standard air density (1.204 kg/m³) adjusted for the local barometric pressure (standard barometric pressure corrected for altitude, ASHRAE 1997 HOF pg. 6.1).

The outside air minimum fraction is then calculated as the ratio of the minimum outside air flow rate calculated above to the maximum air flow rate defined in the Branch statement for the main air loop (converted to mass flow rate).

$$OutAirMinFrac = \left(\frac{\dot{m}_{\min OA}}{\dot{m}_{\max branch}} \right)$$

where:

$\dot{m}_{\max branch}$ = Max branch air volume flow rate times *RhoStd*, kg/s

The program then calculates the minimum outside air flow fraction (*MechVentOutsideAirMinFrac*) according to the VRP based on the information provided in the object Controller:MechanicalVentilation and the maximum branch air flow rate.

The algorithms then uses the larger of these outside air minimum fractions in subsequent calculations, and also makes sure that the resulting fraction is between 0 and 1.

$$OutAirMinFrac = MAX (OutAirMinFrac, MechVentOutsideAirMinFrac)$$

$$OutAirMinFrac = MIN \left(MAX \left(OutAirMinFrac, 0.0 \right), 1.0 \right)$$

The algorithm goes on to determine if economizer operation is possible based on the user inputs and the current conditions of the outdoor air and return air. If conditions permit economizer operation, the outside air flow fraction is increased beyond the minimum fraction to meet the mixed air setpoint temperature (setpoint temperature assigned to the node defined in field "Control_Node" of Controller:OutdoorAir).

$$OAFrac = MAX \left(OutAirMinFrac, EconomizerOAFrac \right)$$

The mass flow rate of outside air is then calculated based on the outside air fraction determined above and the mixed (supply) air mass flow rate:

$$\dot{m}_{OutsideAir} = OAFrac * \dot{m}_{MixedAir}$$

where:

$\dot{m}_{OutsideAir}$ = mass flow rate of outside air, kg/s

$OAFrac$ = fraction of outside air in the mixed (supply) air stream

$\dot{m}_{MixedAir}$ = mass flow rate of the mixture of return air and outside ventilation air, kg/s

The algorithm checks to make sure the calculated outside air mass flow rate is greater than or equal to the air flow rate being exhausted.

$$\dot{m}_{OutsideAir} = MAX \left(\dot{m}_{OutsideAir}, \dot{m}_{ExhaustAir} \right)$$

If a fixed minimum outside air flow rate is specified (field Minimum Limit Type in Controller:OutdoorAir) for a continuous air flow system, the program makes sure that the outside air mass flow rate is greater than or equal to the minimum outside air flow rate specified by the user.

$$\dot{m}_{OutsideAir} = MAX \left(\dot{m}_{OutsideAir}, \dot{m}_{minOA} \right)$$

The outside air mass flow rate should be less than or equal to the mixed (supply) air flow rate, and the outside air flow rate is reset if necessary.

$$\dot{m}_{OutsideAir} = MIN \left(\dot{m}_{OutsideAir}, \dot{m}_{MixedAir} \right)$$

The outside air mass flow rate should also be less than or equal to the maximum outside air flow rate specified by the user, and the outside air flow rate is reset if necessary.

$$\dot{m}_{OutsideAir} = MIN \left(\dot{m}_{OutsideAir}, \dot{m}_{maxOA} \right)$$

where:

$\dot{m}_{\max OA}$ = maximum outside air mass flow rate, kg/s = maximum outside air volume flow rate from Controller:OutdoorAir times $RhoStd$

Finally, the relief air flow rate is calculated as the difference between the outside and exhaust air mass flow rates.

$$\dot{m}_{\text{ReliefAir}} = \text{MAX} \left(\dot{m}_{\text{OutsideAir}} - \dot{m}_{\text{ExhaustAir}}, 0.0 \right)$$

References

ASHRAE. 2008. *ANSI/ASHRAE Standard 62-2007, Ventilation for acceptable indoor air quality*. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Emmerich, S.J. and A.K. Persily. 1997. Literature review on CO₂-based demand-controlled ventilation. *ASHRAE Transactions* 103(2):229-243.

Schell, M.B., S.C. Turner, R.O. Shim. 1998. Application of CO₂-based demand controlled ventilation using ASHRAE Standard 62-1989. *ASHRAE Transactions* 104(2):1213-1225.

Schell, M. and D. Int-Hout. 2001. Demand control ventilation using CO₂. *ASHRAE Journal*, February.

Evaporative Coolers

This section describes the evaporative coolers models for HVAC in EnergyPlus.

Direct Evaporative Cooler

The input object EvaporativeCooler:Direct:CellDekPad provides a model of a direct stage evaporative cooler, shown in the figure below, that consists of a rigid media evaporative pad, with water recirculated from a reservoir. The water is pumped from the reservoir to a water distribution header, for water feed by gravity from above the media. The evaporative pad provides the area for the adiabatic saturation of the air. While the process provides a lower dry-bulb temperature, the moisture content of the leaving air is higher than the entering condition. The direct stage is used for comfort cooling in a building where adding humidity to the air can be tolerated.

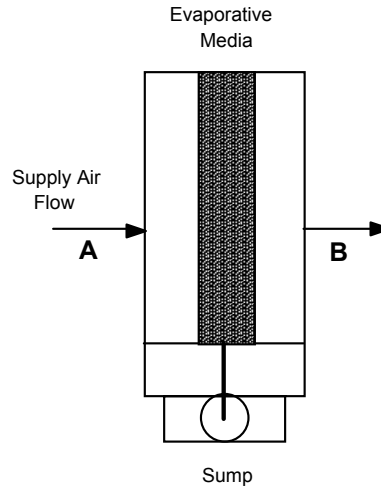


Figure 157. Direct Stage Evaporative Cooler

The thermodynamic process is a simultaneous heat and mass transfer, or adiabatic cooling, and follows a constant enthalpy line on the psychrometric chart; it is shown in the figure below as a process from A to B. Since the deviation of the constant wet-bulb line and the constant enthalpy line is small, it is assumed that the wet-bulb temperature is constant across the direct evaporative stage.

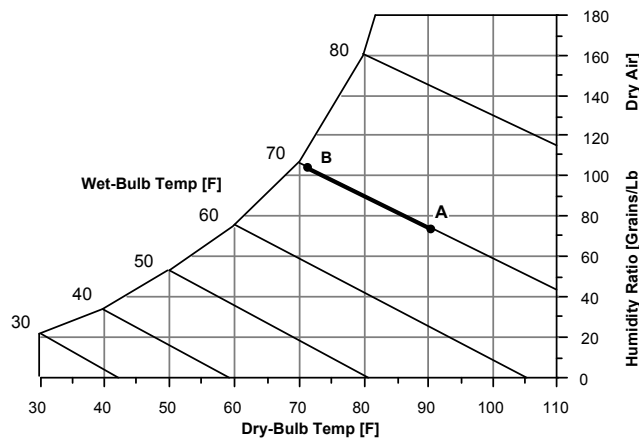


Figure 158. Psychrometric Chart -- Constant Enthalpy

If the direct evaporative process were 100% efficient, the leaving dry-bulb temperature would equal the entering wet-bulb temperature. The efficiency of the direct evaporative process is less than 100% and by defining saturation efficiency (ϵ_{se}) for the direct stage or evaporative pad, the leaving dry-bulb temperature can be expressed by the following equation.

$$T_{db\,sup\,out} - T_{db\,sup\,in} = \epsilon_{se} \cdot (T_{o\,db} - T_{o\,wb})$$

Saturation Efficiency

Since the evaporative process is not 100% efficient the saturation efficiency is defined by.

$$\epsilon_{se} = \frac{T_{db\,sup\,in} - T_{db\,sup\,out}}{T_{o\,db} - T_{o\,wb}}$$

The saturation efficiency is determined from manufacturer's data, and the least squares curve fit is discussed in Curve Fitting Evaporative Media section.

Using the saturation efficiency (ϵ_{SE}) for the direct stage evaporative pad, the leaving dry-bulb temperature can be determined directly. The evaporative process approximately follows a constant wet-bulb line. Therefore, with the leaving dry-bulb temperature and assuming adiabatic heat transfer across the direct stage, the outlet conditions for the direct stage are known.

The saturation efficiency of the direct evaporative cooler is a function of the pad geometry and airflow rate. The pad geometry is constant throughout the simulation, but the airflow rate can change from hour to hour when the evaporative cooler is used with an air economizer. The saturation efficiency would then be determined from the flow for that hour with the geometry of the direct evaporative cooler. This gives the dry-bulb temperature leaving the evaporative cooler. Assuming adiabatic heat transfer across the direct stage, the evaporative process follows the constant wet-bulb line or the constant enthalpy line on the psychrometric chart, therefore the wet-bulb temperature is constant from inlet to outlet.

Some things that can cause departure from the ideal adiabatic saturation process in the direct evaporative cooler are:

- makeup water entering the sump,
- friction from water re-circulation,
- heat transfer from surroundings,
- solar radiation (sun upon a cooler).

Thus, adiabatic saturation in evaporative cooling is only an approximation, however the adiabatic saturation assumption in the rigid-media cooler is good, since the water recirculates rapidly and approximates the wet-bulb temperature at steady state operation.

Curve Fitting Evaporative Media

The saturation efficiency is usually reported as a function of airflow, pad face velocity, and pad thickness. The Figure below shows a typical graph of manufacturer's data for the saturation efficiency. A multi-variate least squares curve fit of the data was used to generate saturation efficiency functions for the evaporative models that use the CelDek rigid media pad.

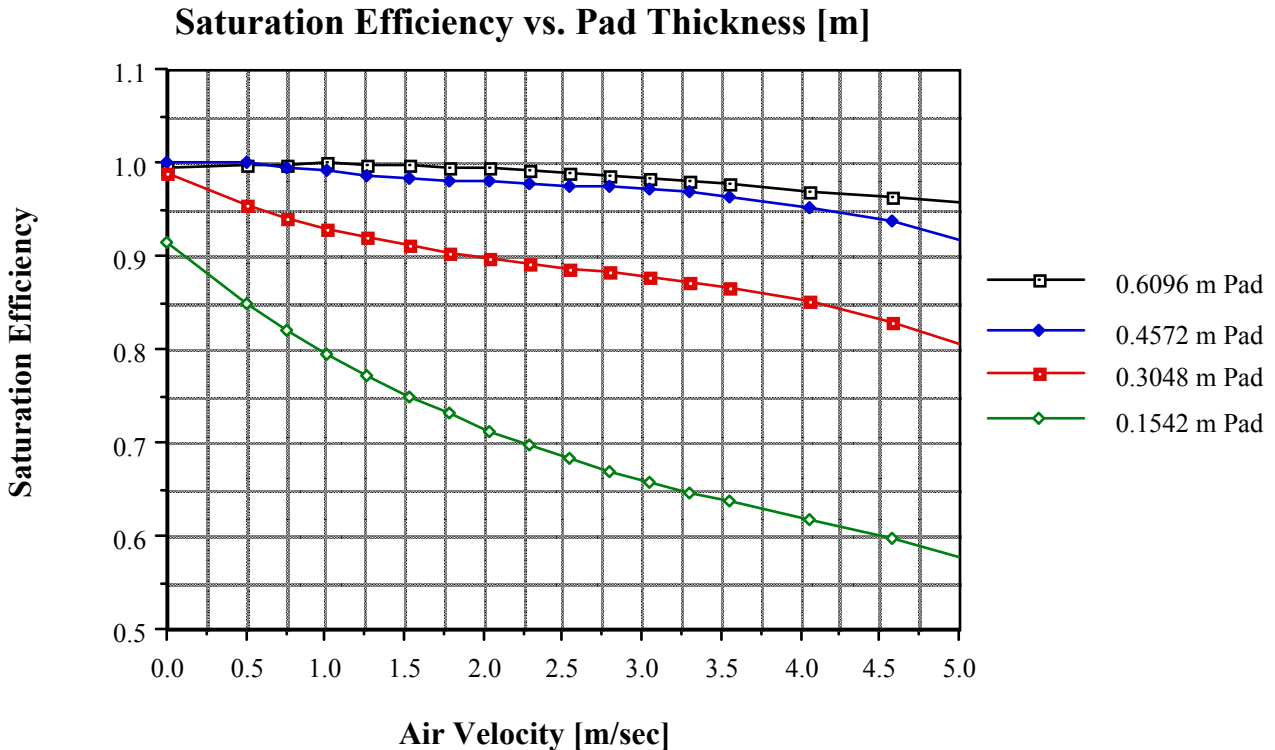


Figure 159. Graph of Saturation Efficiency

The curve fit for saturation efficiency was obtained using the functions listed below. The model uses the air velocity (Airvel) through the pad and the depth of the media (Depth). The least squares routine produced the following model that is used for the evaporative cooling rigid media pad. The least squares routine produced an eleven-term multi-variate fit using a third order quadratic.

$$\begin{aligned} \varepsilon_{se} = & 0.792714 + 0.958569 (\text{Depth}) - 0.25193 (\text{Airvel}) - 1.03215 (\text{Depth}^2) + \\ & 0.0262659 (\text{Airvel}^2) + 0.914869 (\text{Depth} * \text{Airvel}) - 1.48241 (\text{Airvel} * \text{Depth}^2) - 0.018992 \\ & (\text{Airvel}^3 * \text{Depth}) + 1.13137 (\text{Depth}^3 * \text{Airvel}) + 0.0327622 (\text{Airvel}^3 * \text{Depth}^2) - 0.145384 \\ & (\text{Depth}^3 * \text{Airvel}^2) \end{aligned}$$

Where Airvel is in meters per second and Depth is in meters. This curve fit is used for the rigid media in the EvapCooler:Direct:CeIDekPad and EvapCooler:InDirect:CeIDekPad.

Dry Coil Indirect Evaporative Cooler

The input object EvaporativeCooler:Indirect:CeIDekPad provides a model of a dry coil indirect evaporative cooler, shown in the figure below, that has a rigid media pad, similar to the direct evaporative stage, where the adiabatic cooling takes place. The secondary air leaves the rigid media pad and enters an air-to-air heat exchanger where it cools the supply air flowing through the heat exchanger tubes. The moist secondary air is then exhausted to the environment. The secondary air stream has its own fan and includes consists of a rigid media evaporative pad, with water recirculated from a reservoir. The water is pumped from the reservoir to a water distribution header, for water feed by gravity from above the media. The evaporative pad provides the area for the adiabatic saturation of the air.

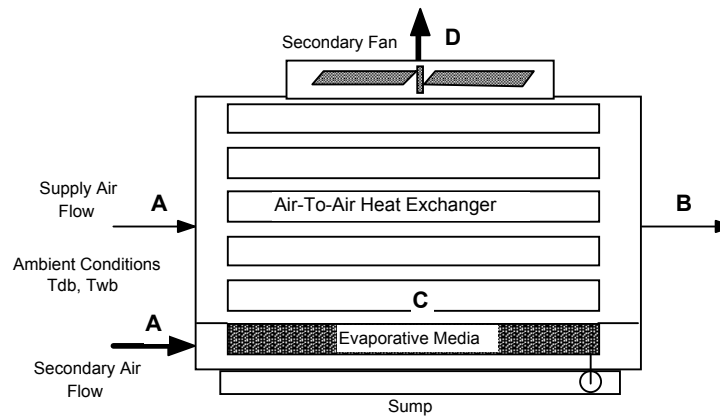


Figure 160. Evaporative Cooler -- Indirect Dry Coil

The process that the secondary air goes through, A to C to D, is shown by the dashed lines in the following figure. Process A to C is adiabatic cooling in the rigid media pad. Then the air enters the shell side of the heat exchanger and is sensibly heated from C to D by the warm supply air passing through the tube side.

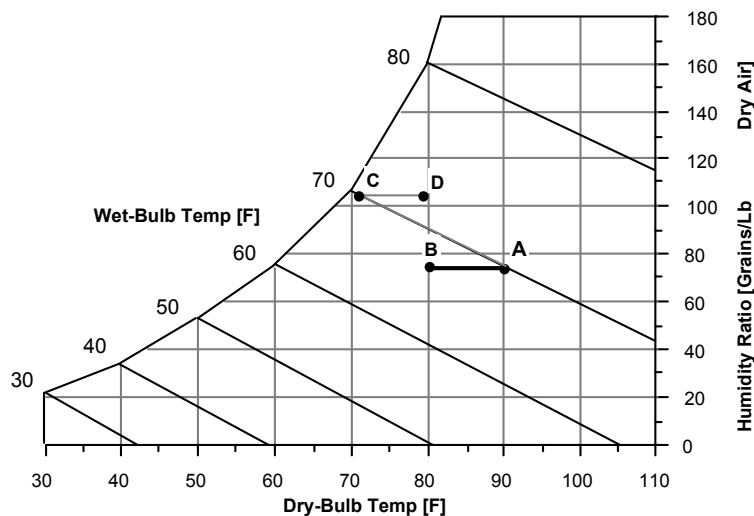


Figure 161. Secondary Air Process -- Indirect Dry Coil Evap Cooler

The advantage of the dry coil heat exchanger is that the heat exchanger does not have the evaporation taking place on the outside of the tubes, thus no mineral deposits are left on the heat exchange surface to reduce the efficiency of the heat exchanger. The rigid media pads are designed to flush the mineral deposits to the sump, so the saturation efficiency of the pad stays relatively constant.

The following equations are used to determine the dry-bulb temperature leaving the evaporative media, given pad geometry and secondary airflow information. The heat transfer in the heat exchanger can be determined with the effectiveness of the heat exchanger according.

$$T_{db \text{ sup out}} = T_{db \text{ sup in}} - \epsilon_{se} \cdot (T_{odb} - T_{owb})$$

$$Q_{HX} = \epsilon_{HX} \cdot \text{Min}(CFM_{sec}, CFM_{supply}) \cdot \rho_{air} \cdot c_{p \text{ air}} \cdot (T_{odb} - T_{db \text{ sec out}})$$

After the heat transfer for the heat exchanger has been determined, an energy balance is done on the supply airside to determine the dry-bulb temperature leaving the indirect

evaporative cooler. This assumes all the energy for is provided by the primary air stream so the effectiveness value includes the air-to-air effectiveness of the heat exchanger.

$$T_{db\ sup\ out} = T_{db\ sup\ in} - \frac{Q_{Hx}}{r_{air} * c_{pair} * CFM_{supply}}$$

The wet-bulb temperature is determined from psychrometric routines using the leaving dry-bulb temperature, humidity ratio, and barometric pressure, since humidity ratio is constant for the supply air across the indirect stage. The effectiveness of the heat exchanger is determined from a parameter estimation using manufacturer's performance data. For the indirect evaporative cooler it was found that a value of 0.67 represented reasonable default effectiveness.

Wet Coil Indirect Evaporative Cooler

The input object EvaporativeCooler:Indirect:WetCoil provides a model for a wetted coil evaporative cooler, shown in the figure below, that has water sprayed directly on the tubes of the heat exchanger where latent cooling takes place. The vaporization of the water on the outside of the heat exchanger tubes allows the simultaneous heat and mass transfer which removes heat from the supply air on the tube side. Then the moist secondary air is exhausted. The secondary air stream has its own fan.

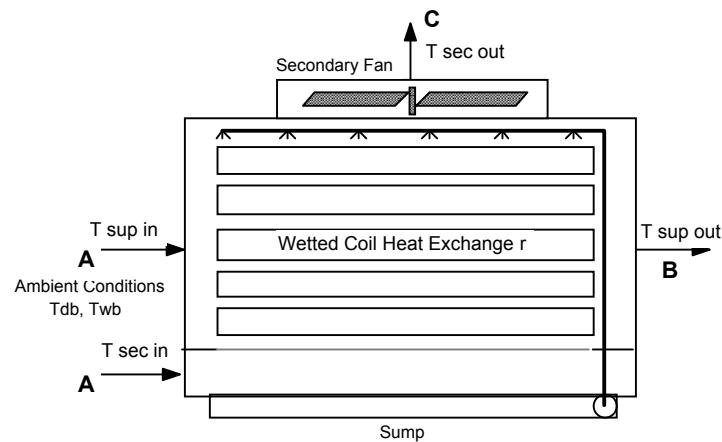


Figure 162. Wet Coil Indirect Evaporative Cooler

The process that the secondary air goes through, A to C on the following figure, is a path of simultaneous heat and mass transfer, but it does not follow a line of constant enthalpy as in the direct stage. The process is not adiabatic due to the heat gain from the supply air flowing through the tubes of the heat exchanger.

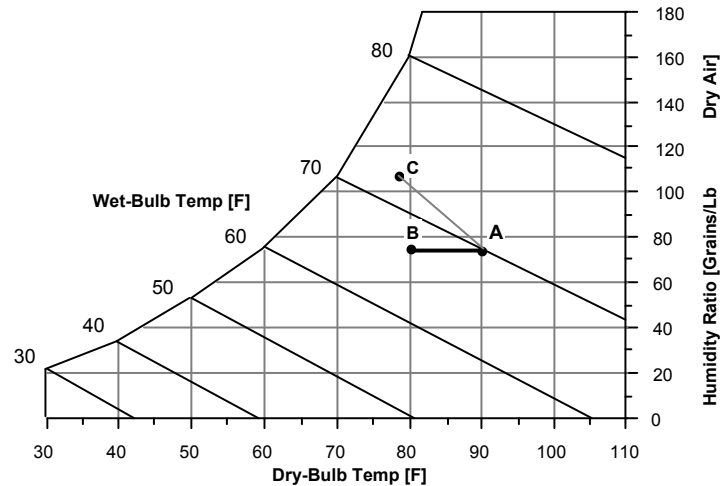


Figure 163. Secondary Air Process – Indirect Wet Coil Evaporative Cooler

The wet coil heat exchanger can have a higher stage efficiency than the dry coil due to a higher heat transfer rate on the outside of the heat exchanger tubes. Over the operating lifetime of the heat exchanger, the vaporization taking place on the heat exchange surface can leave mineral deposits that will decrease the effectiveness of the heat exchanger.

Efficiencies of the Indirect Stage

In an indirect stage of an evaporative cooler, the secondary or wet side air stream acts as a heat sink for the supply air. The efficiency of the indirect stage is given as the effectiveness of the sensible heat exchange, ϵ_{HX} , and the saturation efficiency on the wet streamside, ϵ_{se} . These are expressed as:

$$\epsilon_{HX} = \frac{q}{q_{max}} = \frac{C_{sup}(T_{supin} - T_{supout})}{C_{min}(T_{secin} - T_{secout})},$$

$$\epsilon_{se} = \frac{T_{dbsecin} - T_{dbsecout}}{T_{odb} - T_{owb}},$$

where $T_{db\ sup\ in} = T_{db\ sec\ in}$ for the indirect cooler. The maximum heat transfer possible would be obtained if the supply stream was cooled all the way to the wet-bulb temperature. So the efficiency of the indirect evaporative cooler is defined by:

$$\epsilon_{ind} = \frac{(T_{dbsupin} - T_{dbsupout})}{(T_{odb} - T_{owb})}.$$

Using the combination of the effectiveness and saturation efficiency, the total efficiency of the indirect stage can be expressed by:

$$\epsilon_{ind} = \epsilon_{HX} \epsilon_{se} \frac{C_{sup}}{C_{min}}.$$

In many cases $C_{sup} = C_{min}$ and the efficiency of the indirect stage reduces to:

$$\epsilon_{ind} = \epsilon_{HX} \epsilon_{se}.$$

An intuitive model determining the performance of the wet coil indirect model was developed. This model can be used for all indirect models by curve fitting data from the evaporative

cooler of interest. The model development starts with the total efficiency of the indirect evaporative cooler:

$$\epsilon_{\text{ind}} = \frac{(T_{\text{dbsupin}} - T_{\text{dbsupout}})}{(T_{\text{odb}} - T_{\text{owb}})}$$

Solving for $T_{\text{db sup out}}$ gives the leaving conditions of the supply air at a constant humidity ratio:

$$T_{\text{db sup out}} = T_{\text{db sup in}} - \epsilon_{\text{ind}} * (T_{\text{odb}} - T_{\text{owb}})$$

A form for the efficiency of the indirect stage was devised using a maximum efficiency with a coefficient to reduce the efficiency depending on the ratio of the airflows.

$$\epsilon_{\text{ind}} = \epsilon_{\text{max}} - C_1 * \left(\frac{CFM_{\text{sup}}}{CFM_{\text{sec}}} \right)$$

C_1 is the "Flow Efficiency Ratio" and is determined from performance data.

A check of limits will verify that it makes physical sense. As the magnitude of the secondary flow increases, the second term of equation above becomes smaller. This would make the efficiency tend to go to the maximum efficiency. Physically this would be true since the convective terms for heat and mass transfer would increase on the outside of the tube with the additional mass flow rate. Similarly, if the supply air flow goes to zero with a constant secondary air flow, the second term of the equation above again becomes small, and the overall efficiency of the stage would approach the maximum. The constant C_1 tells how quickly the efficiency of the stage would decrease with a mismatch of the supply and secondary flows.

The maximum efficiency of the stage is a combination of the efficiency due to the simultaneous heat and mass transfer on the outside of the tube and the efficiency of the heat exchanger. This value can be higher than the dry coil overall efficiency since the convective coefficients on the outside of the tube are larger. For example, a least squares fit for the maximum efficiency showed this value was approximately 0.8 compared to the dry coil indirect value of approximately 0.65 ($0.67 * 0.97$). (The maximum efficiency for the dry coil indirect was determined at the condition where flow through the evaporative pad in the secondary air stream approached zero, for a 12-inch thick pad.) It should be noted again that over the operating life of the wet coil heat exchanger, the mineral deposits that are left can decrease the effectiveness of the heat exchanger unless appropriate maintenance has taken place. Therefore, if modeling an older system, the manufacturer's data may no longer describe the equipment effectiveness.

Two Stage Direct/Indirect Evaporative Cooler

A two stage cooler can be modeled by combining either a wet coil or the dry coil indirect evaporative cooler staged with a direct evaporative cooler. The figure below shows a dry coil indirect evaporative cooler with a direct evaporative cooler. This configuration is mainly used for total comfort cooling for a building and would not normally be used as a pre-cooler for a refrigeration coil, since the direct stage would increase the latent load on a refrigeration coil.

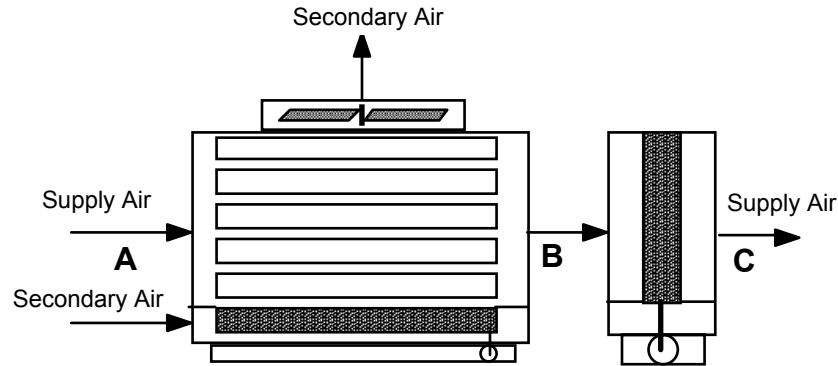


Figure 164. Two Stage Evaporative Cooler

The thermodynamic process for the supply air is shown below, going from A to B to C. The process from A to B is sensible cooling in the indirect stage. The process from B to C is simultaneous heat and mass transfer following a constant enthalpy line. The air leaving the final stage has a lower dry-bulb and wet-bulb temperature, and an increase in moisture from the direct stage.

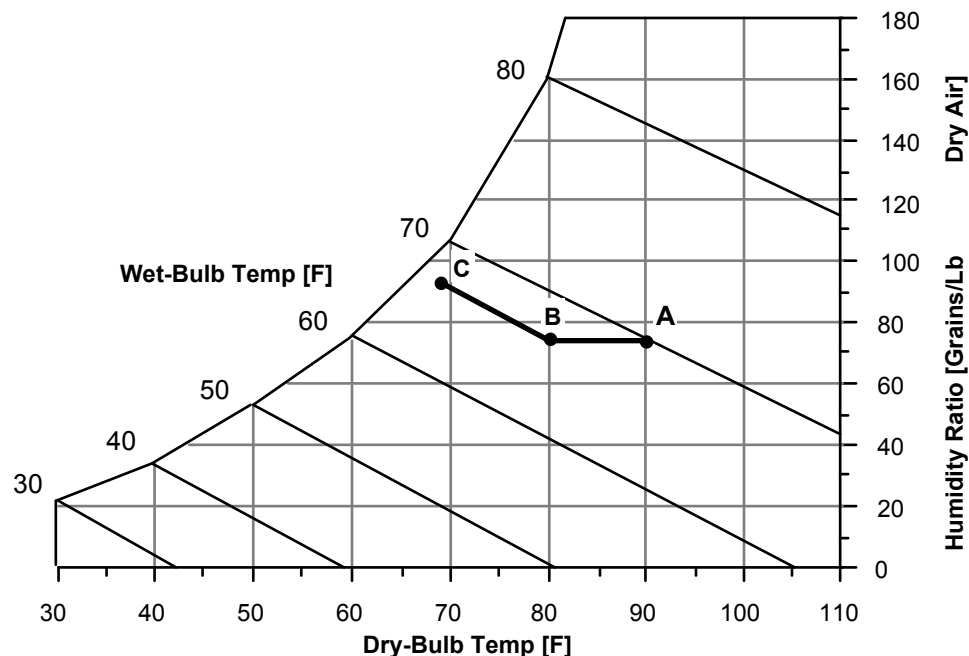


Figure 165. Thermodynamic Process for Supply Air Through Two Stage Evaporative Cooler

Two stage evaporative coolers can be accomplished by putting the EvaporativeCooler:Direct:CeIDekPad, EvaporativeCooler:Indirect:CeIDekPad, EvaporativeCooler:Indirect:WetCoil in series in any combination the user desires in the supply air loop.

Indirect Evaporative Cooler Special Research Model

This section summarizes the model implemented in the component EvaporativeCooler:Indirect:ResearchSpecial. Examples of this evaporative cooler are shown in the following figures, without and with a relief valve. This model differs from the other indirect evaporative coolers in that, under part load conditions, it can modulate so that the air leaving the cooler just meets a drybulb temperature setpoint. It is also a simple model with a constant effectiveness.

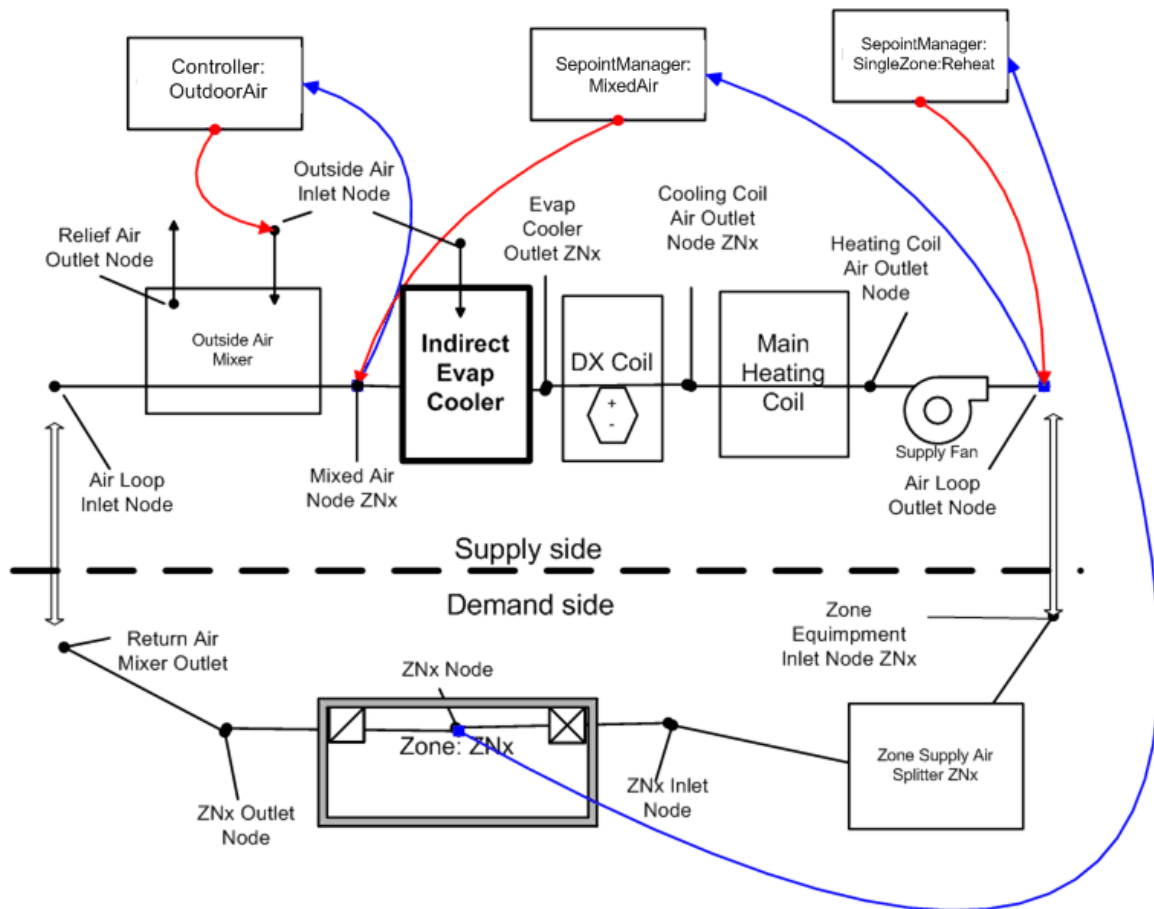


Figure 166. Research Special Indirect Evaporative Cooler

The algorithm used to determine the cooling provided to the system air proceeds in these three steps:

- 1) calculate full load performance using $PLF=1$ and Equation (411) and Equation (412)
- 2) calculate PLF using Equation (413), Equation (414), and Equation (415), and
- 3) recalculate performance using PLF from step 2.
 - If $PLF = 1$ then use Equation (411) and Equation (412)
 - If $PLF < 1$ then outlet temp = desired outlet temp (as by magic)
 - Auxiliary fan energy adjusted by PLF
 - Water consumption based on change in enthalpy in air system

$$T_{db,out,sys} = T_{db,in,sys} - \varepsilon (T_{db,in,sys} - T_{wb,in,purge}) \quad (411)$$

where,

$T_{db,out,sys}$ is the dry-bulb of the system air leaving the cooler [°C]

$T_{db,in,sys}$ is the dry-bulb of the system air entering the cooler [°C]

ε is a cooler effectiveness (eg. 0.7 to 1.2)

$T_{wb,in,purge}$ is the wet-bulb of the purge air entering the wet side of cooler [°C]

The purge air, or secondary airside, is the stream that evaporates water and indirectly cools the primary, or system air. The result from Equation (411) is then compared to a lower bound, $T_{db,out,bound}$, determined from the dewpoint of the incoming purge air using Equation (412).

$$T_{db,out,bound} = T_{db,in,sys} - \beta(T_{db,in,sys} - T_{dew,in,purge}) \quad (412)$$

where,

$T_{dew,in,purge}$ is the dewpoint of purge air entering the wet side of cooler [°C]

β is a factor for how close to dewpoint is possible (eg. 0.9)

The final result (for PLF = 1) is the larger of the results from Equations (411) and (412).

The indirect cooler has the ability to overcool the air and therefore needs some form of modulation. A Part Load Fraction, PLF, is used to model the implications of controlling the amount of cooling. It is assumed that through on/off cycling that the cooling power can be varied to exactly meet the desired temperature when PLF is less than unity. The auxiliary fan power is then varied linearly using a Part Load Fraction.

$$\dot{Q}_{Full} = \dot{m} (h_{out,sys} - h_{in,sys}) \quad (413)$$

$$\dot{Q}_{Required} = \dot{m} (h_{out,desired} - h_{in,sys}) \quad (414)$$

$$PLF = \frac{\dot{Q}_{Required}}{\dot{Q}_{Full}} \quad (415)$$

where,

PLF is the Part Load Fraction

When PLF is less than 1.0 it is assumed that the cooler will deliver the desired temperature air (as long as it is less than the inlet; it doesn't need heating). The PLF is used to modify the auxiliary fan power and find when the unit will overcool.

$$P_{fan} = \Delta P \cdot \dot{V} \cdot e \cdot PLF \quad (416)$$

Water pump power is also derated using the PLF.

A third air stream input to the cooler was implemented in order to allow mixing building exhaust air with outdoor air on the purge/secondary side of the cooler. The assumption when relief/tertiary air is used is that all of the available relief zone air is used and the remainder made up with outdoor air. Moisture and energy balances are drawn to compute humidity ratio and enthalpy of mixed secondary air. The volume is determined by the design volume flow rate (from secondary fan size).

Water Consumption

Water consumption is an important consideration when evaluating evaporative coolers. Water consumption of the evaporative cooler is modeled using Equation (417).

$$\dot{V}_{water} = \dot{V}_{evap} + \dot{V}_{drift} + \dot{V}_{blowdown} \quad (417)$$

The three components of water consumption are evaporation, drift, and blowdown. Evaporation is the water evaporated as the normal part of the evaporative cooler thermodynamic process and is calculated using:

$$\dot{V}_{evap} = \frac{\dot{Q}_{IEC}}{\rho_{water} h_{fg}}$$

where,

\dot{V}_{evap} is the volume flow rate of useful water evaporation [m³/s]

h_{fg} is the heat of vaporization of water (taken as 2,500,000 J/kg)

\dot{Q}_{IEC} is the rate of heat transfer calculated as by Equation (413) or Equation (414) [W]

ρ_{water} is the density of water [kg/m³]

Drift is water that leaves the secondary side as droplets and does not contribute to the evaporative cooling process in a useful manner. It is calculated using a user input factor that describes drift as a fraction of \dot{V}_{evap} .

$$\dot{V}_{drift} = \dot{V}_{evap} * f_{drift}$$

Blowdown is water drained from the sump to counter the build up of solids in the water that would otherwise occur because of evaporation. It is calculated using a user input factor for the blowdown concentration ratio, $R_{concentration}$, which is the ratio of solids in the blowdown water compared to the solids in the fresh makeup water and is limited to values of 2 or higher. The make up water needed for blowdown is calculated using:

$$\dot{V}_{blowdown} = \frac{\dot{V}_{evap}}{(R_{concentration} - 1.0)} - \dot{V}_{drift}$$

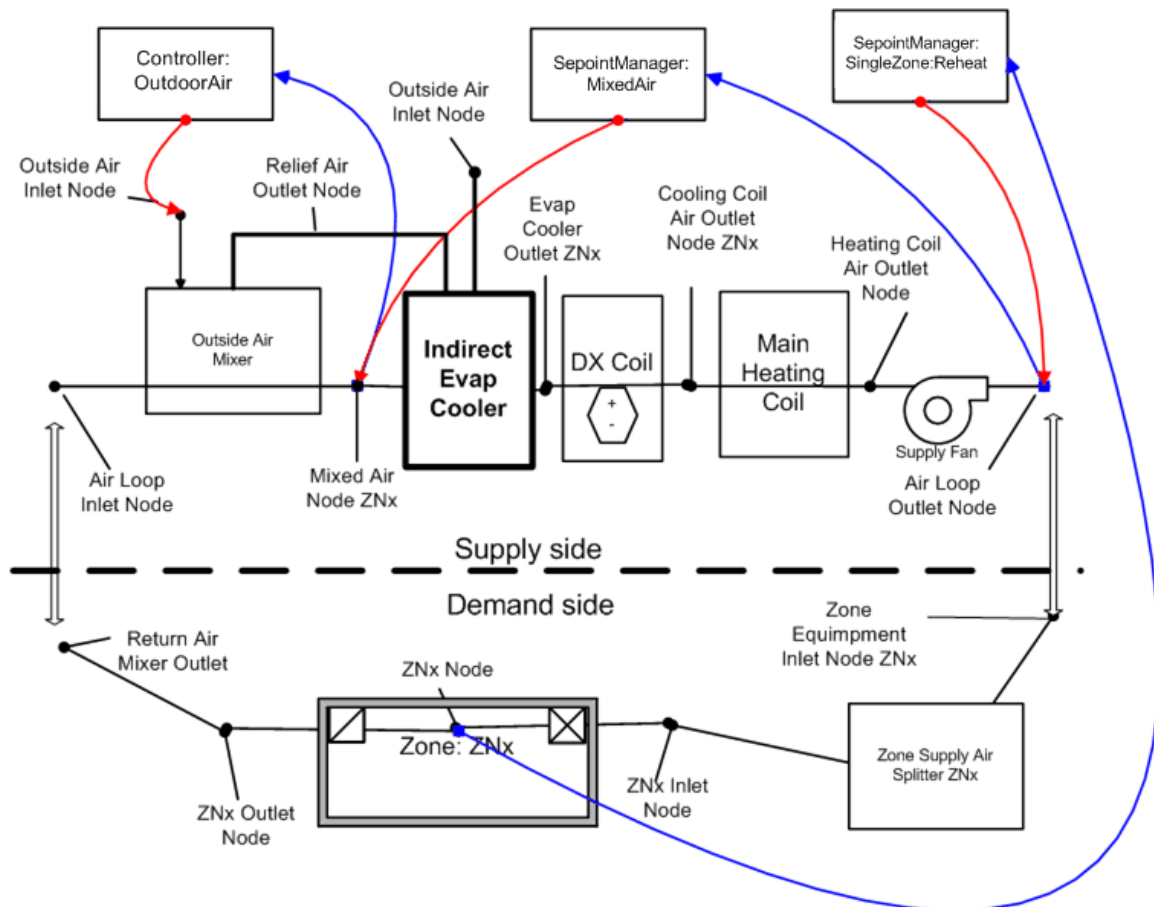


Figure 167. Research Special Indirect Evaporative Cooler Using Relief Air

Direct Evaporative Cooler Special Research Model

This section summarizes the model implemented in the component `EvaporativeCooler:Direct:ResearchSpecial`. This is a simple constant effectiveness model that, under part load conditions, can modulate so that the air leaving the cooler just meets a drybulb temperature setpoint. The algorithm used to determine the changes to the system air proceeds in three steps:

- 1) calculate full load performance using a part load fraction (PLF)=1 and Equation (418).
- 2) calculate PLF using Equations (419), (420), and (421).
- 3) recalculate cooler performance using the PLF.

$$T_{db,out} = T_{db,in} - \varepsilon (T_{db,in} - T_{wb,in}) \quad (418)$$

where,

$T_{db,out}$ is the drybulb temperature of the air leaving the cooler [°C],

$T_{db,in}$ is the drybulb temperature of the air entering the cooler [°C],

$T_{wb,in}$ is the wetbulb temperature of the air entering the cooler [°C], and

ε is the cooler effectiveness.

The wetbulb temperature of air leaving a direct cooler is the same as the wetbulb temperature entering the cooler. The leaving humidity ratio of the air is calculated using psychrometric functions with leaving drybulb and wetbulb temperatures and outdoor air pressure as inputs. The leaving enthalpy of air is calculated using psychrometric functions with leaving drybulb temperature, leaving humidity ratio, and outdoor air pressure as inputs.

The direct cooler sometimes has the ability to overcool the air and therefore some form of modulation is useful for analysis. The special research model includes a Part Load Fraction, PLF, used to model the implications of controlling the amount of cooling. It is assumed that through some sort of on/off cycling or wetness control that the cooling power can be varied to exactly meet the desired temperature when PLF is less than unity. The auxiliary water pump power is then varied linearly using a Part Load Fraction.

$$\text{FullOutput} = T_{db,out} - T_{db,in} \quad (419)$$

$$\text{RequiredOutput} = T_{db,desired} - T_{db,in} \quad (420)$$

$$PLF = \frac{\text{RequiredOutput}}{\text{FullOutput}} \quad (421)$$

When PLF is less than 1.0 it is assumed that the cooler will deliver the desired temperature air (as long as it is less than the inlet; it doesn't need heating). Water pump power is also derated using the PLF.

Water Consumption

Water consumption is an important consideration when evaluating evaporative coolers. Water consumption of the evaporative cooler is modeled using Equation (422).

$$\dot{V}_{water} = \dot{V}_{evap} + \dot{V}_{drift} + \dot{V}_{blowdown} \quad (422)$$

The three components of water consumption are evaporation, drift, and blowdown. Evaporation is the water evaporated as the normal part of the evaporative cooler thermodynamic process and is calculated using:

$$\dot{V}_{evap} = \frac{\dot{m}(w_{out} - w_{in})}{\rho_{water}}$$

where,

\dot{V}_{evap} is the volume flow rate of useful water evaporation [m^3/s]

w_{out} is the humidity ratio of the air leaving the cooler [kg/kg]

w_{in} is the humidity ratio of air entering the cooler [kg/kg]

\dot{m} is the mass flow rate of air moving through the cooler [kg/s]

ρ_{water} is the density of water [kg/m^3]

Drift is water that leaves the cooler (and supply air duct) as droplets and does not contribute to the evaporative cooling process in a useful manner. It is calculated using a user input factor that describes drift as a fraction of \dot{V}_{evap} .

$$\dot{V}_{drift} = \dot{V}_{evap} * f_{drift}$$

Blowdown is water drained from the sump to counter the build up of solids in the water that would otherwise occur because of evaporation. It is calculated using a user input factor for the blowdown concentration ratio, $R_{concentration}$, which is the ratio of solids in the blowdown water compared to the solids in the fresh makeup water and is limited to values of 2 or higher. The make up water needed for blowdown is calculated using:

$$\dot{V}_{blowdown} = \frac{\dot{V}_{evap}}{(R_{concentration} - 1.0)} - \dot{V}_{drift}$$

Air System Fans

Overview

The input objects Fan:ConstantVolume, Fan:VariableVolume, and Fan:OnOff provide models for fans that are the prime mover in most of the air loop and zonal air conditioning systems in EnergyPlus. A fan model must calculate not only the fan energy usage – often a significant portion of the building energy consumption – but also the temperature rise in the air stream caused by the static pressure increase as the air goes through the fan. If the motor is in the air stream, waste heat from the motor will also act to raise the air stream temperature. EnergyPlus uses a simple curve-fit model to describe the relation between the air mass flow rate and the fan power. In central air systems this type of model is only appropriate for systems with a fixed static pressure setpoint.

Model

The fan model is a forward model: the model inputs are the conditions of the air at the fan inlet; the outputs are the conditions at the fan outlet and the fan electrical power consumption. The model algorithms and data are contained in the *Fans* module in EnergyPlus.

Inputs and Data

The user describes the fan by inputting design values for the design pressure increase across the fan, the design volumetric flow rate, the fan total efficiency and the fan motor efficiency. The user also needs to specify the fraction of the fan waste heat that will enter the air stream (usually 0 or 1). For variable speed fans the user must also enter the coefficients of a 4th order polynomial that relates the fan mass flow rate to the fan power consumption. The dependent variable is the mass flow fraction, the independent variable is the fan power part load ratio. For multi-speed fans the user must enter a fan power ratio as a function of speed ratio performance curve name. Multi-speed fans can only be simulated in a parent object which allows multiple fan speeds (e.g., AirLoopHVAC:Unitary:Furnace:HeatCool, ZoneHVAC:PackagedTerminalAirConditioner, etc.). An optional efficiency curve may also be used when simulating multi-speed fans to adjust the fan total efficiency as the fan speed changes.

Control

The model must decide whether the fan is on or off. The primary on/off trigger is the fan schedule. This is an on/off schedule associated with each fan: a value of 1 indicates the fan is on; a value of 0 indicates the fan is off. The fan schedule can be overruled by flags set by system availability managers. If the flag *TurnFansOn* is true, a zero fan schedule value will be overridden and the fan will be turned on. If the flag *TurnFansOff* is true the fan will be forced off. The inlet air mass flow rate must be greater than zero for the fan to be on.

Generally the fan is a passive component: it accepts the mass flow rate on its inlet node, uses it in its calculations of energy consumption and temperature rise, and passes it to the outlet node. However the fan maximum and minimum air flow rates act as absolute limits on the air stream flow rate.

For multi-speed fans, the parent object determines the fan speed ratio (i.e., the selected speed of the fan motor) and uses this value in conjunction with the fan power ratio performance curve to calculate the full load fan power. This full load fan power is then used to determine the part-load performance of the fan and motor assembly.

Simulation

The following equations define the model for the simple (single speed) fan.

$$\dot{Q}_{tot} = \dot{m} \cdot \Delta P / (e_{tot} \cdot \rho_{air})$$

$$\dot{Q}_{shaft} = e_{motor} \cdot \dot{Q}_{tot}$$

$$\dot{Q}_{toair} = \dot{Q}_{shaft} + (\dot{Q}_{tot} - \dot{Q}_{shaft}) \cdot f_{motortoair}$$

$$h_{out} = h_{in} + \dot{Q}_{toair} / \dot{m}$$

$$w_{out} = w_{in}$$

$$T_{out} = PsyTdbFnHW(h_{out}, w_{out})$$

The on/off fan model is similar to the simple fan model with the exception that the on/off fan may cycle on and off during a simulation time step. The cycling rate of the fan is known as the run time fraction. The calculation of run time fraction accounts for the part-load losses of other equipment used in the HVAC system. A part-load factor (a.k.a. part-load ratio) is first calculated for the fan as the ratio of the actual operating mass flow rate to the maximum fan mass flow rate. The run time fraction is then calculated as the part-load factor divided by the part-load fraction. The part-load fraction is determined by other HVAC equipment in the simulation (Ref. DX coil) for use by this specific fan model.

$$f_{flow} = \frac{\dot{m}}{\dot{m}_{max}}$$

$$RTF = f_{flow} / PLF$$

The total fan power is then calculated as the maximum fan power multiplied by the run time fraction.

$$\dot{Q}_{tot} = RTF \left[\dot{m} \cdot \Delta P / (e_{tot} \cdot \rho_{air}) \right]$$

The remaining calculations are the same as those described in the simple single-speed fan mode.

The model used to simulate a multi-speed fan relies on the general fan laws to correct for speed changes in the fan's motor and the corresponding change in fan power and fan total efficiency. Two performance curves are used to define the change in power and efficiency as shown below. The power ratio curve must be used to simulate a multi-speed fan. The power ratio curve modifies the fan power based on a change in fan speed according to the cubic fan law. Exponents other than 3 are allowed. The efficiency ratio curve is used to correct for changes in nominal efficiency at alternate fan speeds. If either or both of these curves are not provided, the ratio is assumed to be 1.

The power ratio term is evaluated using an exponent performance curve. The form of the exponent curve equation is shown below.

$$\text{ExponentCurveOutput} = C1 + C2 * x^{C3}$$

The exponent performance curve is used to evaluate the power ratio as a function of speed ratio (x in the equation above). For typical fan performance, coefficient $C1 = 0$ and coefficient $C2 = 1$. The exponent ($C3$) is typically equal to 3 (cubed fan law) but other values are allowed.

$$\text{PowerRatio} = f(N_{ratio})^n = \text{ExponentCurveOutput}$$

The efficiency ratio curve is a quadratic or cubic curve used to evaluate the efficiency ratio as a function of speed ratio (x in the following equation). The cubic equation form is shown below.

$$\text{EfficiencyCurveOutput} = C1 + C2(x)^2 + C3(x)^3$$

The curve coefficients are determined by correlating the normalized fan total efficiency (i.e., the actual fan total efficiency divided by the nominal fan total efficiency at a speed ratio of 1) to the speed ratio (i.e., the ratio of actual air flow rate through the fan to the maximum fan air flow rate).

$$\text{EfficiencyRatio} = f(N_{ratio}) = \text{EfficiencyCurveOutput}$$

To account for the change in fan speed and to ensure that the run time fraction is accurately calculated, an adjustment to the previous calculation for run time fraction is made. This adjusts the calculated run time fraction to account for times when the fan speed has been reduced and the new speed is now considered the fan's maximum speed. The speed adjusted run time fraction is then used in the fan power calculation.

$$RTF = RTF / N_{ratio}$$

$$\dot{Q}_{tot} = RTF \left[\frac{\dot{m} \Delta P}{e_{tot} \rho_{air}} \right] \left(\frac{\text{PowerRatio}}{\text{EfficiencyRatio}} \right)$$

Each of the performance curves described above may be used to model the performance of a multi-speed fan motor, however, the power ratio curve must be used to invoke the multi-speed simulation. These curves are used when the fan is used in an HVAC system having

multiple flow rates (i.e., different flow rates in cooling and heating mode). If an HVAC system operates at the same speed in either cooling or heating mode, these curves are not required. When these curves are not used, the associated ratio term in the equation above is assumed to be 1. The remaining calculations are identical to the simple single-speed fan model described above.

The model for the variable speed fan is similar to the simple single-speed fan model except for a part load factor that multiplies the fan power consumption.

$$f_{flow} = \dot{m} / \dot{m}_{design}$$

$$f_{pl} = c_1 + c_2 \cdot f_{flow} + c_3 \cdot f_{flow}^2 + c_4 \cdot f_{flow}^3 + c_5 \cdot f_{flow}^4$$

$$\dot{Q}_{tot} = f_{pl} \cdot \dot{m}_{design} \cdot \Delta P / (e_{tot} \cdot \rho_{air})$$

The rest of the calculation is the same as for the simple fan.

Here

\dot{Q}_{tot} is the fan power in watts;

\dot{m} is the air mass flow in kg/s;

\dot{m}_{design} is the design (maximum) air flow in kg/s;

ΔP is the fan design pressure increase in Pascals;

e_{tot} is the fan total efficiency;

ρ_{air} is the air density at standard conditions in kg/m³;

e_{motor} is the motor efficiency;

\dot{Q}_{shaft} is the fan shaft power in watts;

\dot{Q}_{toair} is the power entering the air in watts;

N_{ratio} is the ratio of actual fan flow rate (or speed) to maximum fan flow rate (or speed)

h_{in}, h_{out} are the inlet and outlet air stream specific enthalpies in J/kg;

w_{in}, w_{out} are the inlet and outlet air stream humidity ratios;

T_{out} is the outlet air temperature in degrees C;

$PsyTdbFnHW$ is the EnergyPlus psychrometric routine relating enthalpy and humidity ratio to temperature;

f_{flow} is the flow fraction or part-load ratio;

f_{pl} is the part load factor.

References

ASHRAE. 1993. HVAC 2 Toolkit: A Toolkit for Secondary HVAC System Energy Calculations. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
 Selection and Application of Multispeed Motors, Heredos, Francis P., IEEE Transactions on Industry Applications, Vol. 1A-23, No. 2, March/April 1987.

Air System Compound Component Groups

Forced-Air Furnace and Central Air Conditioning

Overview

The input objects `AirLoopHVAC:Unitary:Furnace:HeatOnly` and `AirLoopHVAC:Unitary:Furnace:HeatCool` provide a “virtual” component that collect and control a set of components: an on/off or constant volume fan component and a gas or electric heating coil component. If the HeatCool version is selected, then a DX cooling coil is also modeled as part of the system as shown in Figure 168 below. For the HeatCool version, an optional reheat coil may also be modeled for controlling high zone humidity levels and the furnace’s configuration when specifying this option is shown in Figure 169 below. The unit may be configured to have either a blow through or draw through fan. If a blow through fan configuration is specified, the furnace fan is placed before the heating coil as shown in the figures below. If a draw through fan configuration is specified, the fan is placed after the heating coil for the HeatOnly version, or after the cooling coil for the HeatCool version.

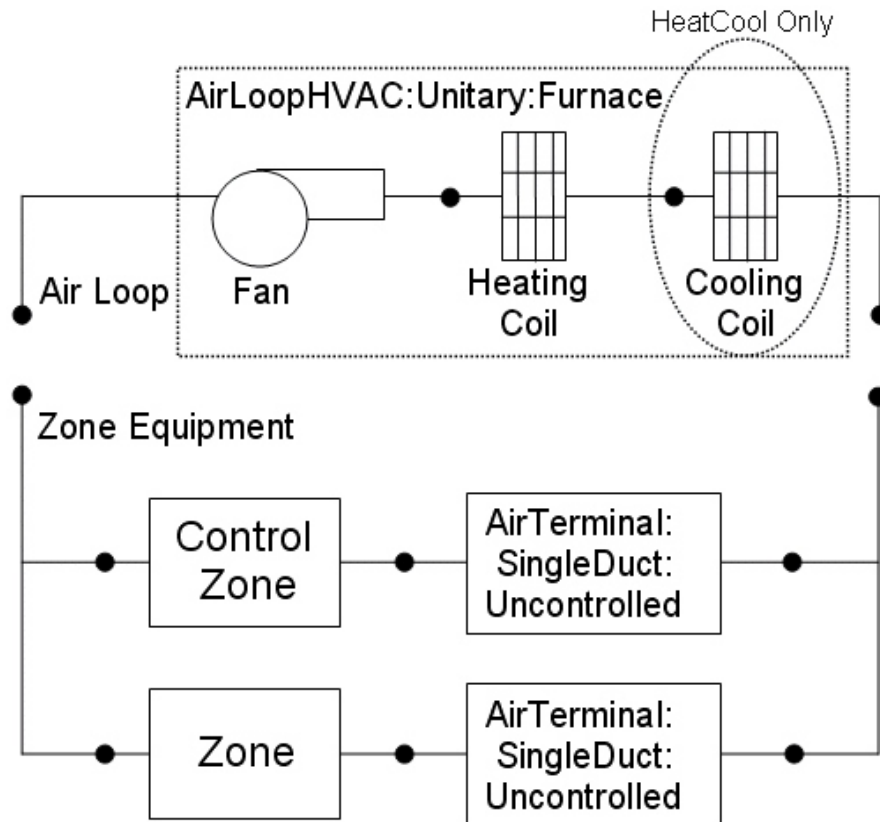


Figure 168. Schematic of the EnergyPlus Furnace (Blow Through Configuration)

While the furnace may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single “control” zone. One of the key input parameters for the furnace component is the fraction of the total system volumetric air flow that goes through the controlling zone. The furnace module scales the calculated load for the control zone upward based on this fraction to determine the total load to be met by the furnace. The module then proceeds to calculate the required part-load ratio for the system coil and the supply air fan to meet this total load. The heating or cooling capacity delivered by the furnace is distributed to all of the zones served by this system via the direct air units that supply air to each zone.

The furnace component is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). Fan:OnOff must be used to model AUTO fan, while Fan:OnOff or Fan:ConstantVolume can be used to model fan ON. The fan operation mode is specified using a supply air fan operating mode schedule where schedule values of 0 denote cycling fan operation and schedule values other than 0 (a 1 is usually used) denote continuous fan operation. Using this schedule, the furnace fan may be cycled with cooling or heating coil operation or operated continuously based on time of day (e.g., cycling fan operation at night and continuous fan operation during the daytime). The fan operating mode schedule specified here overrides the Supply Air Fan Mode Operation input in the DX coil object (Ref. Coil:Cooling:DX:SingleSpeed). If the fan operating mode schedule name field is left blank in the furnace object, the furnace assumes cycling or AUTO fan mode operation throughout the simulation.

The only output variables reported by the furnace object are the fan part-load ratio and the compressor part-load ratio (HeatCool only). The fan part-load ratio is defined as the actual air mass flow rate through the system for the time step divided by the design supply air mass flow rate specified for the furnace ($\dot{m}_{actual} / \dot{m}_{design}$). The furnace operates based on the user-specified (or autosized) design supply air flow rate(s). The ‘design’ supply air mass flow rate may be different for cooling, heating, and when no cooling or heating is required and the fan operates continuously based on user-specified inputs (HeatCool only). For the HeatCool version, the compressor part-load ratio is reported as the ratio of the actual cooling load to the full-load sensible capacity (see Eqn. (431)). Reporting of other variables of interest for the furnace (heating rate, cooling rate, energy consumption, etc.) is done by the individual system components (fan, heating coil and DX cooling coil).

Model Description

As described previously, the furnace is a “virtual” component consisting of a fan, heating coil and, for the HeatCool version, a cooling coil with an optional reheat coil. The sole purpose of the furnace model is to properly coordinate the operation of the various system components. The following sections describe the flow of information within the model for both the HeatOnly and HeatCool configurations, as well as the differences between cycling and continuous supply air fan operation. The last section describes the optional control of high zone humidity with a reheat coil for the HeatCool configuration.

HeatOnly Configuration

The HeatOnly configuration consists of an on/off or constant volume fan and an electric or gas heating coil. When the model is first called during an EnergyPlus simulation, all of the input data specified for each furnace in the input data file are read into data structures for use throughout the remainder of the simulation.

For each simulation time step when the performance of a heat-only furnace is being modeled, the first step is to retrieve the heating load required to meet the thermostat setpoint for the “control” zone (see Figure 168. Schematic of the EnergyPlus Furnace). See the section “Summary of Predictor-Corrector Procedure” elsewhere in this document for more details regarding load calculations. Since the furnace may be specified to serve several zones but controlled based on the load calculated for the “control” zone, the total heating load to be met by the furnace is determined from the following equation:

$$\text{Furnace Heating Load} = \frac{\text{Control Zone Heating Load}}{\text{Control Zone Air Flow Fraction}} \quad (423)$$

The model then calculates the furnace’s sensible heating energy rate delivered to the zones being served when the system runs at full-load conditions and when the heating coil is OFF. If the supply air fan cycles on and off with the heater, then the sensible heating energy rate is zero when the heating coil is OFF. However if the fan is scheduled to run continuously

regardless of coil operation, then the sensible heating energy rate will not be zero when the heating coil is OFF. Calculating the sensible heating energy rate involves modeling the supply air fan (and associated fan heat) and the heating coil. For each of these cases (full load and heating coil OFF), the sensible heating energy rate delivered by the furnace is calculated as follows:

$$Full\ Heat\ Output = (Mass\ Flow\ Rate_{full\ load}) (h_{out,full\ load} - h_{control\ zone})_{HR\ min} \quad (424)$$

$$No\ Heat\ Output = (Mass\ Flow\ Rate_{coil\ off}) (h_{out,coil\ off} - h_{control\ zone})_{HR\ min} \quad (425)$$

where:

$Mass\ Flow\ Rate_{full\ load}$ = air mass flow rate through furnace at full-load conditions, kg/s

$h_{out,full\ load}$ = enthalpy of air exiting the furnace at full-load conditions, J/kg

$h_{control\ zone}$ = enthalpy of air in the control zone (where thermostat is located), J/kg

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the furnace exiting air or the air in the control zone

$Mass\ Flow\ Rate_{coil\ off}$ = air mass flow rate through the furnace with the heating coil OFF, kg/s

$h_{out,coil\ off}$ = enthalpy of air exiting the furnace with the heating coil OFF, J/kg

With the calculated sensible heating energy rates and the total sensible heating load to be met by the system, the part-load ratio for the furnace is estimated.

$$PartLoadRatio = MAX \left(0.0, MIN \left(1.0, \frac{ABS(FurnaceHeatingLoad - NoHeatOutput)}{ABS(FullHeatOutput - NoHeatOutput)} \right) \right) \quad (426)$$

The part-load ratio calculated above is used to determine the required heating coil capacity as $Q_{heating\ coil} = Q_{design} * PartLoadRatio$ where Q_{design} is the nominal heating coil capacity as specified in the heating coil object. If the fan cycles on and off with the heating coil (i.e., when the supply air fan operating mode schedule values are equal to 0), then this part-load ratio is also used to determine the operating mass flow rate of the furnace as

$\dot{m}_{furnace} = \dot{m}_{design} * PartLoadRatio$. If the fan operates continuously (i.e. fan ON), the operating mass flow rate is specified as \dot{m}_{design} . The furnace's fan and heating coil are then re-simulated to determine the furnace's delivered sensible heating capacity at the above calculated part-load ratio.

$$Q_{furnace} = (\dot{m}_{furnace}) (h_{out,actual} - h_{control\ zone})_{HR\ min} \quad (427)$$

where:

$Q_{furnace}$ = sensible heating capacity delivered by the furnace (W)

$\dot{m}_{furnace}$ = air mass flow rate through the furnace (kg/s)

$h_{out,actual}$ = enthalpy of air exiting the furnace (J/kg)

$h_{out,control\ zone}$ = enthalpy of air in the control zone (J/kg)

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the furnace exiting air or the air in the control zone

Since the part-load performance of the heating coil can be non-linear, and the supply air fan heat varies based on heating coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the heating coil and fan are determined through iterative calculations (successive modeling of the heating coil and fan) until the furnace's heating output matches the heating load to be met within the heating convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the furnace's heating output divided by the load to be met.

$$Tolerance = \frac{(FurnaceHeatingLoad - Q_{furnace})}{FurnaceHeatingLoad} = 0.001$$

If the furnace has been specified with cycling fan/cycling coil (AUTO fan), then the furnace's design air mass flow rate is multiplied by PartLoadRatio to determine the average air mass flow rate for the system simulation time step. The air conditions at nodes downstream of the heating coil represent the full-load (steady-state) values when the coil is operating. If the supply air fan is specified to run continuously (fan ON), then the air mass flow rate remains at the furnace's design air mass flow rate. In this case, the air conditions at nodes downstream of the heating coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and inlet air conditions when the coil is OFF).

For the case where the furnace is scheduled to operate with continuous supply air fan operation, but no heating load is required to meet the setpoint temperature in the control zone, the supply air fan model is still called to determine the fan exiting air conditions. The heating coil model is also called, but for the case with no heating load the heating coil model simply passes the inlet air conditions and mass flow rate from its inlet node to its outlet node. The air exiting the heating coil is then sent to the direct air units for distribution to each zone served by the furnace, where the zone heat balance is performed to determine the resulting zone air conditions. The furnace exiting air conditions and energy consumption are calculated and reported by the individual component models (fan and heating coil).

HeatCool Configuration

The HeatCool configuration consists of an on/off or constant volume fan, an electric or gas heating coil, and a DX cooling coil. For the cases where a heating load is calculated for the control zone or no heating/cooling load is calculated for the control zone, the model follows nearly identical computational steps as described in the HeatOnly Configuration section above. The only difference is the air mass flow rate during no cooling/heating operation with continuous supply air fan operation can be different from the air mass flow rate during heater operation for the HeatCool configuration. If a cooling load is calculated by EnergyPlus for the control zone, the solution methodology is also virtually identical and is described here for completeness.

If EnergyPlus determines that the furnace must supply cooling to the control zone to meet the zone air temperature setpoint, then the model computes the total sensible cooling load to be met by the furnace based on the control zone sensible cooling load and the fraction of the furnace air flow that goes through the control zone.

$$FurnaceCoolingLoad = \frac{ControlZoneCoolingLoad}{ControlZoneAirFlowFraction} \quad (428)$$

If the supply air fan operating mode schedule requests cycling fan operation, the model first checks for the presence of an economizer in the outside air system serving the furnace's air loop (Ref. AirLoopHVAC:OutdoorAirSystem). If an outside air system is not present or if an air-side economizer is not used, the furnace's compressor is used to meet the furnace

cooling load. If an air-side economizer is used and is active (i.e., economizer controls indicate that conditions are favorable to increase the outside air flow rate), the furnace will try to meet the cooling load by operating only the supply air fan. If the fan is able to satisfy the furnace cooling load, the compressor remains off for the entire simulation time step. If the operation of the fan alone is unable to meet the entire cooling load, then the compressor is enabled and additional calculations are performed to determine the compressor's part-load ratio.

The model then calculates the furnace's sensible cooling energy rate delivered to the zones being served when the system runs at full-load conditions and when the DX cooling coil is OFF. If the supply air fan cycles with the compressor, then the sensible cooling energy rate is zero when the cooling coil is OFF. However if the fan is configured to run continuously regardless of coil operation, then the sensible cooling energy rate will not be zero when the cooling coil is OFF. Calculating the sensible cooling energy rate involves modeling the supply air fan (and associated fan heat), the heating coil (simply to pass the air properties and mass flow rate from its inlet node to its outlet node) and the DX cooling coil. For each of these cases (full load and DX cooling coil OFF), the sensible cooling energy rate delivered by the furnace is calculated as follows:

$$Full\ Cool\ Output = (Mass\ Flow\ Rate_{full\ load}) (h_{out,full\ load} - h_{control\ zone})_{HR\ min} \quad (429)$$

$$No\ Cool\ Output = (Mass\ Flow\ Rate_{coil\ off}) (h_{out,coil\ off} - h_{control\ zone})_{HR\ min} \quad (430)$$

where:

$Mass\ Flow\ Rate_{full\ load}$ = air mass flow rate through furnace at full-load conditions, kg/s

$h_{out,full\ load}$ = enthalpy of air exiting the furnace at full-load conditions, J/kg

$h_{control\ zone}$ = enthalpy of air in the control zone (where thermostat is located), J/kg

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the furnace exiting air or the air in the control zone

$Mass\ Flow\ Rate_{coil\ off}$ = air mass flow rate through the furnace with the cooling coil OFF, kg/s

$h_{out,coil\ off}$ = enthalpy of air exiting the furnace with the cooling coil OFF, J/kg

With the calculated sensible cooling energy rates and the total sensible cooling load to be met by the system, the part-load ratio for the furnace is estimated.

$$PartLoadRatio = MAX \left(0.0, \frac{ABS(Furnace\ Cooling\ Load - NoCoolOutput)}{ABS(FullCoolOutput - NoCoolOutput)} \right) \quad (431)$$

Since the part-load performance of the DX cooling coil is frequently non-linear (Ref: DX Cooling Coil Model), and the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the cooling coil and fan) until the furnace's cooling output matches the cooling load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the furnace's cooling output divided by the load to be met.

$$Tolerance = \frac{(Furnace\ Cooling\ Load - Q_{furnace})}{Furnace\ Cooling\ Load} = 0.001$$

where:

$Q_{furnace}$ = Furnace delivered sensible capacity (W)

If the furnace has been specified with cycling fan/cycling coil (AUTO fan) and high humidity control *has not* been specified, then the furnace's operating supply air mass flow rate is multiplied by PartLoadRatio to determine the average air mass flow rate for the system simulation time step. The air conditions at nodes downstream of the cooling coil represent the full-load (steady-state) values when the coil is operating. If a cycling fan is used and high humidity control *has* been specified, the calculation of average air mass flow rate is based on the greater of the heating or cooling part-load ratio (see following section on high humidity control). When the heating part-load ratio is greater than the cooling part-load ratio, the air conditions at nodes downstream of the cooling coil represent the weighted average of full-load conditions when the cooling coil is operating and inlet air conditions when the cooling coil is off (i.e., the fan continues to operate due to a heating requirement where the heating PLR is greater than the cooling PLR). If the supply air fan is specified to run continuously (fan ON), then the air mass flow rate continues to operate at the user-specified supply air mass flow rate when no cooling or heating is required. In this case, the air conditions at nodes downstream of the cooling coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and inlet air conditions when the coil is OFF). The furnace exiting air conditions and energy consumption are calculated and reported by the individual component models (fan and DX cooling coil).

High Humidity Control with HeatCool Configuration

An optional reheat coil can be specified with the HeatCool configuration to allow the furnace to control high zone humidity levels. The specific configuration of the HeatCool Furnace with high humidity control option is shown in Figure 169. The figure below shows the fan placement when a blow through fan is specified. If a draw through fan is specified, the fan is located between the heating coil and the reheat coil. The system is controlled to keep the relative humidity in the control zone from exceeding the setpoint specified in the object ZoneControl:Humidistat. This option is available when the supply air fan operates continuously (i.e., the supply air fan operating mode schedule values are never equal to 0) or the supply air fan cycles with the compressor. For the case of cycling fan mode when the high humidity control is specified as CoolReheat and the heating part-load ratio is greater than the compressor part-load ratio, the air conditions at nodes downstream of the cooling coil are calculated as the average conditions during the duration of fan operation (i.e., the weighted average of full load conditions when the coil is operating and inlet air conditions when the coil is off and the fan continues to operate to meet the heating load). For the case where cycling fan mode is used and the heating part-load ratio is less than the compressor part-load ratio, the air conditions at the nodes downstream of the cooling coil are simply the full load (steady-state) values when the coil is operating. In addition, when high humidity control is specified and the compressor operates, the furnace operates at the cooling air flow rate when a zone heating load is present as determined by the zone thermostat.

High humidity control is specified as either None, MultiMode, or CoolReheat in the Dehumidification Control Type input field. MultiMode is specified when a heat exchanger is used to improve the dehumidification performance of the cooling coil. The heat exchanger will be activated when the sensible part-load ratio is insufficient to meet the zone latent load. CoolReheat is specified when a DX cooling coil is used to over-cool the supply air stream in order to meet the zone latent load. When a heat exchanger is used in conjunction with a DX cooling coil and CoolReheat is specified as the Dehumidification Control Type, the heat exchanger is "locked on" to meet either the sensible or latent cooling load. If the dehumidification control type is selected as None and a heat exchanger assisted cooling coil is used, the heat exchanger is "locked on" and the air conditioner runs only to meet the sensible cooling load.

The model first calculates the PartLoadRatio required to meet the sensible cooling load as described above (see Eqn. (431)) to maintain the dry-bulb temperature setpoint in the control zone. If a moisture (latent) load exists because the control zone humidity has exceeded the

setpoint, the total moisture load to be met by the HeatCool furnace (SystemMoistureLoad) is calculated based on the control zone moisture load and the control zone air flow fraction. The model then calculates the LatentPartLoadRatio required to meet the humidistat setpoint.

$$\text{SystemMoistureLoad} = \frac{\text{Control Zone Moisture Load}}{\text{Control Zone Air Flow Fraction}} \quad (432)$$

$$\text{LatentPartLoadRatio} = \text{MAX} \left(\text{MinPLR}, \frac{\text{ABS}(\text{SystemMoistureLoad} - \text{NoLatentOutput})}{\text{ABS}(\text{FullLatentOutput} - \text{NoLatentOutput})} \right) \quad (433)$$

where:

FullLatentOutput = the furnace's latent cooling energy rate at full-load conditions, W

NoLatentOutput = the furnace's latent cooling energy rate with the cooling coil OFF, W

MinPLR = the minimum part-load ratio, which is usually 0.0. For the case when the latent capacity degradation model is used (Ref: DX Cooling Coil Model), this value is the minimum part-load ratio at which the cooling coil will dehumidify the air.

The model uses the greater of the two part-load ratios, PartLoadRatio or LatentPartLoadRatio, to determine the operating part-load ratio of the furnace's DX cooling coil. As previously described, iterations are performed to converge on the solution within the convergence tolerance.

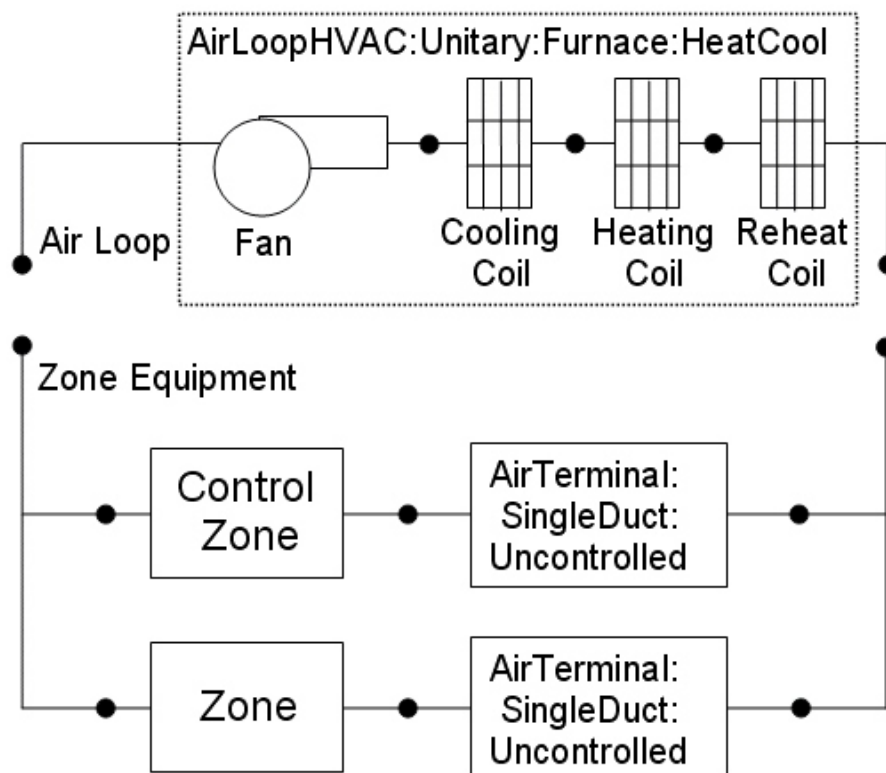


Figure 169. Schematic for Blow Through Furnace with High Humidity Control

If the model determines that the LatentPartLoadRatio is to be used as the operating part-load ratio of the furnace's cooling coil, the reheat coil is used to offset the excess sensible capacity provided by the unit. The model first checks the sensible load that exists for the current simulation time step (predicted zone temperature with no HVAC operation compared to the

thermostat setpoint temperatures). If a sensible cooling load or no sensible cooling or heating load exists (Figure 170), the model calculates the difference between the sensible heating load required to reach or maintain the heating dry-bulb temperature setpoint and the actual sensible cooling energy rate delivered by the unit (with LatentPartLoadRatio). In this case, the reheat coil is used to offset the excess sensible cooling energy provided by the DX cooling coil (if any) that could have caused an overshoot of the heating dry-bulb temperature setpoint. Note that when a humidistat is used and high humidity control is required, the zone dry-bulb temperature will typically move toward the heating temperature setpoint. If a heating load exists (Figure 171), the reheat coil is used to offset the entire sensible cooling energy rate of the DX cooling coil (to meet the humidistat setpoint) and the heating coil is used to meet the entire heating load as described in the HeatOnly configuration section above. Note that when a heating load exists and high humidity control is required, the furnace operates at the user-specified cooling air flow rate for the entire simulation time step. As with the fan, DX cooling coil, and heating coil, report variables associated with reheat coil performance (e.g., heating coil energy, heating coil rate, heating coil gas or electric consumption, heating coil runtime fraction, etc.) are managed in the reheat (heating) coil object.

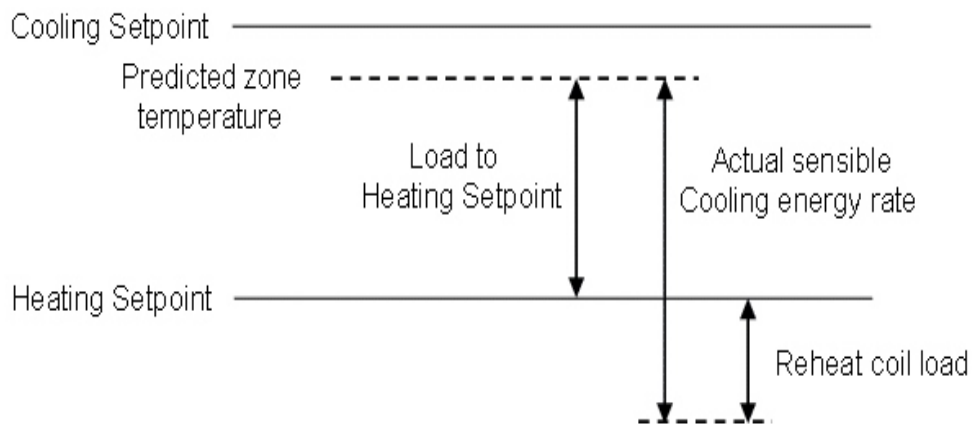


Figure 170. Reheat Coil Load when Predicted Zone Temperature is Above Heating Setpoint

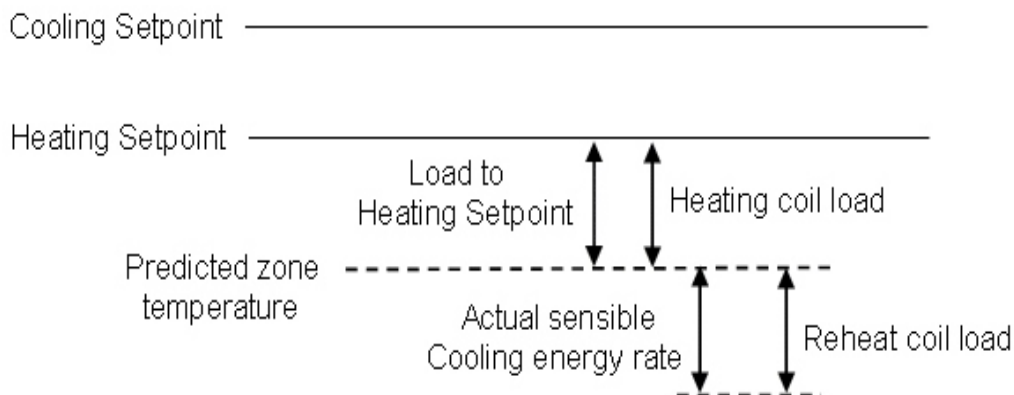


Figure 171. Reheat Coil Load when Predicted Zone Temperature is Below Heating Setpoint

Unitary Systems

The input objects `AirLoopHVAC:UnitaryHeatCool` and `AirLoopHVAC:UnitaryHeatOnly` provides models that are identical to the Furnace and Central Air models described above. Please reference the previous section for details.

Unitary System with Changeover-Bypass-Variable Air Volume

Overview

The input object `AirLoopHVAC:UnitaryHeatCool:VAVChangeoverBypass` provides a model for a changeover-bypass variable air volume (CBVAV) unitary system that is a compound object made up of other components. Each CBVAV system consists of an outside air mixer, direct expansion (DX) cooling coil, heating coil, and a supply air fan as shown in the figures below. Zone thermostats and terminal units are required in each zone served by this system. The terminal units are specific to this system type and are either `AirTerminal:SingleDuct:VAV:HeatAndCool:Reheat` or `AirTerminal:SingleDuct:VAV:HeatAndCool:NoReheat`. A zone humidistat and single zone max humidity setpoint manager may also be specified to help control high humidity levels. These individual components are described elsewhere in this document. The CBVAV unitary system object coordinates the operation of these components and is modeled as a type of air loop equipment (Ref. Air Primary Loop).

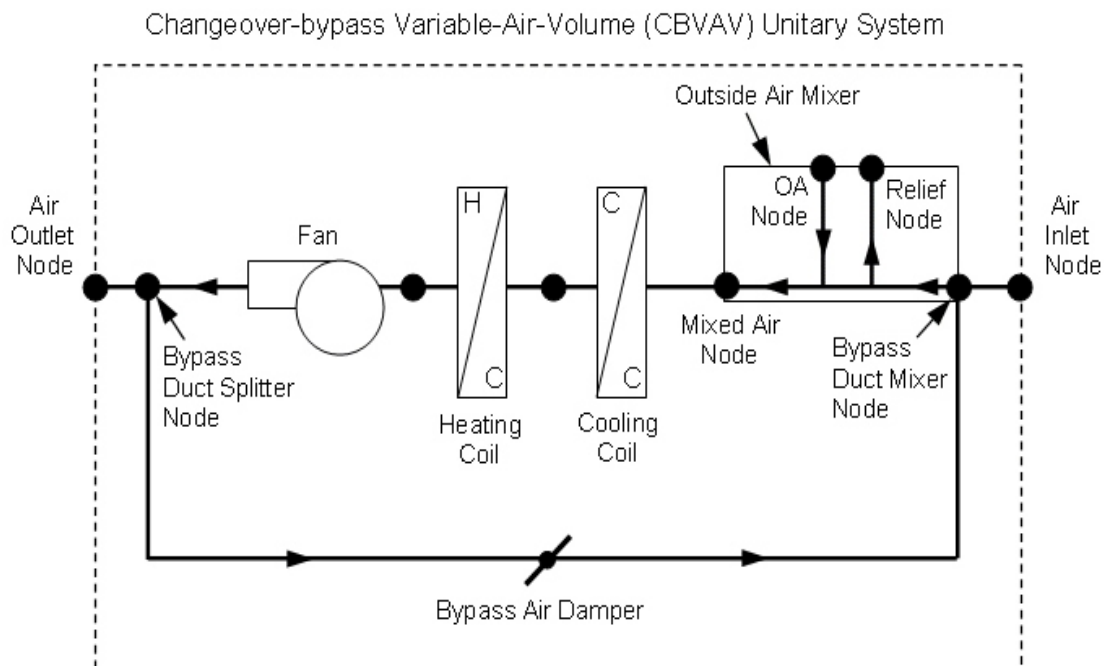


Figure 172. Schematic of a Changeover Bypass VAV Unitary System with Draw Through Fan

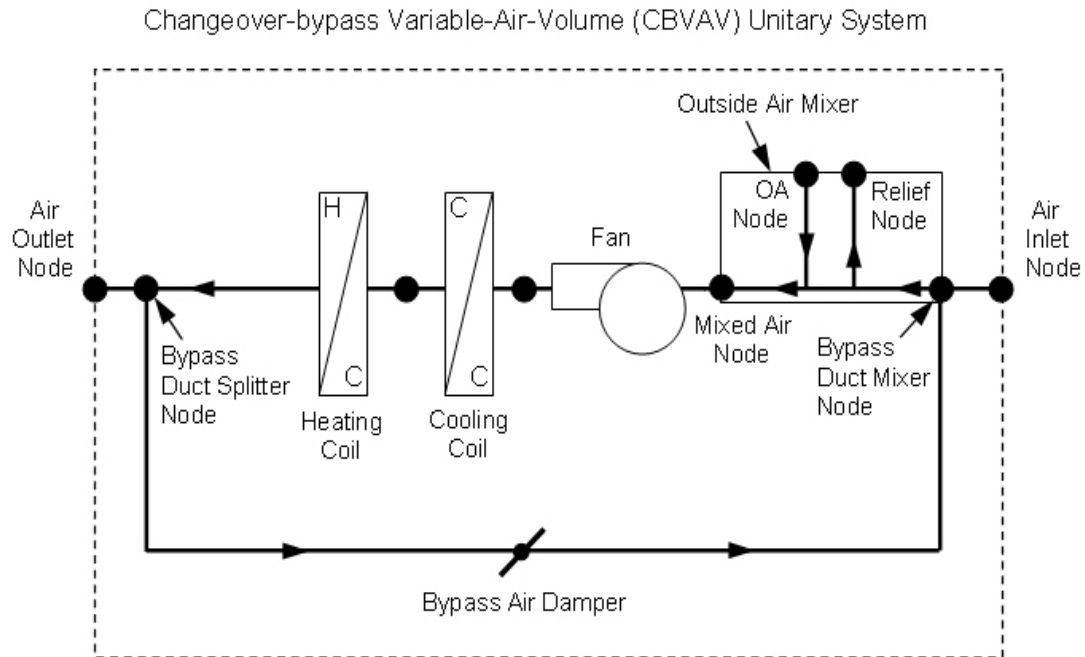


Figure 2.

Figure 173. Schematic of Changeover Bypass VAV Unitary System with Blow Through Fan

The CBVAV unitary system conditions one or more zones and is controlled by thermostats located in each zone (the use of a single humidistat is also allowed when using multi-mode DX cooling coils). The CBVAV system operates to meet the zone sensible cooling or sensible heating requirements as dictated by the thermostat schedule(s). The priority control input determines the mode of operation and is specified as Cooling Priority, Heating Priority, or Zone Priority. If Cooling Priority is specified, the system operates to meet the cooling load when any zone served by this system (air loop) requires cooling. If Heating Priority is specified, the system operates to meet the heating load when any zone requires heating. If Zone Priority is specified, the system operates based on the maximum number of zones requiring either heating or cooling.

Once the operating mode is determined, the CBVAV model calculates a target supply air temperature required to operate a single terminal unit at its maximum air flow rate. The remaining terminal units will modulate as required to maintain the dry-bulb temperature in the zone they are serving according to the thermostat schedule for their respective zone. The system air flow rate (through the supply air fan, cooling coil and heat coil) remains constant during cooling operation, heating operation, and no cooling/heating mode as specified by the user. Therefore, as the zone terminal units modulate to reduce zone air flow rates, the excess system air flow is “bypassed” from the bypass duct splitter node back to the bypass duct mixer node of the CBVAV system (see figures above).

The CBVAV system is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). A Fan:OnOff or Fan:ConstantVolume is used in either case. Since the excess system air flow is bypassed while the zone terminal units modulate, the supply air fan operates the entire simulation time step when heating or cooling is required. For this reason, AUTO fan only allows the supply air fan to turn off when no cooling or heating is required. If fan ON is specified, the supply air fan runs the entire time the system is scheduled to operate (via its availability schedule). The mode of operation for the supply air fan is specified through a fan operating mode schedule where a value of 0 in the schedule indicates cycling fan mode and a value greater than 0

indicates continuous fan mode. If the schedule is not provided, the supply air fan operating mode is considered to be continuous (fan ON).

Output variables reported for the CBVAV system include the supply air fan part-load ratio, the compressor part-load ratio, and the electric consumption of the CBVAV system. Additional output variables report the total heating rate and total cooling rate provided by the CBVAV system. The sensible and latent components for total system cooling and heating are also available as output variables. Reporting of other variables of interest for the CBVAV system (DX coil cooling rate, heating rate, crankcase heater power, supply air fan power, etc.) is done by the individual system components (fan, DX cooling coil, and heating coil).

Model Description

As described previously, the CBVAV system conditions one or more zones and is controlled by zone thermostats (Ref. ZoneControl:Thermostat). For each simulation time step, EnergyPlus performs a zone air heat balance to determine if cooling or heating is required to meet the zone thermostat setpoints, excluding any impacts from CBVAV system operation. CBVAV system performance is then modeled with all heating/cooling coils off but with the supply air fan operating according to the user specified supply air fan operating mode schedule. If the zone air heat balance plus the impact of CBVAV system operation with coils off results in no requirement for heating or cooling by the CBVAV coils, or if the CBVAV system is scheduled off (via its availability schedule), then the CBVAV coils do not operate and the compressor part-load ratio output variable is set to 0. If the model determines that cooling or heating is required and the CBVAV system is scheduled to operate, the model calculates a target supply air temperature required to maintain a single terminal unit at its maximum air flow rate, the amount of bypass air returning to the bypass duct mixer node, and the part-load ratio of the cooling or heating coils in order to meet the target supply air temperature.

When high humidity control is desired, a ZoneControl:Humidistat input object and a SetpointManager:SingleZone:Humidity:Maximum object are required. The air outlet node of the CBVAV system is used as the control node for the setpoint manager. For this reason, only a single humidistat should be specified for one of the zones being served by this system. If humidistat/setpoint manager objects are specified for more than one zone served by this system, only the last of the SetpointManager:SingleZone:Humidity:Maximum objects is used (the setpoint manager objects are read in order of occurrence in the input file). Since the outlet node of the CBVAV system is always used as the control node, the setpoints calculated by the previous setpoint managers are overwritten by the last setpoint manager in the input. When a heat exchanger is used to improve the dehumidification performance of the cooling coil (i.e. CoilSystem:Cooling:DX:HeatExchangerAssisted), the heat exchanger is always active and can not be turned on and off based on zone humidity levels.

The remainder of this section describes the calculations performed when cooling or heating coil operation is required. For any HVAC simulation time step, the CBVAV system can only cool or heat the air, not both. Because the CBVAV system bypasses system air flow back to the inlet of the CBVAV unit, the system operates for the entire simulation time step. If the user specifies continuous fan operation, then the supply air fan continues to operate at a user-specified flow rate even during periods when the coils cycle off. If the user specifies AUTO fan operation, then the supply air fan cycles off for the entire simulation time step only when no cooling or heating is required.

Operating Mode

The first step in modeling a CBVAV system is to obtain the cooling or heating load for each zone as calculated by EnergyPlus based on the zone thermostat setpoint temperature(s). The calculated loads for each zone are used to determine the total cooling and heating requirements for all zones served by this system. In addition to summing the zone cooling and heating loads, the number of zones in cooling and the number of zones in heating are totaled. The priority control mode specified by the user is then used to determine the operating mode for this simulation time step.

If Cooling Priority is specified and the total cooling requirement in all zones is not equal to zero, then cooling is selected as the operating mode. If the total cooling requirement is equal to zero and the total heating requirement is not equal to zero, then heating is selected as the operating mode. If the total cooling requirement and total heating requirement are equal to zero then the zones are allowed to float (no heating or cooling provided).

If Heating Priority is specified and the total heating requirement in all zones is not equal to zero, then heating is selected as the operating mode. If the total heating requirement is equal to zero and the total cooling requirement is not equal to zero, then cooling is selected as the operating mode. If the total cooling requirement and total heating requirement are equal to zero then the zones are allowed to float (no heating or cooling provided).

If Zone Priority is specified and the total number of zones requiring cooling is greater than the total number of zones requiring heating, then cooling is selected as the operating mode. If the total number of zones requiring heating is greater than the total number of zones requiring cooling, then heating is selected as the operating mode. If the total number of zones requiring cooling is equal to the total number of zones requiring heating, then the magnitude of the total cooling and heating requirements for all zones sets the operating mode. In this case, if the magnitudes of the cooling and heating requirements are zero, then the zones are allowed to float (no heating or cooling provided). If the magnitudes of the cooling and heating requirements are non-zero and identical, then cooling is selected as the operating mode.

Calculation of Bypass Duct Mixer Node Conditions

The operation of this system is unique in that it uses constant-air-volume equipment to provide variable air volume to the conditioned zones. This is accomplished with the use of a bypass duct (shown in the figures above) which shunts excess system air flow from the bypass duct splitter node at the system's outlet back to the bypass duct mixer node at the system's inlet. The air conditions at the bypass duct mixer node are based on the inlet air to the CBVAV system, the system outlet air conditions required to meet the zone loads, and the fraction of bypassed air. The following calculations are performed each simulation time step.

$$BypassFrac = 1.0 - \left(\frac{\dot{m}_{supply}}{\dot{m}_{system}} \right)$$

$$T_{bypassductmixernode} = (1 - BypassFrac)T_{inlet} + (BypassFrac)T_{outlet}$$

$$\omega_{bypassductmixernode} = (1 - BypassFrac)\omega_{inlet} + (BypassFrac)\omega_{outlet}$$

$$h_{bypassductmixernode} = PsyHFnTdbW(T_{mixerinlet}, \omega_{mixerinlet})$$

where:

$BypassFrac$ = fraction of system air directed to the bypass duct mixer node

\dot{m}_{supply} = Air mass flow rate at the system's air outlet node, kg/s

\dot{m}_{system} = Air mass flow rate through CBVAV system (coils and supply air fan), kg/s

$T_{bypassductmixernode}$ = Air temperature at the bypass duct mixer node, °C

T_{inlet} = Air inlet node temperature, °C

T_{outlet} = Air outlet node temperature, °C

$\omega_{bypassductmixernode}$ = Air humidity ratio at the bypass duct mixer node, kg/kg

ω_{inlet} = Air inlet node humidity ratio, kg/kg

ω_{outlet} = Air outlet node humidity ratio, kg/kg

$h_{bypassductmixernode}$ = Air enthalpy at the bypass duct mixer node, J/kg

$PsyHFnTdbW$ = Psychrometric function calculating enthalpy given dry-bulb temperature and humidity ratio

Cooling Operation

If the model determines that the CBVAV system must supply cooling to the zones, the model first calculates a target outlet air dry-bulb temperature required to operate a single terminal unit at its maximum air flow rate. The minimum target temperature for all zones is used to control the CBVAV system in cooling mode.

$$T_{target}^i = T_{zone}^i + \left(\frac{Q_{zone}^i}{\dot{m}_{zone,max}^i * C_p} \right); i = 1, n$$

$$T_{target} = MIN(T_{target}^i); i = 1, n$$

where:

T_{target}^i = Target supply air temperature for zone i , °C

T_{zone}^i = Air temperature in zone i , °C

Q_{zone}^i = Cooling or heating load for zone i (cooling loads are negative values, heating loads are positive values), W

C_p = Specific heat of supply air with coils off, J/kg-K

$\dot{m}_{zone,max}^i$ = Maximum terminal unit mass flow rate in zone i , kg/s

T_{target} = Target supply (outlet) air temperature for the CBVAV system, °C

The model then calculates the part-load ratio of the DX compressor required to meet the target supply (outlet) air temperature. Since the part-load performance of the DX cooling coil is frequently non-linear (Ref: Coil:Cooling:DX:SingleSpeed model), the actual part-load ratio for the cooling coil compressor is determined through iterative calculations (successive modeling of the DX cooling coil model) until the CBVAV system's outlet air temperature (including on/off cycling effects of the DX coil) matches the target supply (outlet) air temperature within a small temperature convergence tolerance (1E-5 °C).

Since the supply air fan operates continuously for each simulation time step (or is OFF for the entire HVAC system time step), the air conditions at nodes downstream of the cooling coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of coil's outlet node conditions when the coil is operating and the coil's inlet node conditions when the coil is OFF).

Heating Operation

Modeling the CBVAV system's DX heating coil is identical to the calculations described above for cooling operation except that the maximum target supply air temperature for all zones is used to control the CBVAV system in heating mode.

$$T_{target} = MAX(T_{target}^i); i = 1, n$$

Iterative calculations (successive modeling of the DX heating coil model) are used to determine the final heating coil part-load ratio to account for the non-linear performance of the DX heating coil at part-load conditions.

When a gas or electric heating coil is used instead of a DX heating coil, the amount of heat required by the coil is calculated based on the target supply (outlet) air temperature and the coil inlet air temperature as follows:

$$Q_{heating} = \dot{m}_{system} C_p (T_{target} - T_{inlet,heatcoil})$$

where:

$Q_{heating}$ = Heating coil load, W

C_p = Specific heat of heating coil inlet air, J/kg-K

$T_{inlet,heatcoil}$ = Heating coil inlet air temperature, °C

Floating Operation (no cooling or heating)

When the zone thermostats determine that there is no cooling or heating requirement in any zone, the CBVAV coils are OFF and the zone temperatures are allowed to float. However, a special case exists where the user specifies continuous supply air fan operation and no cooling or heating requirements exist in any zone. In this case, the outlet air temperature of the CBVAV system (with fan ON and coils OFF) is compared to the target outlet air temperatures required to maintain the zone's cooling and heating setpoint temperatures. If the outlet air temperature with the supply fan ON and coils OFF is below the target outlet air temperature required to maintain the heating setpoint, then heating mode is enabled to avoid overshooting the zone heating setpoint temperature. Conversely, if the outlet air temperature with the supply fan ON and coils OFF is above the target outlet air temperature required to maintain the cooling setpoint, then cooling mode is enabled to avoid overshooting the zone cooling setpoint temperature. This special case is handled at any time a no load condition is reported by the thermostats and overshooting the zone air temperature setpoint is possible regardless of the priority control mode selected by the user.

Minimum and Maximum Outlet Air Temperature in Cooling/Heating Operation

The user also specifies a minimum outlet air temperature during cooling operation and a maximum outlet air temperature during heating operation. The target outlet air temperature, as calculated above, is compared to each of these limits during each simulation time step. The resulting target outlet air temperature is used to control the heating and cooling coils.

$$T_{target} = MAX(T_{target}, T_{min,cooling})$$

$$T_{target} = MIN(T_{target}, T_{max,heating})$$

where:

$T_{min,cooling}$ = Minimum outlet air temperature during cooling operation, °C

$T_{max,heating}$ = Maximum outlet air temperature during heating operation, °C

Air Flow Calculations

The changeover-bypass VAV system operates based on user-specified (or autosized) air flow rates. The CBVAV system air flow rate (i.e., air flow through the supply air fan and heating/cooling coils) during cooling operation may be different than the system air flow rate during heating operation. In addition, the system air flow rate when no cooling or heating is required but the supply air fan remains ON can be different than the air flow rates when cooling or heating is required. The outside air flow rates can likewise be different in these various operating modes. The model takes these different air flow rates into account when simulating the performance of the CBVAV system. The system air flow rate for each simulation time step is based on the current operating mode (cooling, heating, or no cooling/heating) and is reported on the inlet/outlet air nodes of the various CBVAV components (e.g. fan, cooling coil, and heating coil). The supply air flow rate delivered to the individual zones is calculated based on the sum of the air flow rates through each terminal unit and is reported at the air inlet and outlet nodes of the CBVAV system each simulation time step. The difference between the system air mass flow rate and the supply air mass flow rate delivered to all zones is equal to the bypass air mass flow rate which is also reported each simulation time step.

The system and outside air flow rates when a system coil is ON (e.g., system air volumetric flow rate during cooling operation, system air volumetric flow rate during heating operation, outside air volumetric air flow rate during cooling operation, and outside air volumetric air flow rate during heating operation) or the system coils are OFF and the fan operates continuously (e.g., system air volumetric flow rate when no cooling or heating is needed and outside air volumetric air flow rate when no cooling or heating is needed) are specified by the user, or can be autosized, and are converted from volumetric flow rate to mass flow rate at standard conditions. If the user has specified cycling fan operation, then the system air and outside air mass flow rates when the system coils are OFF the entire time step are zero. If the user has specified constant fan operation, then the air flow rates when no cooling or heating is needed are used when the system coils are OFF the entire time step.

There is one special case. If the user has specified constant fan operation and they specify that the system air volumetric flow rate when no cooling or heating is needed is zero (or if the field is left blank), then the model assumes that the system air mass flow rate when the system coils are OFF the entire time step is equal to the corresponding air mass flow rate when any system coil was last operating (ON). This model handles the zero (or blank) entry for outside air volumetric flow rate when no cooling or heating is needed in an analogous fashion.

$$\dot{m}_{system} = \rho_{air, std} * \dot{V}_{mode}$$

$$\dot{m}_{system, OA} = \rho_{air, std} * \dot{V}_{mode, OA}$$

$$\dot{m}_{supply} = \sum_{i=1}^n \dot{m}_{zone}$$

$$\dot{m}_{bypass} = \dot{m}_{system} - \dot{m}_{supply}$$

where:

$\rho_{air, std}$ = standard air density (1.204 kg/m³) adjusted for the local barometric pressure (standard barometric pressure corrected for altitude, ASHRAE 1997 HOF pg. 6.1), kg/m³

\dot{V}_{mode} = User-specified system volumetric flow rate in cooling, heating, or no cooling or heating mode, m³/s

$\dot{m}_{system,OA}$ = Outdoor air mass flow rate introduced through the CBVAV system, kg/s

$\dot{V}_{mode,OA}$ = User-specified outdoor air volumetric flow rate in cooling, heating, or no cooling or heating mode, m³/s

Calculation of System Heating and Cooling Rates

At the end of each HVAC simulation time step, this object reports the heating or cooling rate and energy delivered by the system, as well as the electric power and consumption by the system. In terms of thermal energy delivered by the system, the sensible, latent and total energy transfer rates for the system are calculated as follows:

$$\dot{Q}_{total} = \left(\dot{m}_{supply} \right) (h_{outlet} - h_{inlet})$$

$$\dot{Q}_{sensible} = \left(\dot{m}_{supply} \right) (h_{outlet} - h_{inlet})_{HRmin}$$

$$\dot{Q}_{latent} = \dot{Q}_{total} - \dot{Q}_{sensible}$$

where:

\dot{Q}_{total} = total energy transfer rate by the system, W

$\dot{Q}_{sensible}$ = sensible energy transfer rate by the system, W

\dot{Q}_{latent} = latent energy transfer rate by the system, W

h_{inlet} = enthalpy of the air entering the unit at its inlet node, J/kg

h_{outlet} = enthalpy of the air leaving the unit at its outlet node, J/kg

HR_{min} = minimum of the inlet air and outlet air humidity ratio, kg/kg

Since each of these energy transfer rates can be calculated as positive or negative values, individual reporting variables are established for cooling and heating and only positive values are reported. The following calculations are representative of what is done for each of the energy transfer rates:

IF ($\dot{Q}_{total} < 0.0$) THEN

$$\dot{Q}_{total,cooling} = \text{ABS}(\dot{Q}_{total})$$

$$\dot{Q}_{total,heating} = 0.0$$

ELSE

$$\dot{Q}_{total,cooling} = 0.0$$

$$\dot{Q}_{total,heating} = \dot{Q}_{total}$$

where:

$$\dot{Q}_{total,cooling} = \text{output variable 'Changeover-bypass VAV Total Cooling Rate, W'}$$

$$\dot{Q}_{total,heating} = \text{output variable 'Changeover-bypass VAV Total Heating Rate, W'}$$

In addition to heating and cooling rates, the heating and cooling energy supplied by the system are also calculated for the time step being reported. The following example for total zone cooling energy is representative of what is done for the sensible and latent energy as well as the heating counterparts.

$$Q_{total,cooling} = \dot{Q}_{total,cooling} * TimeStepSys * 3600$$

where:

$$Q_{total,cooling} = \text{output variable 'Changeover-bypass VAV Total Cooling Energy, J'}$$

TimeStepSys = HVAC system simulation time step, hr

3600 = conversion factor, sec/hr

Changeover-bypass VAV Unitary System Sizing

The changeover-bypass VAV system is modeled as an air loop component and is sized according to the methodology described elsewhere in this document (Ref. System Design Loads and Air Flow Rates). The following volumetric air flow rates may be autosized:

- system air volumetric flow rate during cooling operation
- system air volumetric flow rate during heating operation
- system air volumetric flow rate when no cooling or heating is needed
- outside air volumetric air flow rate during cooling operation
- outside air volumetric air flow rate during heating operation
- outside air volumetric air flow rate when no cooling or heating is needed

Unitary Single Speed Air-To-Air Heat Pump

Overview

The input object AirLoopHVAC:UnitaryHeatPump:AirToAir provides a model for an air-to-air heat pump that is a “virtual” component that consists of an on/off or constant volume fan component, a DX cooling coil, a DX heating coil, and a gas or electric supplemental heating coil. The specific configuration of the blow through heat pump is shown in the following figure. For a draw through heat pump, the fan is located between the DX heating coil and the supplemental heating coil.

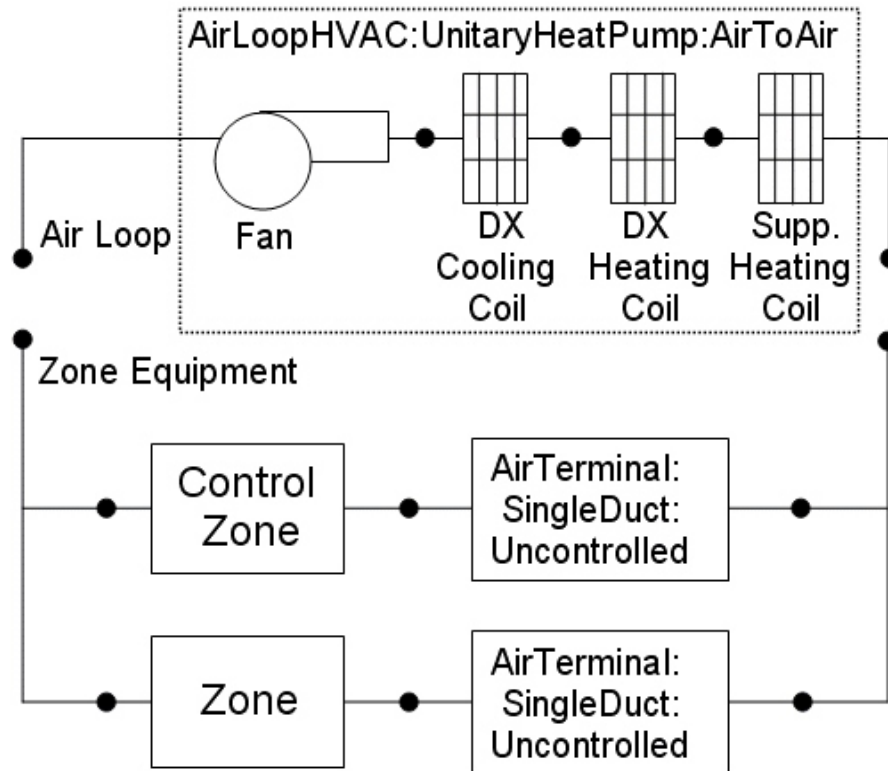


Figure 174. Schematic of a Blow Through Air-to-Air Heat Pump

While the heat pump may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single “control” zone. One of the key input parameters for the heat pump component is the fraction of the total system volumetric airflow that goes through the controlling zone. The heat pump module scales the calculated load for the control zone upward based on this fraction to determine the total load to be met by the heat pump. The module then proceeds to calculate the required part-load ratio for the system coil and the supply air fan to meet this total load. The heating or cooling capacity delivered by the heat pump is distributed to all of the zones served by this system via the direct air units that supply air to each zone.

The heat pump component is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). Fan:OnOff must be used to model AUTO fan, while Fan:OnOff or Fan:ConstantVolume can be used to model fan ON. The fan operation mode is specified using a supply air fan operating mode schedule where schedule values of 0 denote cycling fan operation and schedule values other than 0 (a 1 is usually used) denote continuous fan operation. Using this schedule, the furnace fan may be cycled with cooling or heating coil operation or operated continuously based on time of day (e.g. cycling fan operation at night and continuous fan operation during the day). The fan operating mode schedule specified here overrides the Supply Air Fan Mode Operation input in the DX coil object (Ref. Coil:Cooling:DX:SingleSpeed and Coil:Heating:DX:SingleSpeed). If the fan operating mode schedule name field is left blank in the heat pump object, the heat pump assumes cycling or AUTO fan mode operation throughout the simulation.

The output variables reported by the heat pump object are fan part-load ratio and compressor part-load ratio. Fan part-load ratio is defined as the actual air mass flow rate through the system for the time step divided by the operating supply air mass flow rate specified for the heat pump ($\dot{m}_{actual} / \dot{m}_{ON}$). The operating supply air mass flow rate may be different for cooling, heating, and when no cooling or heating is required and the fan operates continuously. Compressor part-load ratio is the actual load for the time step divided by the

full-load sensible capacity (see Eqn. (437) or Eqn.(441)). Reporting of other variables of interest for the heat pump (heating rate, cooling rate, energy consumption, etc.) is done by the individual system components (fan, DX cooling coil, DX heating coil, and supplemental heating coil).

Model Description

As described previously, the heat pump is a “virtual” component consisting of a fan, DX cooling coil, DX heating coil and a supplemental heating coil. The sole purpose of the heat pump model is to properly coordinate the operation of the various system components. The following sections describe the flow of information within the model, as well as the differences between cycling and continuous supply air fan operation.

Cooling Operation

If EnergyPlus determines that the heat pump must supply cooling to the control zone to meet the zone air temperature setpoint, then the heat pump model computes the total sensible cooling load to be delivered to the zones being served based on the control zone sensible cooling load and the fraction of the heat pump air flow that goes through the control zone.

$$\text{Heat Pump Cooling Load} = \frac{\text{Control Zone Cooling Load}}{\text{Control Zone Air Flow Fraction}} \quad (434)$$

If the supply air fan operating mode schedule requests cycling fan operation, the model first checks for the presence of an economizer in the outside air system serving the heat pump’s air loop (Ref. AirLoopHVAC:OutdoorAirSystem). If an outside air system is not present or if an air-side economizer is not used, the heat pump’s compressor is used to meet the heat pump cooling load. If an air-side economizer is used and is active (i.e., economizer controls indicate that conditions are favorable to increase the outside air flow rate), the heat pump will try to meet the cooling load by operating only the supply air fan. If the fan is able to satisfy the heat pump cooling load, the compressor remains off for the entire simulation time step. If the operation of the fan alone is unable to meet the entire cooling load, then the compressor is enabled and additional calculations are performed to determine the compressor’s part-load ratio.

The model then calculates the heat pump’s sensible cooling energy rate delivered to the zones being served when the system runs at full-load conditions and when the DX cooling coil is OFF. If the supply air fan cycles with the compressor, then the sensible cooling energy rate is zero when the cooling coil is OFF. However if the fan is scheduled to run continuously regardless of coil operation, then the sensible cooling energy rate will not be zero when the cooling coil is OFF. Calculating the sensible cooling energy rate involves modeling the supply air fan (and associated fan heat) and the DX cooling coil. The DX heating coil and the supplemental heating coil are also modeled, but only to pass the air properties and mass flow rate from their inlet nodes to their outlet nodes. For each of these cases (full load and DX cooling coil OFF), the sensible cooling energy rate delivered by the heat pump is calculated as follows:

$$\text{Full Cool Output} = \left(\text{Mass Flow Rate}_{\text{full load}} \right) \left(h_{\text{out,full load}} - h_{\text{control zone}} \right)_{\text{HR min}} \quad (435)$$

$$\text{No Cool Output} = \left(\text{Mass Flow Rate}_{\text{coil off}} \right) \left(h_{\text{out,coil off}} - h_{\text{control zone}} \right)_{\text{HR min}} \quad (436)$$

where:

$\text{Mass Flow Rate}_{\text{full load}}$ = air mass flow rate through heat pump at full-load conditions, kg/s

$h_{out, full\ load}$ = enthalpy of air exiting the heat pump at full-load conditions, J/kg

$h_{control\ zone}$ = enthalpy of air leaving the control zone (where thermostat is located), J/kg

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the heat pump exiting air or the air leaving the control zone

$Mass\ Flow\ Rate_{coil\ off}$ = air mass flow rate through the heat pump with the cooling coil OFF, kg/s

$h_{out, coil\ off}$ = enthalpy of air exiting the heat pump with the cooling coil OFF, J/kg

With the calculated sensible cooling energy rates and the total sensible cooling load to be met by the system, the part-load ratio for the heat pump is estimated.

$$PartLoadRatio = MAX \left(0.0, \frac{ABS(Heat\ Pump\ Cooling\ Load - NoCoolOutput)}{ABS(FullCoolOutput - NoCoolOutput)} \right) \quad (437)$$

Since the part-load performance of the DX cooling coil is frequently non-linear, and the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the cooling coil and fan) until the heat pump's cooling output matches the cooling load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's cooling output divided by the load to be met.

$$Tolerance = \frac{(Heat\ Pump\ Cooling\ Load - Q_{HeatPump})}{Heat\ Pump\ Cooling\ Load} = 0.001$$

where:

$Q_{HeatPump}$ = Heat pump delivered sensible capacity (W)

If the heat pump has been specified with cycling fan/cycling coil (AUTO fan), then the heat pump's operating supply air mass flow rate is multiplied by PartLoadRatio to determine the average air mass flow rate for the system simulation time step. In this case, the air conditions at nodes downstream of the cooling coil represent the full-load (steady-state) values when the coil is operating.

If the fan operates continuously (i.e., when the supply air fan operating mode schedule values are NOT equal to 0), the operating air mass flow rate through the heat pump is calculated as the average of the user-specified air flow rate when the cooling coil is ON and the user-specified air flow rate when the cooling coil is OFF (user-specified supply air volumetric flow rates converted to dry air mass flow rates).

$$\dot{m}_{furnace} = PartLoadRatio \left(\dot{m}_{CoolCoilON} \right) + (1 - PartLoadRatio) \left(\dot{m}_{CoilOFF} \right)$$

where:

$\dot{m}_{CoolCoilON}$ = air mass flow rate through heat pump when the cooling coil is ON (kg/s)

$\dot{m}_{CoilOFF}$ = air mass flow rate through heat pump when no cooling or heating is needed (kg/s)

In this case, the air conditions at nodes downstream of the cooling coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and inlet air conditions when the coil is OFF).

Heating Operation

Calculations for heating operation are similar to those for cooling operation in most respects. However, due to the inclusion of a supplemental heating coil, additional calculations are necessary to properly meet the total heating load for the zones being served.

If EnergyPlus determines that the heat pump must supply heating to the control zone to meet the zone air temperature setpoint, then the heat pump model computes the total sensible heating load to be delivered to the zones being served based on the control zone sensible heating load and the control zone airflow fraction.

$$\text{Heat Pump Heating Load} = \frac{\text{Control Zone Heating Load}}{\text{Control Zone Air Flow Fraction}} \quad (438)$$

The model then calculates the heat pump's sensible heating energy rate delivered to the zones being served when the system runs at full-load conditions and when the DX heating coil is OFF (without supplemental heater operation in either case). If the supply air fan cycles with the compressor, then the sensible heating energy rate is zero when the compressor is OFF. However if the fan is scheduled to run continuously regardless of coil operation, then the sensible heating energy rate will not be zero when the compressor is OFF. Calculating the sensible heating energy rate involves modeling the supply air fan (and associated fan heat), the DX cooling coil (simply to pass the air properties and mass flow rate from its inlet node to its outlet node), the DX heating coil, and the supplemental heating coil (simply to pass the air properties and mass flow rate from its inlet node to its outlet node). For each of these cases (full load and DX heating coil OFF, without supplemental heater operation in either case), the sensible heating energy rate delivered by the heat pump is calculated as follows:

$$\text{Full Heat Output} = \left(\text{Mass Flow Rate}_{\text{full load}} \right) \left(h_{\text{out,full load}} - h_{\text{control zone}} \right)_{HR_{\min}} \quad (439)$$

$$\text{No Heat Output} = \left(\text{Mass Flow Rate}_{\text{coil off}} \right) \left(h_{\text{out,coil off}} - h_{\text{control zone}} \right)_{HR_{\min}} \quad (440)$$

where:

$\text{Mass Flow Rate}_{\text{full load}}$ = air mass flow rate through heat pump at full-load conditions, kg/s

$h_{\text{out,full load}}$ = enthalpy of air exiting the heat pump at full-load conditions, J/kg

$h_{\text{control zone}}$ = enthalpy of air leaving the control zone (where thermostat is located), J/kg

HR_{\min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the heat pump exiting air or the air leaving the control zone

$\text{Mass Flow Rate}_{\text{coil off}}$ = air mass flow rate through the heat pump with the heating coil OFF, kg/s

$h_{\text{out,coil off}}$ = enthalpy of air exiting the heat pump with the heating coil OFF, J/kg

With the calculated sensible heating energy rates and the total sensible heating load to be met by the system, the part-load ratio for the heat pump is estimated.

$$\text{PartLoadRatio} = \text{MAX} \left(0.0, \frac{\text{ABS}(\text{Heat Pump Heating Load} - \text{NoHeatOutput})}{\text{ABS}(\text{FullHeatOutput} - \text{NoHeatOutput})} \right) \quad (441)$$

Since the part-load performance of the DX heating coil is frequently non-linear (Ref: Single-Speed Electric Heat Pump DX Air Heating Coil), and the supply air fan heat varies based on heating coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the heating coil compressor and fan are determined through iterative calculations (successive modeling of the heating coil and fan) until the heat pump's heating output matches the heating load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's heating output divided by the load to be met.

$$\text{Tolerance} = \frac{(\text{Heat Pump Heating Load} - Q_{\text{HeatPump}})}{\text{Heat Pump Heating Load}} = 0.001$$

where:

$Q_{HeatPump}$ = Heat pump delivered sensible capacity (W)

If the heat pump's DX heating coil output at full load is insufficient to meet the entire heating load, PartLoadRatio is set equal to 1.0 (compressor and fan are not cycling) and the remaining heating load is passed to the supplemental heating coil. If the heat pump model determines that the outdoor air temperature is below the minimum outdoor air temperature for compressor operation, the compressor is turned off and the entire heating load is passed to the supplemental gas or electric heating coil. The heat pump exiting air conditions and energy consumption are calculated and reported by the individual component models (fan, DX heating coil, and supplemental gas or electric heating coil).

If the heat pump has been specified with cycling fan/cycling coil (AUTO fan), then the heat pump's operating supply air mass flow rate is multiplied by PartLoadRatio to determine the average air mass flow rate for the system simulation time step. The air conditions at nodes downstream of the heating coils represent the full-load (steady-state) values when the coils are operating. If the fan operates continuously (i.e., when the supply air fan operating mode schedule values are NOT equal to 0), the operating air mass flow rate through the heat pump is calculated as the average of the user-specified air flow rate when the heat pump heating coil is ON and the user-specified air flow rate when the heat pump heating coil is OFF (user-specified supply air volumetric flow rates converted to dry air mass flow rates).

$$\dot{m}_{HeatPump} = PartLoadRatio \left(\dot{m}_{HeatCoilON} \right) + (1 - PartLoadRatio) \left(\dot{m}_{CoilOFF} \right)$$

where:

$\dot{m}_{HeatCoilON}$ = air mass flow rate through heat pump when the heating coil is ON (kg/s)

$\dot{m}_{CoilOFF}$ = air mass flow rate through heat pump when no heating or cooling is needed (kg/s)

In this case, the air conditions at nodes downstream of the heating coils are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coils are operating and inlet air conditions when the coils are OFF).

Unitary Multi-Speed Air-To-Air Heat Pump

Overview

The input object AirLoopHVAC:UnitaryHeatPump:AirToAir:MultiSpeed provides a model for a multispeed air-to-air heat pump that is a "virtual" component that consists of an on/off or constant volume fan component, a multispeed DX cooling coil, a multispeed DX heating coil, and a gas or electric supplemental heating coil. The main difference between this heat pump object and other EnergyPlus heat pump objects is that this object allows from two to four discrete compressor speeds for heating and cooling operation (instead of a single speed for each mode). The specific configuration of the blow through heat pump is shown in the following figure. For a draw through heat pump, the fan is located between the DX heating coil and the supplemental heating coil.

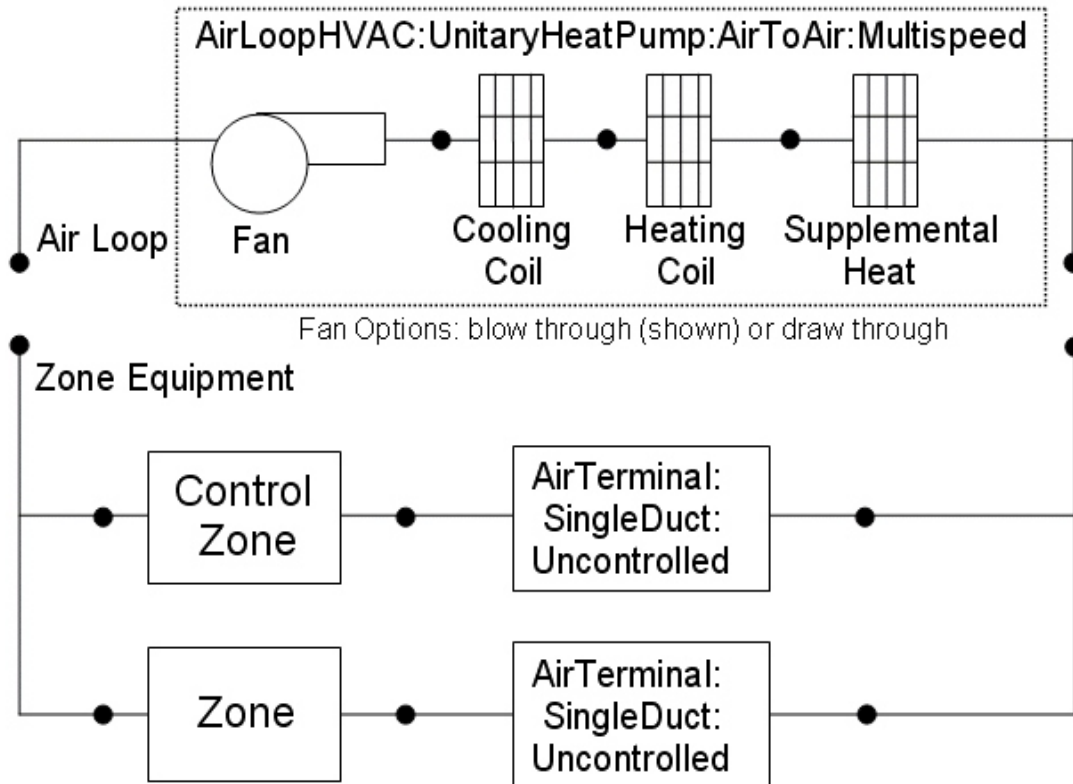


Figure 175. Schematic of a Multispeed Air-to-Air Heat Pump (Blow-through Configuration)

While the heat pump may be configured to serve multiple zones, system operation is controlled by a thermostat located in a single “control” zone. One of the key input parameters for the heat pump component is the fraction of the total system volumetric airflow that goes through the controlling zone. The heat pump module scales the calculated load for the control zone upward based on this fraction to determine the total load to be met by the heat pump. The module then proceeds to calculate the required cycling ratio, speed ratio and speed number for the system coil and determines the supply air mass flow rate to meet this total load based on the speed number. The cycling ratio is the ratio of the sensible load (heating or cooling) to the steady-state capacity of the multispeed heat pump’s DX heating or cooling coil at Speed 1 for the entire system time step. It is equivalent to the part load ratio for a single speed DX coil. The value is between 0.0 and 1.0 when the system operates at its lowest speed (Speed 1) and 1.0 when the multispeed heat pump operates at speeds above 1. The speed ratio is the ratio of time in a system time step that the compressor is at rated speed between two consecutive speed numbers ($[\text{Compressor Speed} - \text{Compressor speed at Speed } i-1] / [\text{Compressor speed at Speed } i - \text{Compressor speed at Speed } i-1]$). The compressor speed ratio is between 0.0 and 1.0 when the speed number is above 1 and is 0.0 during Speed 1 operation. The speed number is the lowest index number whose corresponding full-load sensible capacity at the given air mass flow rate is greater than or equal to the sensible load (heating or cooling) in a system time step. The heating or cooling capacity delivered by the heat pump is distributed to all of the zones served by this system via the direct air units that supply air to each zone.

The heat pump component is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). Fan:OnOff must be used to model AUTO fan, while Fan:OnOff or Fan:ConstantVolume can be used to model fan ON. The fan operation mode is specified using a supply air fan operating mode schedule where schedule values of 0 denote cycling fan operation and schedule values other than 0 (a 1 is usually used) denote continuous fan operation. Using this schedule, the supply air fan

may be cycled with cooling or heating coil operation or operated continuously based on time of day (e.g. cycling fan operation at night and continuous fan operation during the day). The fan operating mode schedule specified here overrides the Supply Air Fan Mode Operation input in the DX coil objects (Ref. Coil:Cooling:DX:MultiSpeed and Coil:Heating:DX:MultiSpeed).

Several output variables are reported by the heat pump object including fan part-load ratio, compressor part-load ratio, cycling ratio, speed ratio and speed number. Fan part-load ratio is defined as the actual air mass flow rate through the system for the time step divided by the

operating supply air mass flow rate specified for the heat pump ($\dot{m}_{actual} / \dot{m}_{ON}$) at speed 1.

Fan part-load ratio is set to 1.0 when the heat pump operates at speeds above 1. The operating supply air mass flow rate may be different for cooling, heating, and when no cooling or heating is required. Compressor part-load ratio is the actual load for the time step divided by the full-load sensible capacity (see Eqn. (444) or Eqn.(448)). If the defrost strategy is reverse cycle for a DX heating coil, the compressor part-load ratio is the sum of the actual load and the defrost load divided by the full-load sensible capacity. Therefore, the compressor part load ratio for the DX heating coil may be greater than the cycling ratio. This heat pump object also reports the sensible, latent and total cooling and heating rate, as well as the electricity consumption for the unit with separate accounting of auxiliary electric consumption. Furthermore, five report variables related to waste heat recovery are available if the user chooses to model this option.

Model Description

As described previously, the heat pump is a “virtual” component consisting of a fan, multispeed DX cooling coil, multispeed DX heating coil and supplemental heating coil. The sole purpose of the heat pump model is to properly coordinate the operation of the various system components. The following sections describe the flow of information within the model, as well as the differences between cycling and continuous supply air fan operation.

Cooling Operation

The description of heat pump cooling operation is divided in two sections: sensible capacity and average supply air flow rate. Actually, the determinations of capacity and supply air flow rate are related, so these calculations are performed in unison.

Capacity calculation

If EnergyPlus determines that the heat pump must supply cooling to the control zone to meet the zone air temperature setpoint, then the heat pump model computes the total sensible cooling load (negative) to be delivered to the zones being served based on the control zone sensible cooling load and the fraction of the heat pump air flow that goes through the control zone.

$$\text{Heat Pump Cooling Load} = \frac{\text{Control Zone Cooling Load}}{\text{Control Zone Air Flow Fraction}} \quad (442)$$

The model then calculates the heat pump’s sensible cooling energy rate delivered to the zones being served when the system runs at full-load conditions at the highest speed and when the DX cooling coil is OFF. If the supply air fan cycles with the compressor, then the sensible cooling energy rate is zero when the cooling coil is OFF. However if the fan is scheduled to run continuously regardless of coil operation, then the sensible cooling energy rate will not be zero when the cooling coil is OFF. Calculating the sensible cooling energy rate involves modeling the supply air fan (and associated fan heat) and the multispeed DX cooling coil. The multispeed DX heating coil and the supplemental heating coil are also modeled, but only to pass the air properties and mass flow rate from their inlet nodes to their outlet nodes. For each of these cases (full load at highest cooling speed and DX cooling coil OFF), the sensible cooling energy rate delivered by the heat pump is calculated as follows:

$$\begin{aligned}
 FullCoolOutput_{HighestSpeed} &= (\dot{m}_{HighestSpeed})(h_{out,fullload} - h_{controlzone})_{HRmin} \\
 NoCoolOutput_{HighestSpeed} &= (\dot{m}_{CoilOff})(h_{out,fullload} - h_{controlzone})_{HRmin}
 \end{aligned}
 \tag{443}$$

where:

$\dot{m}_{HighestSpeed}$ = air mass flow rate through heat pump at the highest cooling speed [kg/s]

$h_{out,fullload}$ = enthalpy of air exiting the heat pump at full-load conditions [J/kg]

$h_{controlzone}$ = enthalpy of air leaving the control zone (where thermostat is located) [J/kg]

HR_{min} = the minimum humidity ratio of the heat pump exiting air or the air leaving the control zone [kg/kg]

$\dot{m}_{CoilOff}$ = air mass flow rate through the heat pump with the cooling coil OFF [kg/s]

$h_{out,coil\ off}$ = enthalpy of air exiting the heat pump with the cooling coil OFF [J/kg]

If the heat pump's sensible cooling rate at the highest speed (full load, no cycling) is insufficient to meet the entire cooling load, the controlled zone conditions will not be met. The reported cycling rate and speed ratio are 1, and the speed number is set to the highest index number. If the total sensible cooling load to be met by the system is less than the sensible cooling rate at the highest speed, then the following steps are performed.

1. Calculate the sensible cooling energy rate at Speed 1

$$FullCoolOutput_{Speed1} = (\dot{m}_{Speed1})(h_{out,fullload} - h_{controlzone})_{HRmin}$$

where

\dot{m}_{Speed1} = air mass flow rate through heat pump at Speed 1 [kg/s]

2. If the sensible cooling energy rate delivered by the heat pump at Speed 1 is greater or equal to the sensible load, the cycling ratio (part-load ratio) for the heat pump is estimated.

$$\begin{aligned}
 CyclingRatio &= \frac{ABS(HeatingCoilSensibleLoad)}{FullHeatingCoilCapacity} \\
 &= MAX\left(0.0, \frac{ABS(HeatPumpHeatingLoad - AddedFanHeat)}{ABS(FullHeatOutput_{Speed1} - AddedFanHeat_{Speed1})}\right)
 \end{aligned}
 \tag{444}$$

where

AddedFanHeat = generated supply air fan heat, which is a function of part load ratio and as internal component cooling load [W].

AddedFanHeat_{Speed1} = generated supply air fan heat at Speed 1 (part load ratio=1) [W].

Since the part-load performance of the DX cooling coil is frequently non-linear, and the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the cooling coil and fan) until the heat pump's cooling output matches the cooling load to be met within

the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's cooling output divided by the load to be met.

$$Tolerance = \frac{Heat\ Pump\ Cooling\ Load - HeatPumpOutput_{Cycling}}{Heat\ Pump\ Cooling\ Load} = 0.001$$

where:

HeatPumpOutput_{Cycling} = heat pump delivered sensible capacity for Speed 1 operating at a specific cycling ratio (W)

$$HeatPumpOutput_{Cycling} = \dot{m}_{HeatPump} (h_{out} - h_{control,zone})_{HR\ min}$$

where

$\dot{m}_{HeatPump}$ = average air mass flow rate defined in the next section [kg/s]

h_{out} = enthalpy of air exiting the heat pump at part load conditions [J/kg]

For this case where speed 1 operation was able to meet the required cooling load, the speed ratio is set to zero and speed number is equal to 1.

3. If the heat pump's cooling output at full load for Speed 1 is insufficient to meet the entire cooling load, the Cycling ratio is set equal to 1.0 (compressor and fan are not cycling). Then the cooling speed is increased and the delivered sensible capacity is calculated. If the full load sensible capacity at Speed n is greater than or equal to the sensible load, the speed ratio for the heat pump is estimated:

$$Speed\ Ratio = \frac{ABS(Heat\ Pump\ Cooling\ Load - AddeFanHeat - FullCoolOutput_{Speed\ n-1})}{ABS(FullCoolOutput_{Speed\ n} - FullCoolOutput_{Speed\ n-1})}$$

Although a linear relationship is assumed by applying the speed ratio to obtain the effective capacity and mass flow rate between speed n and n-1, the outlet air node conditions are dependent on the combined outputs and may not be linear. In addition, the supply air fan heat varies with the speed ratio due to different supply mass flow rates between speed n and n-1. Therefore, the final speed ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the cooling coil and fan) until the heat pump's cooling output matches the cooling load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's cooling output divided by the load to be met.

$$Tolerance = \frac{Heat\ Pump\ Cooling\ Load - HeatPumpOutput_{SpeedRatio}}{Heat\ Pump\ Cooling\ Load} = 0.001$$

where:

HeatPumpOutput_{Speed,n} = heat pump delivered sensible capacity between two consecutive speeds at a specific speed ratio (W)

$$\begin{aligned} \text{HeatPumpOutput}_{\text{SpeedRatio}} &= (\text{SpeedRatio})\text{FullCoolOutput}_{\text{Speed } n} \\ &= (1 - \text{SpeedRatio})\text{FullCoolOutput}_{\text{Speed } n-1} - \text{AddedFanHeat}_{\text{SpeedRatio}} \end{aligned}$$

Where

AddedFanHeat_{SpeedRatio} = generated supply air fan heat at a specific speed ratio [W]

In this case, the reported cycling ratio is 1 and speed number is equal to n.

Air Mass Flow Rate Calculation

Speed 1 operation

If the heat pump has been specified with cycling fan/cycling coil (AUTO fan), then the heat pump's operating supply air mass flow rate is determined by the cycling ratio (PartLoadRatio) for Speed 1. The supply air mass flow rate is multiplied by the cycling ratio to determine the average air mass flow rate for the system simulation time step. The air conditions at nodes downstream of the cooling coils represent the full-load (steady-state) values when the coil is operating.

$$\dot{m}_{\text{HeatPump}} = (\text{CyclingRatio})\dot{m}_{\text{Speed1}}$$

If the fan operates continuously (i.e., when the supply air fan operating mode schedule values are NOT equal to 0), the operating air mass flow rate through the heat pump is calculated as the average of the user-specified air flow rate when the heat pump cooling coil is ON at Speed 1 and the user-specified air flow rate when the heat pump cooling coil is OFF (user-specified supply air volumetric flow rates converted to dry air mass flow rates).

$$\dot{m}_{\text{HeatPump}} = (\text{CyclingRatio})\dot{m}_{\text{Speed1}} + (1.0 - \text{CyclingRatio})\dot{m}_{\text{CoilOff}}$$

where:

$\dot{m}_{\text{HeatPump}}$ = average air mass flow rate through heat pump [kg/s]

\dot{m}_{Speed1} = air mass flow rate through heat pump when cooling coil is ON at Speed 1 [kg/s]

\dot{m}_{CoilOff} = air mass flow rate through heat pump when no heating or cooling is needed [kg/s]

In this case, the air conditions at nodes downstream of the cooling coils are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and inlet air conditions when the coil is OFF).

Higher Speed Operation

When the heat pump operates at higher speeds to meet the required cooling load, the supply air mass flow rate is linearly interpolated between two consecutive speeds:

$$\dot{m}_{\text{HeatPump}} = (\text{SpeedRatio})\dot{m}_{\text{Speed } n} + (1.0 - \text{SpeedRatio})\dot{m}_{\text{Speed } n-1}$$

where:

- $\dot{m}_{HeatPump}$ = average air mass flow rate through the heat pump for the time step [kg/s]
- $\dot{m}_{Speed\ n}$ = air mass flow rate through heat pump when cooling coil is ON at Speed n [kg/s]
- $\dot{m}_{Speed\ n-1}$ = air mass flow rate through heat pump when cooling coil is ON at Speed n-1 [kg/s]

For this case of higher speed operation, the air conditions at nodes downstream of the cooling coils are determined by the delivered cooling capacity and supply air mass flow rates between two consecutive speeds.

Although the above sections present the capacity and air mass flow rate calculation separately, they are dependent and change every iteration until convergence is reached for the time step being simulated.

Heating Operation

The description of heat pump heating operation is divided in two sections: total (sensible) capacity and average supply air flow rate. Actually, the determinations of capacity and supply air flow rate are related, so these calculations are performed in unison.

Capacity calculation

If EnergyPlus determines that the heat pump must supply cooling to the control zone to meet the zone air temperature setpoint, then the heat pump model computes the total sensible cooling load (positive) to be delivered to the zones being served based on the control zone sensible cooling load and the fraction of the heat pump air flow that goes through the control zone.

$$\text{Heat Pump Heating Load} = \frac{\text{Control Zone Heating Load}}{\text{Control Zone Air Flow Fraction}} \quad (445)$$

The model then calculates the heat pump's sensible heating energy rate delivered to the zones being served when the system runs at full-load conditions at the highest speed and when the DX heating coil is OFF (without supplemental heater operation in either case). If the supply air fan cycles with the compressor, then the sensible heating energy rate is zero when the compressor is OFF. However if the fan is scheduled to run continuously regardless of coil operation, then the sensible heating energy rate will not be zero when the compressor is OFF. Calculating the sensible heating energy rate involves modeling the supply air fan (and associated fan heat), the DX cooling coil (simply to pass the air properties and mass flow rate from its inlet node to its outlet node), the DX heating coil, and the supplemental heating coil (simply to pass the air properties and mass flow rate from its inlet node to its outlet node). For each of these cases (full load and DX heating coil OFF, without supplemental heater operation in either case), the sensible heating energy rate delivered by the heat pump is calculated as follows:

$$\text{FullHeatOutput}_{Highest\ Speed} = (\dot{m}_{Highest\ Speed})(h_{out,full\ load} - h_{control\ zone})_{HR\ min} \quad (446)$$

$$\text{NoHeatOutput}_{Highest\ Speed} = (\dot{m}_{Coil\ Off})(h_{out,full\ load} - h_{control\ zone})_{HR\ min} \quad (447)$$

where:

- $\dot{m}_{Highest\ Speed}$ = air mass flow rate through heat pump at the highest heating speed [kg/s]
- $h_{out,full\ load}$ = enthalpy of air exiting the heat pump at full-load conditions [J/kg]

$h_{control\ zone}$ = enthalpy of air leaving the control zone (where thermostat is located) [J/kg]

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the heat pump exiting air or the air leaving the control zone

$\dot{m}_{CoilOff}$ = air mass flow rate through the heat pump with the cooling coil OFF [kg/s]

$h_{out,coil\ off}$ = enthalpy of air exiting the heat pump with the cooling coil OFF [J/kg]

If the heat pump's DX heating coil output full load at the highest speed is insufficient to meet the entire heating load, the remaining heating load is passed to the supplemental heating coil. If the heat pump model determines that the outdoor air temperature is below the minimum outdoor air temperature for compressor operation (specified by the user), the compressor is turned off and the entire heating load is passed to the supplemental gas or electric heating coil. The heat pump exiting air conditions and energy consumption are calculated and reported by the individual component models (fan, DX heating coil, and supplemental gas or electric heating coil).

If the total heating load to be met by the system is less than the sensible heating rate at the highest speed, then the following steps are performed.

1. Calculate the sensible cooling energy rate at Speed 1

$$FullHeatOutput_{Speed1} = (\dot{m}_{Speed1})(h_{out,fullload} - h_{control\ zone})_{HR\ min}$$

where:

\dot{m}_{Speed1} = air mass flow rate through heat pump at Speed 1 [kg/s]

2. If the sensible cooling energy rate delivered by the heat pump at Speed 1 is greater or equal to the sensible load, the cycling ratio (part-load ratio) for the heat pump is estimated.

$$CyclingRatio = \frac{ABS(HeatingCoilSensibleLoad)}{FullHeatingCoilCapacity}$$

$$= MAX \left(0.0, \frac{ABS(Heat\ Pump\ Heating\ Load - AddedFanHeat)}{ABS(FullHeatOutput_{Speed1} - AddedFanHeat_{Speed1})} \right) \quad (448)$$

where

AddedFanHeat = generated supply air fan heat, which is a function of part load ratio and as internal component cooling load [W].

AddedFanHeat_{Speed1} = generated supply air fan heat at Speed 1 (part load ratio=1) [W].

Since the part-load performance of the DX heating coil is frequently non-linear (Ref: Single-Speed Electric Heat Pump DX Air Heating Coil), and the supply air fan heat varies based on heating coil operation for the case of cycling fan/cycling coil (AUTO fan), the final part-load ratio for the heating coil compressor and fan are determined through iterative calculations (successive modeling of the heating coil and fan) until the heat pump's heating output matches the heating load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's heating output divided by the load to be met.

$$Tolerance = \frac{Heat\ Pump\ Heating\ Load - HeatPumpOutput_{Cycling}}{Heat\ Pump\ Heating\ Load} = 0.001$$

where:

HeatPumpOutput_{Cycling} = heat pump delivered sensible capacity for Speed 1 operating at a specific cycling ratio (W)

$$HeatPumpOutput_{Cycling} = \dot{m}_{HeatPump} (h_{out} - h_{control,zone})_{HRmin}$$

where

$\dot{m}_{HeatPump}$ = average air mass flow rate defined in the next section [kg/s]
 h_{out} = enthalpy of air exiting the heat pump at part load conditions [J/kg]

For this case where speed 1 operation was able to meet the required cooling load, the speed ratio is set to zero and speed number is equal to 1.

3. If the heat pump's heating output at full load for Speed 1 is insufficient to meet the entire cooling load, the Cycling ratio (PartLoadRatio) is set equal to 1.0 (compressor and fan are not cycling). Then the heating speed is increased and the delivered sensible capacity is calculated. If the full load sensible capacity at Speed n is greater than or equal to the sensible load, the speed ratio for the heat pump is estimated:

$$Speed\ Ratio = \frac{ABS(Heat\ Pump\ Heating\ Load - Added\ Fan\ Heat - Full\ Heat\ Output_{Speed\ n-1})}{ABS(Full\ Heat\ Output_{Speed\ n} - Full\ Heat\ Output_{Speed\ n-1})}$$

Although a linear relationship is assumed by applying the speed ratio to obtain the effective capacity and air mass flow rate between speed n and n-1, the outlet node conditions are dependent on the combined outputs and may not be linear. In addition, the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan). Therefore, the final speed ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the cooling coil and fan) until the heat pump's cooling output matches the cooling load to be met within the convergence tolerance. The convergence tolerance is fixed at 0.001 and is calculated based on the difference between the load to be met and the heat pump's cooling output divided by the load to be met.

$$Tolerance = \frac{Heat\ Pump\ Cooling\ Load - HeatPumpOutput_{SpeedRatio}}{Heat\ Pump\ Cooling\ Load} = 0.001$$

where:

HeatPumpOutput_{SpeedRatio} = heat pump delivered sensible capacity between two consecutive speeds at a specific ratio [W]

$$\begin{aligned} HeatPumpOutput_{SpeedRatio} &= (SpeedRatio) FullHeatOutput_{Speed\ n} \\ &= (1 - SpeedRatio) FullHeatOutput_{Speed\ n-1} - AddedFanHeat_{SpeedRatio} \end{aligned}$$

Where

AddedFanHeat_{SpeedRatio} = generated supply air fan heat at a specific speed ratio [W]

In this case, the reported cycling ratio is 1 and speed number is equal to n.

Air Mass Flow Rate Calculation

The air mass flow rate calculations during heating operation are the same as those described above for cooling operation.

Fan Placement

Supply air fan placement impacts the iteration strategy. When the fan placement type is blow through, the air mass flow rate and coil part load factor (PLF) affect the fan outlet conditions. Since the fan is upstream of the coil components with this fan placement, the fan outlet conditions are calculated without knowing the next component's performance at the beginning of each iteration. DX coil performance is strongly dependent on the inlet conditions, so without correct inlet conditions the DX coil components may not be simulated correctly. Therefore, the heat pump components are called twice for each iteration when fan placement is 'blow through'. The correct part load factor for the fan component is obtained after the first call, so that the more realistic fan outlet conditions are used to simulate the coil performance in the second call. This extra call to the heat pump components is not required for the draw through fan since the supply air fan is located downstream of the DX coils with this fan placement.

Waste Heat Calculation

When the heat recovery is active (the value of the Design Heat Recovery Water Flow Rate field is greater than 0), the outlet node temperature of heat recovery is calculated based on the recoverable waste heat generated by its child objects (Coil:Cooling:DX:MultiSpeed and Coil:Heating:DX:MultiSpeed):

$$T_{outlet} = T_{inlet} + \frac{Q_{WasteHeat}}{C_p \dot{m}_{hr}}$$

where

T_{outlet} = outlet node temperature of heat recovery, C

T_{inlet} = inlet node temperature of heat recovery, C

$Q_{WasteHeat}$ = recoverable waste heat generated by its child objects, W

C_p = inlet node temperature of heat recovery, C

\dot{m}_{hr} = mass flow rate of heat recovery, kg/s

If the outlet node temperature is above the value of the Maximum Temp for Heat Recovery field, the outlet node temperature is reset to the value of Maximum Temp for Heat Recovery.

DX Cooling Package

Overview

The DX cooling package subsystem model provides a "virtual" component that consists of a DX air cooling coil component as shown in the figure below. The AirLoopHVAC:UnitaryCoolOnly input object provides a container component that controls the associated DX cooling coil which is specified in a separate object. This coil can be specified anywhere in the air loop simulation and is controlled by the setpoint(s) on the control node. Optional dehumidification controls may also be specified.

This system does not need any plant specification and will provide the electric consumption necessary to operate the compressor and the condenser fan.

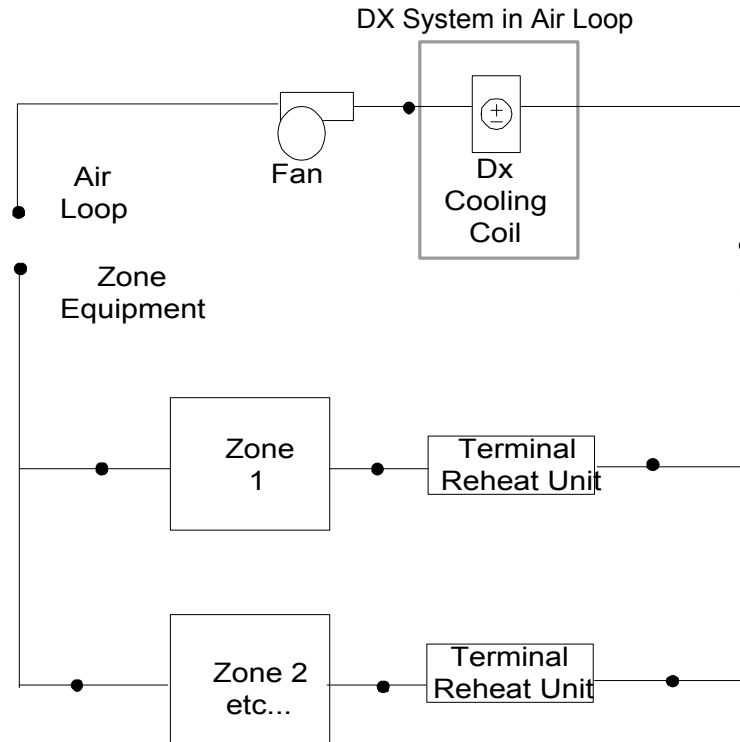


Figure 176. **NEEDS NEW FIGURE** Schematic of Packaged DX Cooling Subsystem in EnergyPlus DX System in Air Loop for a Blow-Thru Application

Controls

The system calculates the current sensible load using the temperature of the inlet node and the System Node Setpoint Temp on the control node. If the control node is not the outlet node, the desired outlet node temperature is adjusted for the current temperature difference between the outlet node and the control node. Likewise, the current latent load is calculated using the humidity ratio of the inlet node and the System Node Humidity Ratio Max on the control node. The controls determine the required coil run-time fraction and dehumidification mode (if applicable) using the steps outlined below.

Step 1 – Meet Sensible Load Requirement

The controls first attempt to meet the sensible requirement. The specified DX coil model is called with a part-load ratio (PLR) of 1.0 to determine the full-load output of the coil. This is compared with the desired outlet node temperature and a sensible PLR is calculated. If the PLR is <1.0, a Regula-Falsi iteration routine is called to determine the coil run-time fraction which results in the desired outlet node temperature.

If the cooling coil type is specified as `CoilSystem:Cooling:DX:CoolingHeatExchangerAssisted` and the dehumidification control type is specified as `CoolReheat`, the heat exchanger is active during this attempt to meet the sensible requirement.

Step 2 – Meet Latent Load Requirement (if activated)

If dehumidification controls are active, the leaving humidity ratio resulting from operation to meet the sensible load (Step 1 above) is compared with the desired outlet node humidity ratio. If the humidity requirement is already met, then no further control action is taken. If the

humidity requirement has not been met, then the coil is re-simulated depending on the type of humidity control.

Step 2a – Humidity Control = MultiMode

If the humidity control type is MultiMode, then the coil's enhanced dehumidification mode is activated when the coil type is Coil:Cooling:DX:TwoStageWithHumidityControlMode or the heat exchanger is activated when the coil type is CoilSystem:Cooling:DX:CoolingHeatExchangerAssisted and Step 1 above is repeated to meet the sensible load using the coil performance resulting from the enhanced dehumidification mode. This is a semi-passive approach to dehumidification which may fall short or may exceed the dehumidification requirement.

Step 2b – Humidity Control = CoolReheat

If the humidity control type is CoolReheat, the coil is re-simulated to achieve the desired outlet node humidity ratio. This option is valid for all cooling coil types. When the coil type is Coil:Cooling:DX:TwoStageWithHumidityControlMode, only the cooling performance mode is used for this step and enhanced dehumidification mode is not activated.

Desiccant Dehumidifier Package

Overview

The input object Dehumidifier:Desiccant:System provides a model that packages components for a desiccant-based subsystem that dehumidifies an air stream, normally called the process air stream. A second heated air stream, called the regeneration air stream, is used to remove the collected moisture from the desiccant heat exchanger and this moisture-laden air is then usually exhausted from the building. This Dehumidifier:Desiccant:System object is similar to the Dehumidifier:Desiccant:NoFans object but has some additional modeling capabilities.

The Dehumidifier:Desiccant:System desiccant dehumidifier object in EnergyPlus is a compound object that can be placed anywhere in an air loop. Common locations for this object are in an AirLoopHVAC:OutdoorAirSystem or in the main air loop downstream of a cooling coil (postcooling desiccant dehumidifier). This compound object coordinates the operation of several 'children' objects: a desiccant heat exchanger, a regeneration air fan, and an optional regeneration air heater. If this dehumidifier is placed in the main air loop immediately downstream of a direct expansion (DX) cooling coil, then the dehumidifier's operation can be coordinated with the operation of the companion DX coil and it is also possible to specify that the DX system's condenser waste heat can be used to help regenerate the desiccant heat exchanger. For the case of condenser waste heat regeneration, an optional exhaust fan can also be modeled by this desiccant dehumidifier compound object to help maintain a setpoint temperature for air entering the regeneration side of the desiccant heat exchanger. Refer to the EnergyPlus Input Output Reference for the specific input syntax for this Dehumidifier:Desiccant:System desiccant dehumidifier object.

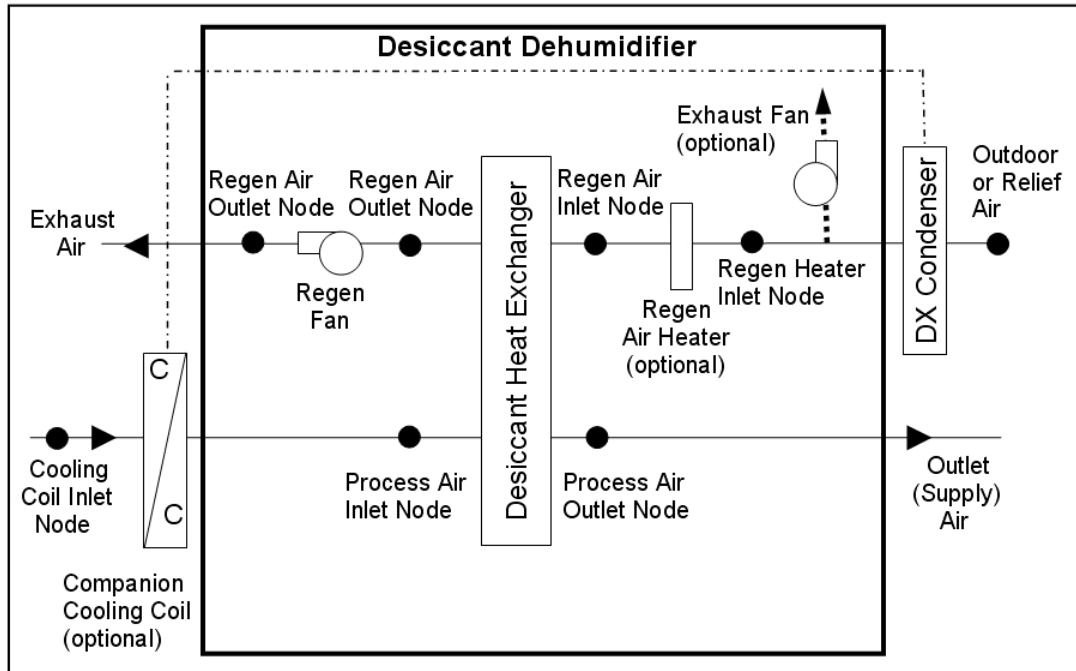


Figure 177. Schematic of a Desiccant Dehumidifier with Draw Through Regeneration Fan Placement

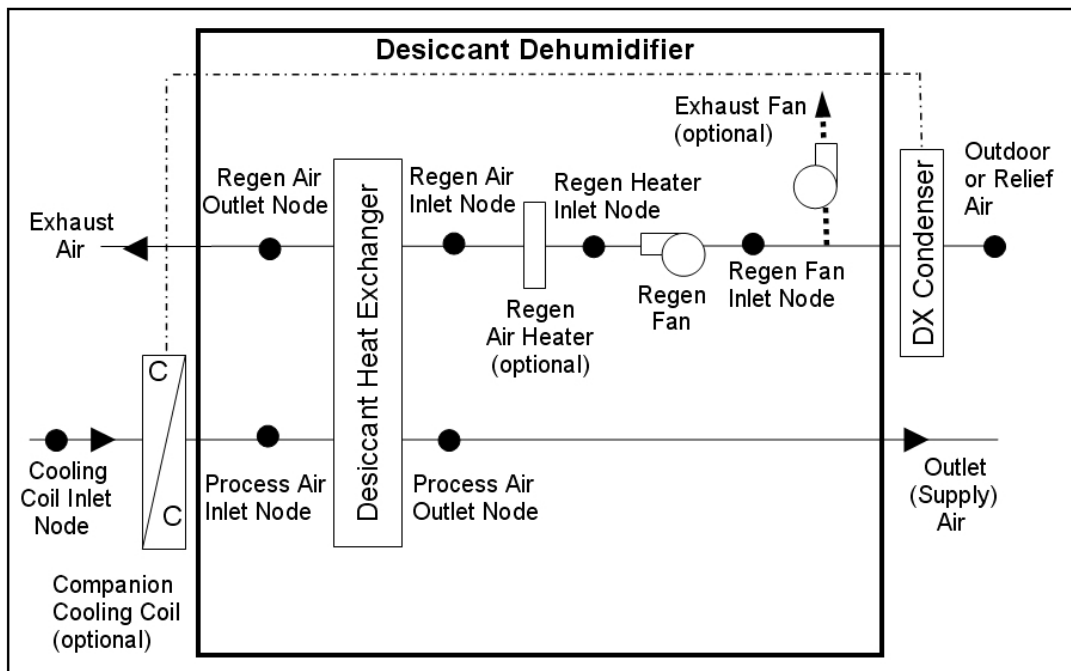


Figure 178. Schematic of a Desiccant Dehumidifier in Blow Through Regeneration Fan Placement

Control Logic

The model first decides if the dehumidifier can operate for the simulation time step, based on its availability schedule and if there is air flow detected on the process air inlet node. If available to operate, then the target humidity ratio for the process outlet air is determined based on the maximum humidity ratio setpoint on the control node (System Node Setpoint

Humidity Ratio Max). If the control node is not the process air outlet node, the target humidity ratio is adjusted for the current humidity ratio difference between the process air outlet node and the control node. If the humidity ratio of the process inlet air is greater than the target humidity ratio for the process outlet air, then the dehumidifier operates to meet the target to the extent possible.

Once it is determined that the dehumidifier should operate, the components upstream of the desiccant heat exchanger's regeneration inlet are modeled. The actual components that are modeled depend on the configuration specified by the user.

If the waste heat from the companion cooling coil is being used to heat the regeneration air, then this model determines the temperature of the air leaving the cooling coil condenser. The user may also specify a setpoint temperature for the regeneration air entering the desiccant heat exchanger and an optional exhaust fan, which can impact the condenser leaving air temperature. For this case, the exhaust fan is used to control the condenser leaving air temperature by boosting the air flow rate through the condenser to meet the setpoint temperature (minus regeneration fan heat if blow through fan placement).

Note: If the desiccant dehumidifier is OFF for a simulation time step but its companion cooling coil is operating and is specified to provide regeneration air heating, then the exhaust fan operates at the maximum air flow rate (i.e., this fan serves as the condenser fan for the companion cooling coil system when regeneration air heating is specified, so the inputs for the companion cooling coil object should not include the condenser fan energy since the condenser fan energy is modeled by the Dehumidifier:Desiccant:SystemDesiccant Dehumidifier object).

The exhaust fan power is determined as follows:

If dehumidifier is operating :

$$PLR_{exhaust} = \frac{\dot{m}_{exhaust}}{\left(\rho_{Std} * \dot{V}_{exhaust,max} \right)}$$

$$P_{exhaust} = P_{exhaust,max} * ExhFanModFac$$

If dehumidifier is not operating but companion coil is operating :

$$P_{exhaust} = P_{exhaust,max} * PLR_{companion\ coil}$$

where:

$PLR_{exhaust}$ = part load ratio of the exhaust fan

$\dot{V}_{exhaust,max}$ = exhaust fan maximum volumetric flow rate, user input (m³/s)

ρ_{Std} = density of air at standard temperature and pressure [dry air at 20°C] (m³/kg)

$P_{exhaust,max}$ = exhaust fan maximum power, user input (W)

$ExhFanModFac$ = exhaust fan power modifier curve evaluated at $PLR_{exhaust}$. If modifier curve not provided by the user, then this factor is assumed to be 1.0.

$PLR_{companion\ coil}$ = part load ratio of the companion cooling coil

$P_{exhaust}$ = output variable 'Desiccant dehumidifier exhaust fan electric power, W'

The exhaust fan electric consumption is then calculated as:

$$E_{exhaust} = P_{exhaust} * TimeStepSys * 3600.$$

where:

$E_{exhaust}$ = output variable 'Desiccant Dehumidifier Exhaust Fan Electric Consumption, J'

$TimeStepSys$ = HVAC system simulation time step, hr

Once the outlet conditions from the companion coil condenser are determined (if present), then the regeneration air fan (if blow through configuration) and regeneration air heater (if present) are simulated. To the extent possible (e.g., if the heater is available to operate based on its availability schedule and it has sufficient heating capacity), the regeneration air heater operates to raise its outlet air temperature to the specified regeneration inlet air setpoint temperature.

With the inlet air conditions to the regeneration and process sides of the desiccant heat exchanger now known, the performance of the desiccant heat exchanger is modeled. If the desiccant dehumidifier is specified with a companion cooling coil upstream of the heat exchanger's process inlet, then the model assumes that the dehumidifier operates at the same time as the companion coil (same part-load ratio). If not, then the desiccant dehumidifier model calculates the fraction of time it must operate to meet the process outlet air maximum humidity target (setpoint) using the following equation:

$$DDPartLoadRatio = \frac{(W_{Proc, in} - W_{max, set point})}{(W_{Proc, in} - W_{Proc, out, HXon})}$$

$$DDPartLoadRatio = MAX(0.0, DDPartLoadRatio)$$

$$DDPartLoadRatio = MIN(1.0, DDPartLoadRatio)$$

where:

$DDPartLoadRatio$ = output variable 'Desiccant Dehumidifier Part Load Ratio'

$W_{Proc, in}$ = process inlet air humidity ratio (kg/kg)

$W_{max, set point}$ = target humidity ratio (setpoint) for the process outlet air (kg/kg)

$W_{Proc, out, HXon}$ = process outlet air humidity ratio when the desiccant heat exchanger operates (kg/kg)

After all of the desiccant dehumidifier components are modeled at the appropriate part load ratio, the water removal rate and water removed are calculated.

$$\dot{m}_{water_removed, Proc} = \dot{m}_{Proc, in} (w_{Proc, in} - w_{Proc, out})$$

$$m_{water_removed, Proc} = \dot{m}_{water_removed, Proc} * TimeStepSys * 3600.$$

where:

$\dot{m}_{water_removed, Proc}$ = output variable 'Desiccant Dehumidifier Water Removal Rate, kg/s'

$\dot{m}_{Proc, in}$ = air mass flow rate at the process air inlet node (kg/s)

$W_{Proc, out}$ = process outlet air humidity ratio (kg/kg)

$m_{water_removed, Proc}$ = output variable 'Desiccant Dehumidifier Water Removed, kg'

References

ASHRAE. 2004. Chapter 22: Desiccant Dehumidification and Pressure-Drying Equipment. 2004 ASHRAE HVAC Systems and Equipment Handbook. Atlanta, GA: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc. <http://www.ashrae.org>

Kosar, D. 2006. Dehumidification Enhancements, ASHRAE Journal, Vol. 48, No. 2, February 2006. <http://www.ashrae.org>

Kosar, D. et al. 2006. Dehumidification Enhancement of Direct Expansion Systems Through Component Augmentation of the Cooling Coil. 15th Symposium on Improving Building Systems in Hot and Humid Climates, July 24-26, 2006. <http://www.hothumidsymposium.org/>

Unitary Single Speed Water-To-Air Heat Pump

Overview

The The input object AirLoopHVAC:UnitaryHeatPump:WaterToAir provides a model for a water-to-Air heat pump that is a “virtual” component that consists of an on/off fan component, a water-to-air heat pump cooling coil, a water-to-air heat pump heating coil, and a gas or electric supplemental heating coil. The specific configuration of the blowthru heat pump is shown in the following figure. For a drawthru heat pump, the fan is located between the water-to-air heat pump heating coil and the supplemental heating coil. The configuration of the water-to-air heat pump in the air loop is similar to an air-to-air heat pump. In addition, a water-to-air heat pump has a water loop connection on its source side. The water loop can be served by a condenser loop (like GHE for Ground source systems), or by a cooling tower/boiler plant loop (for water loop systems).

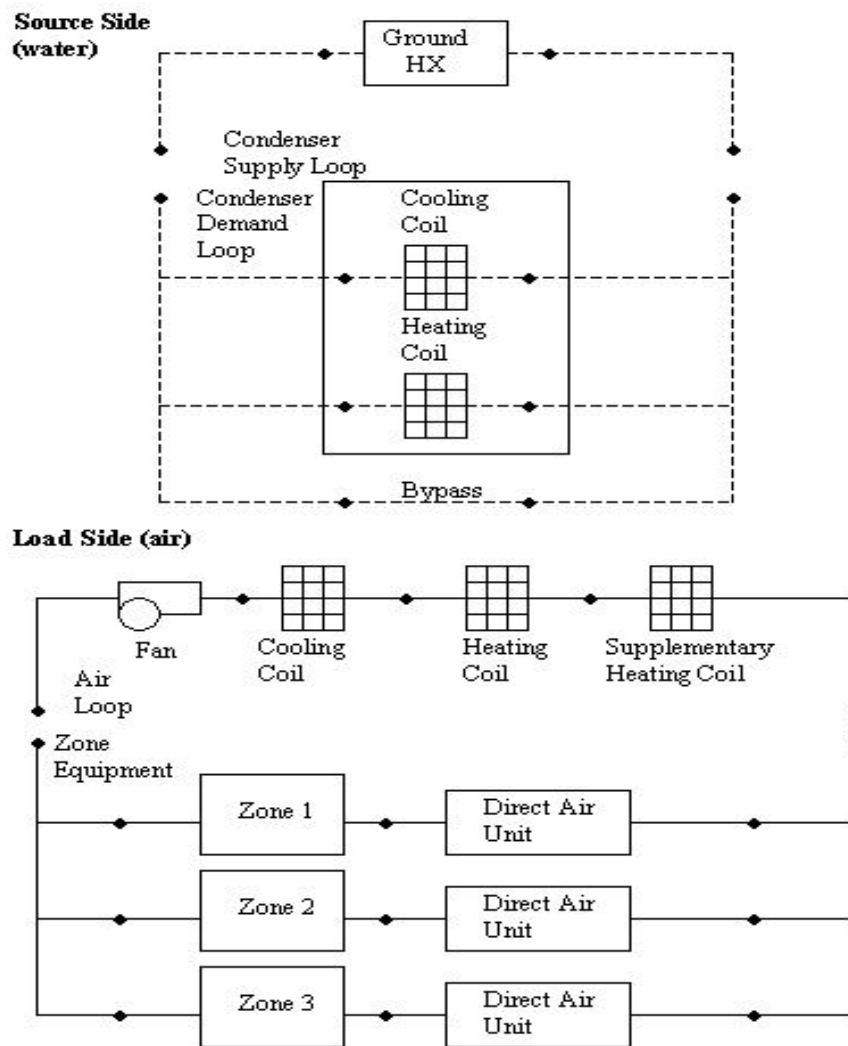


Figure 179. **NEEDS NEW FIGURE** Source Side and Load Side Configuration of a BlowThru WaterToAir Heat Pump

There are two different models for water-to-air heat pump cooling and heating coils. Cooling and heating coils are modeled using an Equation Fit model or a Parameter Estimation model. The parameter estimation procedure is quite detailed and the equation fit model is designed to simplify the user inputs. Each model is discussed in short in the following sections.

Equation-Fit Model:

This section describes the equation-fit model for Water-to-Air heat pump (Object names: Coil:Cooling:WaterToAirHeatPump:EquationFit and Coil:Heating:WaterToAirHeatPump:EquationFit). This documentation is derived from the M.S. dissertation of Tang (2005) which is available on the Oklahoma State University web site <http://www.hvac.okstate.edu/>. The model uses five non-dimensional equations or curves to predict the heat pump performance in cooling and heating mode. The methodology involved using the generalized least square method to generate a set of performance coefficients from the catalog data at indicated reference conditions. Then the respective coefficients and indicated reference conditions are used in the model to simulate the heat pump performance. The variables or inlet conditions that influenced the water-to-air heat pump performance are

load side inlet water temperature, source side inlet temperature, source side water flow rate and load side water flow rate. The governing equations for the cooling and heating mode are as following:

Cooling Mode:

$$\frac{Q_{total}}{Q_{total,ref}} = A1 + A2 \left[\frac{T_{wb}}{T_{ref}} \right] + A3 \left[\frac{T_{w,in}}{T_{ref}} \right] + A4 \left[\frac{\dot{V}_{air}}{\dot{V}_{air,ref}} \right] + A5 \left[\frac{\dot{V}_w}{\dot{V}_{w,ref}} \right]$$

$$\frac{Q_{sens}}{Q_{sens,ref}} = B1 + B2 \left[\frac{T_{db}}{T_{ref}} \right] + B3 \left[\frac{T_{wb}}{T_{ref}} \right] + B4 \left[\frac{T_{w,in}}{T_{ref}} \right] + B5 \left[\frac{\dot{V}_{air}}{\dot{V}_{air,ref}} \right] + B6 \left[\frac{\dot{V}_w}{\dot{V}_{w,ref}} \right]$$

$$\frac{Power_c}{Power_{c,ref}} = C1 + C2 \left[\frac{T_{wb}}{T_{ref}} \right] + C3 \left[\frac{T_{w,in}}{T_{ref}} \right] + C4 \left[\frac{\dot{V}_{air}}{\dot{V}_{air,ref}} \right] + C5 \left[\frac{\dot{V}_w}{\dot{V}_{w,ref}} \right]$$

Heating Mode:

$$\frac{Q_h}{Q_{h,ref}} = E1 + E2 \left[\frac{T_{db}}{T_{ref}} \right] + E3 \left[\frac{T_{w,in}}{T_{ref}} \right] + E4 \left[\frac{\dot{V}_{air}}{\dot{V}_{air,ref}} \right] + E5 \left[\frac{\dot{V}_w}{\dot{V}_{w,ref}} \right]$$

$$\frac{Power_h}{Power_{h,ref}} = F1 + F2 \left[\frac{T_{db}}{T_{ref}} \right] + F3 \left[\frac{T_{w,in}}{T_{ref}} \right] + F4 \left[\frac{\dot{V}_{air}}{\dot{V}_{air,ref}} \right] + F5 \left[\frac{\dot{V}_w}{\dot{V}_{w,ref}} \right]$$

Assuming no losses, the source side heat transfer rate for cooling and heating mode is calculated as following;

$$Q_{source,c} = Q_{total} + Power_c$$

$$Q_{source,h} = Q_h - Power_h$$

where:

$A1 - F5$ = Equation fit coefficients for the cooling and heating mode

T_{ref} = 283K

$T_{w,in}$ = Entering water temperature, K

T_{db} = Entering air dry-bulb temperature, K

T_{wb} = Entering air wet-bulb temperature, K

\dot{V}_{air} = Load side air volumetric flow rate, m³/s

\dot{V}_w = Source side water volumetric flow rate, m³/s

Q_{total}	= Total cooling capacity, W
Q_{sens}	= Sensible cooling capacity, W
$Power_c$	= Power consumption (cooling mode), W
$Q_{source,c}$	= Source side heat transfer rate (cooling mode), W
Q_h	= Total heating capacity, W
$Power_h$	= Power consumption (heating mode), W
$Q_{source,h}$	= Source side heat transfer rate (heating mode), W

The inlet conditions or variables are divided by the reference conditions. This formulation allows the coefficients to fall into smaller range of values. Moreover, the value of the coefficient indirectly represents the sensitivity of the output to that particular inlet variable. The reference conditions used when generating the performance coefficients must be the same as the reference conditions used later in the model. The reference temperature T_{ref} is fixed at 283K. Temperature unit of Kelvin is used instead of Celsius to keep the ratio of the water inlet temperature and reference temperature positive value should the water inlet temperature drop below the freezing point.

For cooling mode, the reference conditions; reference load side air volumetric flow rate ($\dot{V}_{air,ref}$), reference source side water volumetric flow rate ($\dot{V}_{w,ref}$), reference sensible capacity ($Q_{sens,ref}$) and reference power input ($Power_{c,ref}$) are the conditions when the heat pump is operating at the highest cooling capacity or reference cooling capacity ($Q_{total,ref}$) indicated in the manufacturer's catalog. Note that the reference conditions for heating mode might differ from the reference conditions specified for the cooling mode.

Coefficient estimation procedure:

The generalized least square method is used to generate the coefficients. This method utilizes an optimization method which calculates the coefficients that will give the least amount of differences between the model outputs and the catalog data. A set of coefficients for the cooling mode is generated which includes A1-A5 for total cooling capacity, B1-B6 for sensible cooling capacity, and C1-C5 for power consumption. The same procedure is repeated for the heating mode to generate the coefficients E1-E5 and F1-F5. An information flow chart showing the inputs, reference conditions, performance coefficients and outputs are shown in the figure below:

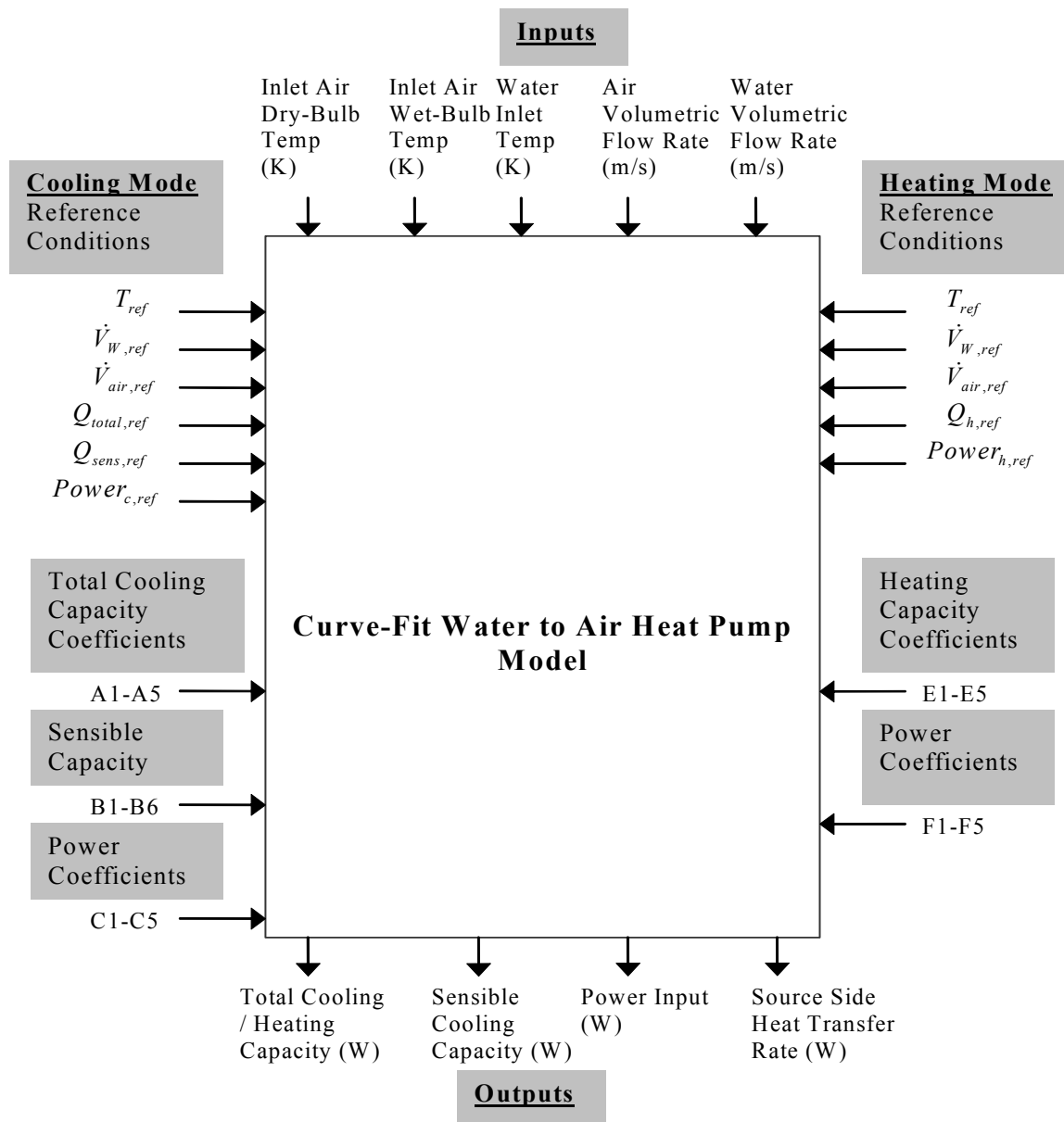


Figure 180. Information Flow Chart for Water-to-Air Heat Pump Equation Fit Model (Tang 2005)

Parameter Estimation Model

The steady state simulation model (Object: AirLoopHVAC:UnitaryHeatPump:WaterToAir) for a water-to-air vapor compression heat pump is described in this section. The model is implemented under the air-loop manager similar to the algorithm discussed under AirLoopHVAC:UnitaryHeatPump. The heat pump 'coil' objects (Coil:Cooling:WaterToAirHeatPump:ParameterEstimation and Coil:Heating:WaterToAirHeatPump:ParameterEstimation) actually consist of a steady state simulation of the unitary heat pump in cooling or heating mode respectively. This documentation is derived from the Ph.D. dissertation of Hui Jin which is available on the Oklahoma State University web site <http://www.hvac.okstate.edu/>. The model parameters, solution technique and solution algorithm are all identical to that of the water to water heat

pump. The only difference between the two models is in the modeling of the water to air heat exchanger.

The effectiveness of this heat exchanger is given by

$$\varepsilon = 1 - e^{-NTU}$$

Where NTU is defined by

$$NTU = \frac{UA}{(\dot{m}C_p)_{water/air}}$$

UA for the source side and the load side are two of the parameters estimated in the parameter estimation procedure.

Additional model details and a discussion of the parameters and parameter estimation technique are described in the following section on Water to Water Heat Pumps.

Water To Water Heat Pumps

There are two water-water heat pump models available in EnergyPlus which are **parameter estimation based model** and **equation-fit model**. Detailed descriptions of the model are available in the references Tang and Jin

Equation Fit Water To Water Heat Pump Model

This section describes the equation-fit model for water-to-water heat pump. (Object names: HeatPump:WaterToWater:EquationFit:Cooling & HeatPump:WaterToWater:EquationFit:Heating). This documentation is derived from the M.S. dissertation of Tang (2005) which is available on the Oklahoma State University web site <http://www.hvac.okstate.edu/>. The model uses four non-dimensional equations or curves to predict the heat pump performance in cooling and heating mode. The methodology involved using the generalized least square method to generate a set of performance coefficients from the catalog data at indicated reference conditions. Then the respective coefficients and indicated reference conditions are used in the model to simulate the heat pump performance. The variables that influenced the water-to-water heat pump performance are load side inlet water temperature, source side inlet temperature, source side water flow rate and load side water flow rate. The governing equations for the cooling and heating mode are as following:

Cooling Mode:

$$\frac{Q_c}{Q_{c,ref}} = A1 + A2 \left[\frac{T_{L,in}}{T_{ref}} \right] + A3 \left[\frac{T_{S,in}}{T_{ref}} \right] + A4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + A5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

$$\frac{Power_c}{Power_{c,ref}} = B1 + B2 \left[\frac{T_{L,in}}{T_{ref}} \right] + B3 \left[\frac{T_{S,in}}{T_{ref}} \right] + B4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + B5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

Heating Mode:

$$\frac{Q_h}{Q_{h,ref}} = C1 + C2 \left[\frac{T_{L,in}}{T_{ref}} \right] + C3 \left[\frac{T_{S,in}}{T_{ref}} \right] + C4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + C5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

$$\frac{Power_h}{Power_{h,ref}} = D1 + D2 \left[\frac{T_{L,in}}{T_{ref}} \right] + D3 \left[\frac{T_{S,in}}{T_{ref}} \right] + D4 \left[\frac{\dot{V}_L}{\dot{V}_{L,ref}} \right] + D5 \left[\frac{\dot{V}_S}{\dot{V}_{S,ref}} \right]$$

Assuming no losses, the source side heat transfer rate for cooling and heating mode is calculated as following;

$$Q_{source,c} = Q_c + Power_c$$

$$Q_{source,h} = Q_h - Power_h$$

Where:

$A1 - D5$ = Equation fit coefficients for the cooling and heating mode

T_{ref} = 283K

$T_{L,in}$ = Entering load side water temperature, K

$T_{S,in}$ = Entering source side water temperature, K

\dot{V}_L = Load side volumetric flow rate, m³/s

\dot{V}_S = Source side volumetric flow rate, m³/s

Q_c = Load side heat transfer rate (cooling mode), W

$Power_c$ = Power consumption (cooling mode), W

$Q_{source,c}$ = Load side heat transfer rate (cooling mode), W

Q_h = Load side heat transfer rate (cooling mode), W

$Power_h$ = Power consumption (heating mode), W

$Q_{source,h}$ = Load side heat transfer rate (heating mode), W

The inlet conditions or variables are divided by the reference conditions. This formulation allows the coefficients to fall into smaller range of values. Moreover, the value of the coefficient indirectly represents the sensitivity of the output to that particular inlet variable. The reference conditions used when generating the performance coefficients must be the same as the reference conditions used later in the model. The reference temperature T_{ref} is fixed at 283K. Temperature unit of Kelvin is used instead of Celsius to keep the ratio of the water inlet temperature and reference temperature positive value should the water inlet temperature drop below the freezing point.

For cooling mode, the reference conditions; reference load side volumetric flow rate, $\dot{V}_{L,ref}$, reference source side volumetric flow rate, $\dot{V}_{S,ref}$, reference power input, $Power_{c,ref}$ and reference source side heat transfer rate, $Q_{source,c,ref}$ are the conditions when the heat pump is operating at the highest cooling capacity or reference cooling capacity, $Q_{c,ref}$ indicated in the manufacturer's catalog. Note that the reference conditions for heating mode might differ from the reference conditions specified for the cooling mode.

Coefficient estimation procedure:

The generalized least square method is used to generate the coefficients. This method utilizes an optimization method which calculates for the coefficients that will give the least amount of differences between the model outputs and the catalog data. A set of coefficients for the cooling mode is generated which includes A1-A5 for load side heat transfer rate and B1-B5 for power consumption. The same procedure is repeated for the heating mode to generate the coefficients C1-C5 and D1-D5. An information flow chart showing the inputs, reference conditions, performance coefficients and outputs are shown in the figure below:

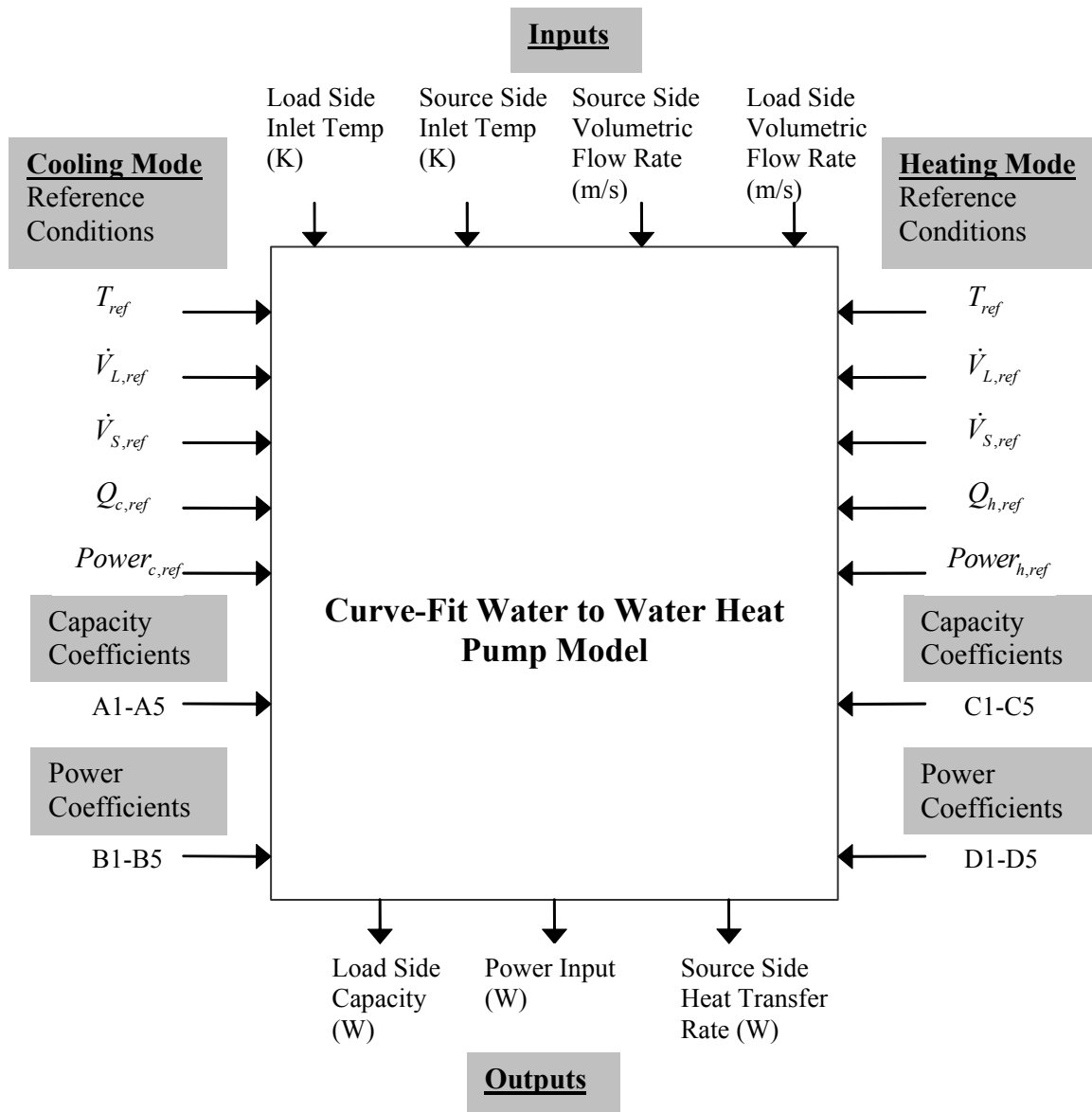


Figure 181. Information Flow Chart for Water-To-Water Heat Pump Equation Fit (Tang 2005)

Parameter Estimation Water-To-Water Heat Pump Model

A steady state simulation model for a water-to-water reciprocating vapor compression heat pump (Object names: HeatPump:WaterToWater:ParameterEstimation:Cooling & HeatPump:WaterToWater:ParameterEstimation:Heating) is described in this section. This

documentation is derived from the Ph.D. dissertation of Hui Jin which is available on the Oklahoma State University web site <http://www.hvac.okstate.edu/>. The model incorporates a multivariable unconstrained optimization algorithm to estimate several unspecific parameters. The aim of the model is to describe the detailed physical geometry and operation of each component and replicate the performance of the actual unit in operation. Assuming the thermodynamic process in the expansion device and the pressure drop at the suction and discharge valves to be isenthalpic the heat balance equation is given by

$$\dot{Q}_s = \dot{W} + \dot{Q}_L$$

Where:

\dot{Q}_s = Source side heat transfer rate

\dot{Q}_L = Load side heat transfer rate

\dot{W} = Compressor power input

The compressor model is based on an isentropic process shown in Figure 182 governed by

$$PV^\gamma = \text{Constant}$$

Where:

P = Pressure

v = Specific volume

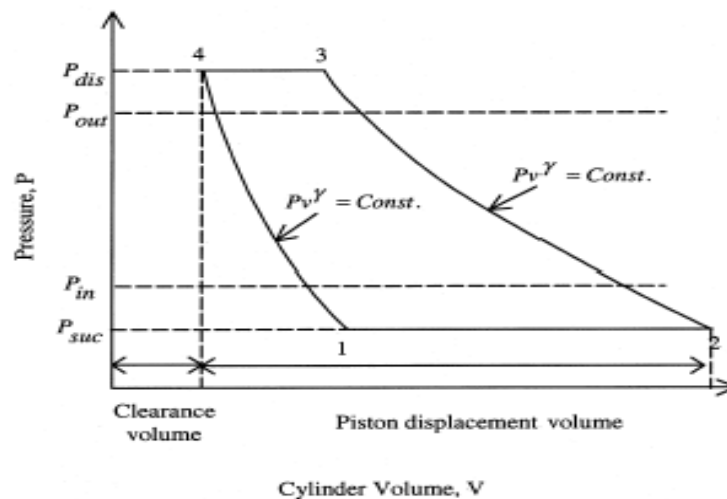


Figure 182. Schematic indicator diagram for a reciprocating Compressor(Jin 2002)

Since the refrigerant vapor in the clearance volume as shown in the previous figure goes through a re-expansion procedure, the mass flow rate of the compressor refrigerant is a decreasing function of the pressure ratio.

$$\dot{m} = \frac{PD}{v_{suc}} \left(1 + C - C \left(\frac{P_{dis}}{P_{suc}} \right)^{\frac{1}{\gamma}} \right)$$

Where:

- \dot{m} = refrigerant mass flow rate
- PD = Piston displacement
- C = Clearance factor
- P_{dis} = discharge pressure
- P_{suc} = Suction pressure
- γ = Isentropic exponent

Parameter estimation procedure:

A set of parameters for the cooling mode is defined on the basis of the equations used in the model. An information flowchart indicating the parameters, inputs to the model and the resulting outputs acquired are shown in Figure 183. The estimation of parameters is conducted using the catalog data.

The parameter definition include:

- Piston displacement, PD
- Clearance factor, C
- Pressure drop across the suction and discharge valves, ΔP
- Loss factor used to define the electromechanical losses supposed to be proportional to the theoretical power, η
- Superheat in °C or F, ΔT_{sh}
- Constant part of the electromechanical losses, W_{loss}
- Source side heat transfer coefficient, $(UA)_s$
- Load side heat transfer coefficient, $(UA)_L$

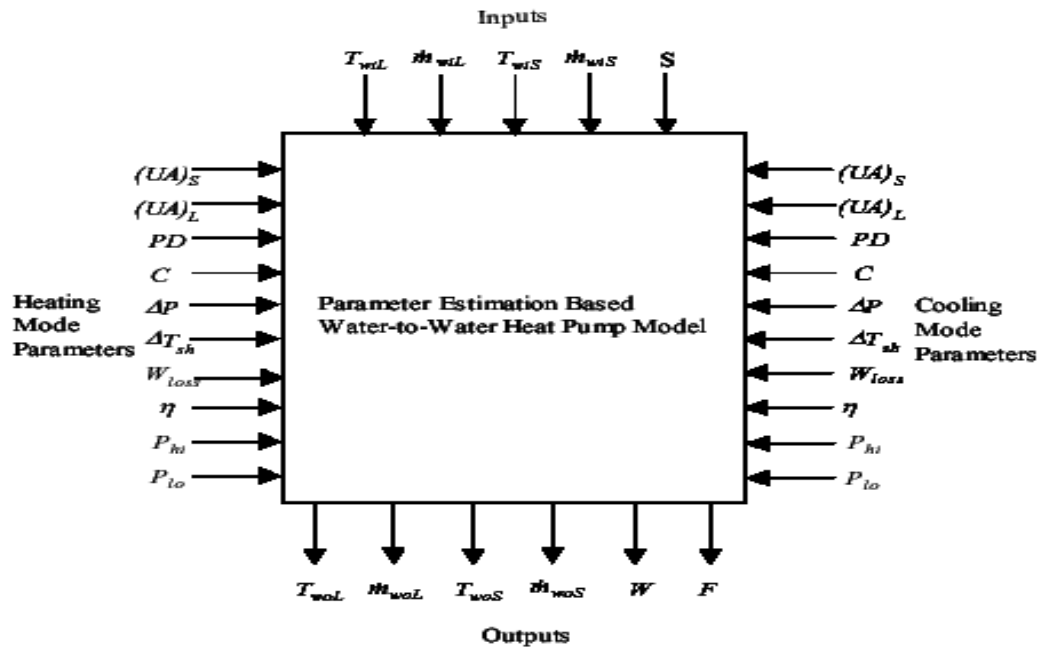


Figure 183. Information Flowchart for Water-To-Water Heat Pump Parameter Estimation Mmodel implementation (Jin 2002)

Where:

T_{wiL} = Entering water Load side temperature

T_{wiS} = Entering water Source side temperature

•

m_{wiL} = Entering water Load side mass flow rate

•

m_{wiS} = Entering water Source side mass flow rate

S = Thermostatic Signal

The parameter estimation procedure incorporates an objective function that computes the difference between the model outputs and the catalog outputs. The objective function is then minimized by using a multi variable unconstrained multi modal Nelder Mead optimization algorithm. As the objective function value lowers after each iteration, the model outputs approach the catalog outputs consequently leading to convergence and the correct parameters are estimated for the respective model. The inputs to the model include the entering water temperatures and mass flow rates on the load side and the source side. The calculation of the objective function is shown in the form of a formula flowchart in Figure 184. The square of the sum of the errors (SSQE) for a given set of parameter values that will be minimized is given by

$$SSQE = \sum_{i=1} \left(\frac{\left(\left(W_{cat} \right) - \left(W \right)_i \right)}{\left(W_{cat} \right)} \right)^2 + \left(\frac{\left(\left(QL_{cat} \right) - \left(QL \right)_i \right)}{\left(QL_{cat} \right)} \right)$$

Where

W_{cat} = Catalog power consumption

W = Model power consumption

QL_{cat} = Catalog load side heat transfer

QL = Model load side heat transfer

Extrapolation beyond the catalog data grants the parameter estimation model an upper hand in comparison with the equation fit and deterministic models. However, the detailed model is computationally more intensive. Moreover, when the model is implemented within a transient system simulation program, it may come across figures that are random and unplanned by the manufacturer such as low water flow rates or extreme temperatures. This oddity may result in unrealistic set of results.

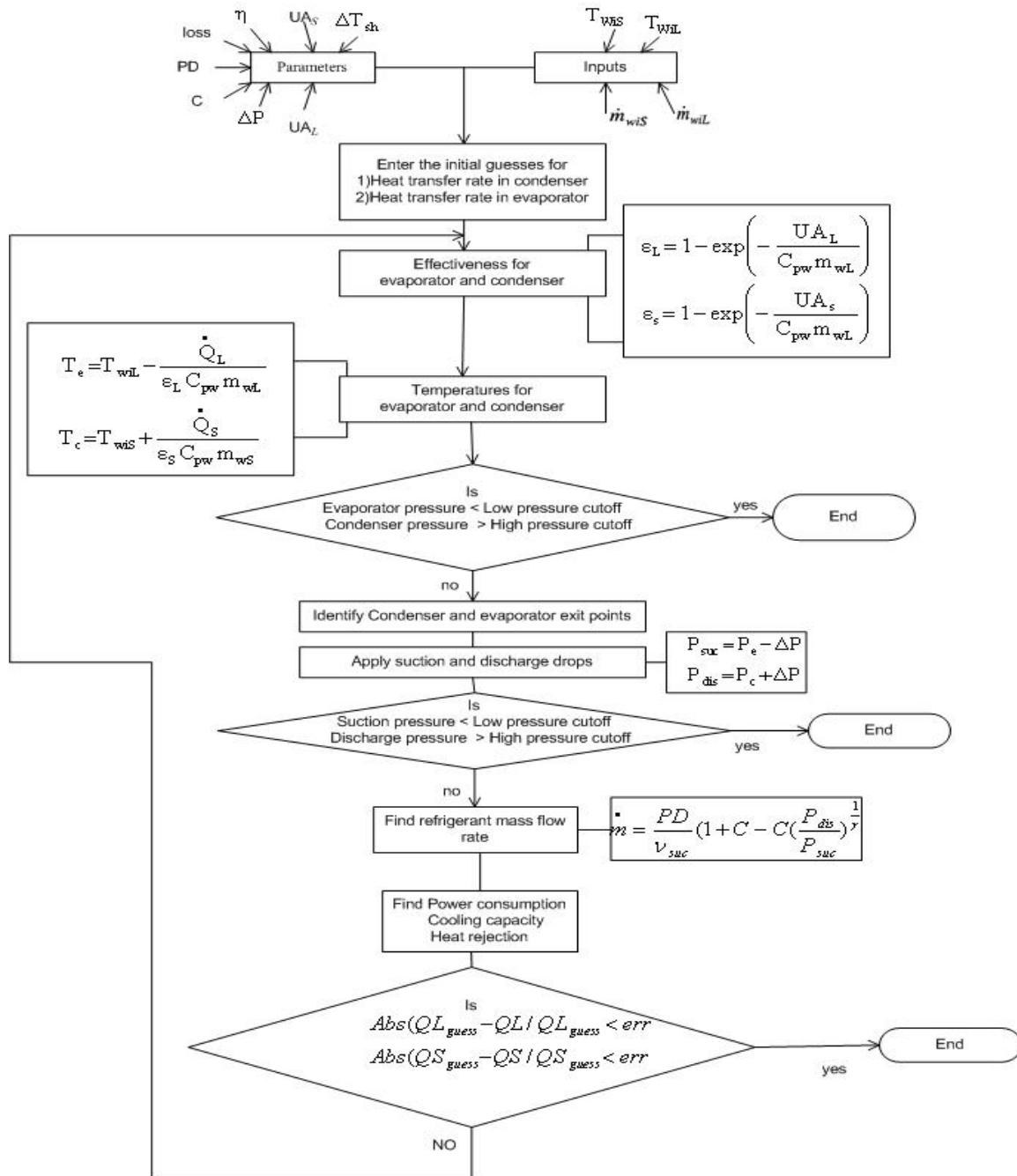


Figure 184. Flow diagram of EnergyPlus Water to Water HeatPump implementation

Control Strategy:

Both water-to-water heat pump models; parameter estimation based model and curve-fit model use the “cycle time control logic” developed by Muruggappan (2002).

This strategy keeps the heat pump from short-cycling whereby the heat pump will stay on or off for the specified cycle time after switching states. The control logic is identical to the operation of a physical heat pump whereby the heat pump does not switch between on and off instantly. Refer to Muruggappan (2002) for the further details on the control strategy.

References

Jin, Hui. 2002. Parameter Estimation Based Models of Water Source Heat Pumps. Phd. Thesis, Department of Mechanical and Aerospace Engineering, Oklahoma State University. (downloadable from <http://www.hvac.okstate.edu/>)

Tang, C. C. 2005. Modeling Packaged Heat Pumps in Quasi-Steady State Energy Simulation Program. M.S. Thesis. Department of Mechanical and Aerospace Engineering, Oklahoma State University. (downloadable from <http://www.hvac.okstate.edu/>)

Murugappan, Arun. 2002. Implementing Ground Source Heat Pump and Ground Loop Heat Exchanger Models in the EnergyPlus Simulation Environment, M.S. Thesis, Department of Mechanical and Aerospace Engineering, Oklahoma State University (downloadable from <http://www.hvac.okstate.edu/>)

Heat Exchangers

Air System Air-To-Air Sensible and Latent Effectiveness Heat Exchanger

Overview

The input object HeatExchanger:AirToAir:SensibleAndLatent provides a model for a generic, sensible and latent air-to-air heat exchanger that is an HVAC air system component that consists of a heat exchanger and primary/secondary airflow bypass dampers. The specific configuration of the component is shown in the following figure.

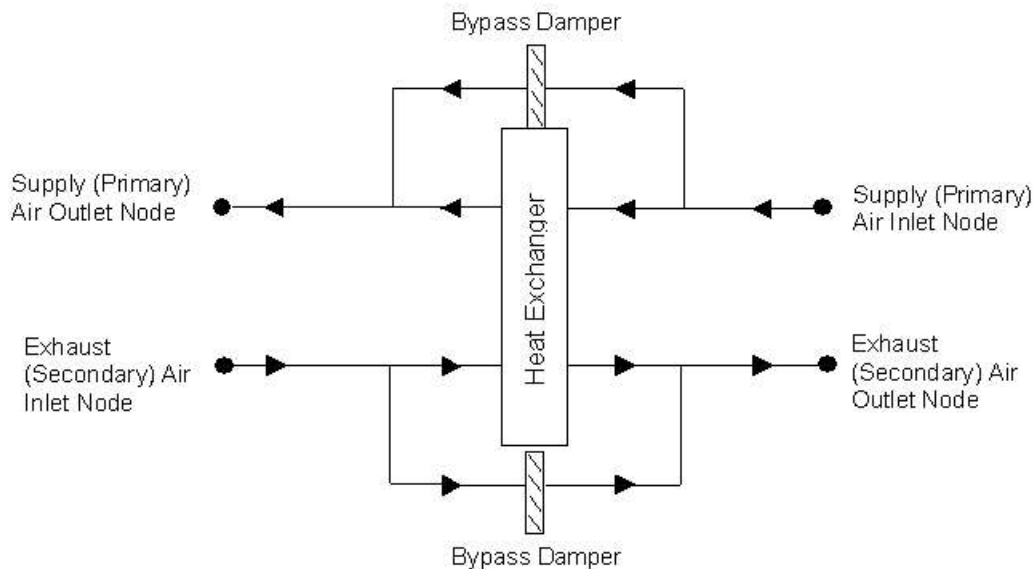


Figure 185. Schematic of the Sensible and Latent Air-to-Air Heat Exchanger

The sensible and latent air-to-air heat exchanger is typically used for exhaust or relief air heat recovery. Heat exchanger performance can be specified to transfer sensible energy, latent energy or both between the supply and exhaust air streams. The input requires no geometric data. Performance is defined by specifying sensible and/or latent effectiveness at 75% and 100% of the nominal (rated) supply air flow rate in both heating and cooling conditions (Table 46).

Table 46. Operating Conditions for Defining Heat Exchanger Performance

Parameter	Conditions	
	Heating	Cooling
Entering supply air temperature:		
Dry-bulb	1.7°C (35°F)	35°C (95°F)
Wet-bulb	0.6°C (33°F)	26°C (78°F)
Entering exhaust air temperature:		
Dry-bulb	21°C (70°F)	24°C (75°F)
Wet-bulb	14°C (58°F)	17°C (63°F)

Note: Conditions consistent with the Air-Conditioning and Refrigeration Institute's Standard 1060 (ARI 2001).

Heat exchange between the supply and exhaust air streams occurs whenever the unit is scheduled to be available (availability schedule) and supply/exhaust air flows are present. This heat exchanger object can be used in conjunction with a conventional air-side economizer (i.e., specify an appropriate economizer control type in the Controller:OutdoorAir object), whereby heat exchange is suspended whenever the air-side economizer is active (i.e., air flow is fully bypassed around a fixed-plate heat exchanger or the rotation of a rotary heat exchanger is stopped). This object is also able to suspend heat exchange for the purpose of providing free cooling operation in the absence of a conventional air-side economizer (i.e., specify MinimumFlowWithBypass in the Controller:Outside Air object). Suspension of heat exchange during economizer mode may be customized as necessary using the economizer lockout field.

Several methods of frost control are available to warm the heat exchanger core to prevent frost formation. Available methods are preheat, exhaust only, exhaust air recirculation, and minimum exhaust temperature. Preheat frost control uses a separate heater object placed in the supply inlet air stream to keep the air temperature above the frost threshold temperature. All other frost control methods are modeled within this heat exchanger object.

This heat exchanger object can also control the supply air outlet temperature to a setpoint when a setpoint manager and temperature schedule are used. This temperature control is accomplished through wheel speed modulation or bypassing supply air around the heat exchanger to maintain the desired setpoint and avoid overheating the supply air.

Model Description

The heat exchanger object models energy transfer between the supply air stream and the exhaust air stream according to the effectiveness values that are specified by the user in the input data file (see IO Reference Document, HeatExchanger:AirToAir:SensibleAndLatent). The operating volumetric air flow rate through the heat exchanger (i.e., the average of the actual supply and exhaust air flow rates for the simulation time step) should be between 50% and 130% of the nominal supply air flow rate specified for the heat exchanger. Operating air flow rates outside this range result in a warning message and a recommendation to adjust air flow rates to within the appropriate range.

The user must enter the sensible and latent effectiveness of the heat exchanger for heating and cooling conditions (Table 46) with balanced air flow (supply flow equal to the exhaust flow) at two flow rates: 75% and 100% of the nominal supply air flow rate. Heat exchanger manufacturers can typically provide this performance information, and it is also available for equipment contained in ARI's Certified Product Directory for Air-to-Air Energy Recovery Ventilation Equipment (ARI 2003). Values may be entered for sensible effectiveness, latent effectiveness, or both. The model assumes default values of 0.0 for sensible and latent effectiveness, thus requiring the user to input representative values for the heat exchanger being modeled.

To obtain the “operating” effectiveness of the heat exchanger at different air flow rates, the model first calculates the average volumetric air flow rate through the heat exchanger (average of the supply and exhaust air flow rates) for each simulation time step. Air flows through the heat exchanger may be unbalanced (supply greater than exhaust, or vice versa), but an unbalanced air flow ratio greater than 2:1 results in a fatal warning message and program termination. The model determines the operating effectiveness of the heat exchanger by linear interpolation or extrapolation of the 100% flow and 75% flow effectiveness values specified in the input data file, using the average volumetric air flow rate through the heat exchanger. Extrapolation is allowed down to 50% and up to 130% of the nominal supply air flow rate (beyond this range a warning message is issued).

$$\mathcal{E}_{operating,sensible} = \mathcal{E}_{sensible,75\%flow} + (\mathcal{E}_{sensible,100\%flow} - \mathcal{E}_{sensible,75\%flow}) \left(\frac{HX_{flowratio} - 0.75}{1 - 0.75} \right) \quad (449)$$

$$\mathcal{E}_{operating,latent} = \mathcal{E}_{latent,75\%flow} + (\mathcal{E}_{latent,100\%flow} - \mathcal{E}_{latent,75\%flow}) \left(\frac{HX_{flowratio} - 0.75}{1 - 0.75} \right) \quad (450)$$

where:

$\mathcal{E}_{operating,sensible}$ = operating sensible effectiveness of the heat exchanger

$\mathcal{E}_{operating,latent}$ = operating latent effectiveness of the heat exchanger

$\mathcal{E}_{sensible,75\%flow}$ = sensible effectiveness at 75% airflow condition

$\mathcal{E}_{sensible,100\%flow}$ = sensible effectiveness at 100% airflow condition

$\mathcal{E}_{latent,75\%flow}$ = latent effectiveness at 75% airflow condition

$\mathcal{E}_{latent,100\%flow}$ = latent effectiveness at 100% airflow condition

$HX_{flowratio}$ = the ratio of the average operating volumetric air flow rate [(supply flow plus exhaust flow) / 2.0] to the nominal supply air flow rate

If the heat exchanger’s supply air inlet temperature is less than the exhaust air inlet temperature, the operating sensible and latent effectivenesses are calculated using the 75% and 100% heating condition values; otherwise, the 75% and 100% cooling effectiveness values are used in Equations (449) and (450).

The supply air conditions leaving the heat exchanger are determined using the heat exchanger operating effectiveness calculated above, the ratio of the air stream with the minimum heat capacity rate to the supply air stream heat capacity rate, and the difference in temperature or humidity ratio between the supply and exhaust inlet air:

$$\dot{m} c_{p, \min} = \text{MIN}(\dot{m} c_{p, \text{sup}}, \dot{m} c_{p, \text{exh}})$$

$$T_{SupAirOut} = T_{SupAirIn} + \mathcal{E}_{operating,sensible} \left(\frac{\dot{m} c_{p, \min}}{\dot{m} c_{p, \text{sup}}} \right) (T_{ExhAirIn} - T_{SupAirIn})$$

$$\omega_{SupAirOut} = \omega_{SupAirIn} + \mathcal{E}_{operating,latent} \left(\frac{\dot{m} c_{p, min}}{\dot{m} c_{p, sup}} \right) (\omega_{ExhAirIn} - \omega_{SupAirIn})$$

where:

- $\dot{m} c_{p, min}$ = minimum heat capacity rate (W/K)
- $\dot{m} c_{p, sup}$ = heat capacity rate of the supply air stream (W/K)
- $\dot{m} c_{p, exh}$ = heat capacity rate of the exhaust air stream (W/K)
- $T_{SupAirOut}$ = supply air temperature leaving the heat exchanger (°C)
- $T_{SupAirIn}$ = supply air inlet temperature (°C)
- $T_{ExhAirIn}$ = exhaust air inlet temperature (°C)
- $\omega_{SupAirOut}$ = supply air humidity ratio leaving the heat exchanger (kg/kg)
- $\omega_{SupAirIn}$ = supply air inlet humidity ratio (kg/kg)
- $\omega_{ExhAirIn}$ = exhaust air inlet humidity ratio (kg/kg)

Using the supply air outlet temperature and humidity ratio, the enthalpy of the supply air leaving the heat exchanger is calculated.

$$h_{SupAirOut} = PsyHFNTdbW(T_{SupAirOut}, \omega_{SupAirOut})$$

where:

- $h_{SupAirOut}$ = enthalpy of the supply air leaving the heat exchanger (J/kg)
- $PsyHFNTdbW$ = psychrometric routine calculating air enthalpy as a function of temperature and humidity ratio

If the predicted conditions of the supply air leaving the heat exchanger exceed the saturation curve (>100% RH), then the temperature and humidity ratio of the air are reset to saturated conditions (= 100% RH) at the enthalpy condition calculated above ($h_{SupAirOut}$).

Next, the sensible and total heat recovery rates of the heat exchanger are calculated:

$$\dot{Q}_{Sensible} = (\dot{m} c_{p, sup}) (T_{SupAirIn} - T_{SupAirOut}) \quad (451)$$

$$\dot{Q}_{Total} = \dot{m}_{SupAir} (h_{SupAirIn} - h_{SupAirOut}) \quad (452)$$

where:

- $\dot{Q}_{Sensible}$ = sensible heat recovery rate (W)
- \dot{Q}_{Total} = total heat recovery rate (W)
- $h_{SupAirIn}$ = supply air inlet enthalpy (J/kg)
- \dot{m}_{SupAir} = mass flow rate of the supply air stream (kg/s)

The conditions of the exhaust (secondary) air leaving the heat exchanger are then calculated:

$$T_{ExhAirOut} = T_{ExhAirIn} + \frac{\dot{Q}_{Sensible}}{\dot{m} C_{p, exh}}$$

$$h_{ExhAirOut} = h_{ExhAirIn} + \frac{\dot{Q}_{Total}}{\dot{m}_{ExhAir}}$$

$$\omega_{ExhAirOut} = PsyWFnTdbH(T_{ExhAirOut}, h_{ExhAirOut})$$

where:

$T_{ExhAirOut}$ = exhaust air temperature leaving the heat exchanger (°C)

$h_{ExhAirOut}$ = exhaust air enthalpy leaving the heat exchanger (J/kg)

\dot{m}_{ExhAir} = mass flow rate of the exhaust air stream (kg/s)

$\omega_{ExhAirOut}$ = exhaust air humidity ratio leaving the heat exchanger (kg/kg)

$PsyWFnTdbH$ = psychrometric routine calculating air humidity ratio as a function of temperature and enthalpy

As was done for the supply air, calculated exhaust air conditions beyond the saturation curve are reset to saturation conditions at the calculated air enthalpy value.

Once the air conditions leaving each side of the heat exchanger (supply and exhaust) are calculated, this air is blended with any bypass air that was directed around the heat exchanger core to determine the final air conditions leaving the heat exchanger unit. These outlet air conditions are used in Equations (451) and (452) to determine the sensible and total heat recovery rate for the overall heat exchanger unit. The latent heat recovery rate for the overall unit is then calculated as the difference between the total and sensible heat recovery rates:

$$\dot{Q}_{Latent} = \dot{Q}_{Total} - \dot{Q}_{Sensible}$$

Heat recovery electric power is the electric consumption rate of the unit in watts. The nominal electric power rate for the heat exchanger is specified in the input data file, and can be used to model controls (transformers, relays, etc.) and/or a motor for a rotary heat exchanger. The model assumes that this electric power is consumed whenever the heat exchanger is scheduled to operate and supply/exhaust air flow rates exist. The electric power is assumed to be zero for all other times or if heat exchange is suspended to provide free cooling (economizer operation). None of this electric power is assumed to contribute thermal load to either of the heat exchanger air streams.

At the end of each HVAC simulation time step, this object reports the sensible, latent and total heat recovery rates for the overall unit as calculated above. The heat recovery rates are reported separately for times when the supply air is heated and when it is cooled (Ref: HeatExchanger:AirToAir:SensibleAndLatent in the EnergyPlus Input Output Reference). The heat recovery electric power is also reported for each simulation time step. In addition to the heat recovery rates and electric power, heating/cooling energy transferred to the supply air and the electric energy consumption by the heat exchanger unit are calculated for the time step being reported as follows:

$$Q_{SensibleCooling} = \dot{Q}_{SensibleCooling} * TimeStepSys * 3600.$$

$$Q_{LatentCooling} = \dot{Q}_{LatentCooling} * TimeStepSys * 3600.$$

$$Q_{TotalCooling} = \dot{Q}_{TotalCooling} * TimeStepSys * 3600.$$

$$Q_{SensibleHeating} = \dot{Q}_{SensibleHeating} * TimeStepSys * 3600.$$

$$Q_{LatentHeating} = \dot{Q}_{LatentHeating} * TimeStepSys * 3600.$$

$$Q_{TotalHeating} = \dot{Q}_{TotalHeating} * TimeStepSys * 3600.$$

$$E_{HXUnit} = P_{HXUnit} * TimeStepSys * 3600.$$

where:

$Q_{SensibleCooling}$ = output variable 'Heat Exchanger Sensible Cooling Energy, J'

$\dot{Q}_{SensibleCooling}$ = output variable 'Heat Exchanger Sensible Cooling Rate, W' = $\dot{Q}_{Sensible}$
during times when the supply air is cooled

$TimeStepSys$ = HVAC system simulation time step, hr

$Q_{LatentCooling}$ = output variable 'Heat Exchanger Latent Cooling Energy, J'

$\dot{Q}_{LatentCooling}$ = output variable 'Heat Exchanger Latent Cooling Rate, W' = \dot{Q}_{Latent}
during times when the supply air is dehumidified

$Q_{TotalCooling}$ = output variable 'Heat Exchanger Total Cooling Energy, J'

$\dot{Q}_{TotalCooling}$ = output variable 'Heat Exchanger Total Cooling Rate, W' = \dot{Q}_{Total} during
times when the supply air enthalpy is reduced

$Q_{SensibleHeating}$ = output variable 'Heat Exchanger Sensible Heating Energy, J'

$\dot{Q}_{SensibleHeating}$ = output variable 'Heat Exchanger Sensible Heating Rate, W' = $\dot{Q}_{Sensible}$
during times when the supply air is heated

$Q_{LatentHeating}$ = output variable 'Heat Exchanger Latent Heating Energy, J'

$\dot{Q}_{LatentHeating}$ = output variable 'Heat Exchanger Latent Heating Rate, W' = \dot{Q}_{Latent}
during times when the supply air is humidified

$Q_{TotalHeating}$ = output variable 'Heat Exchanger Total Heating Energy, J'

$\dot{Q}_{TotalHeating}$ = output variable 'Heat Exchanger Total Heating Rate, W' = \dot{Q}_{Total} during

	times when the supply air enthalpy is increased
E_{HXUnit}	= output variable 'Heat Recovery Electric Consumption, J'
P_{HXUnit}	= output variable 'Heat Recovery Electric Power, W'

Frost Control Methods

In cold weather, frost can form on the heat exchanger causing a reduction in air flow and heat recovery performance. Various strategies can be employed to limit frost formation. Heat exchangers that transfer total energy (sensible plus latent) usually have a lower frost threshold temperature than sensible-only heat exchangers. Frost threshold temperatures for sensible-only heat exchangers may be -1°C to -12°C for plate and rotary heat exchangers respectively, while those for comparable total (sensible plus latent) heat exchangers may be 10°C lower. The frost threshold temperature for a specific application is dependent on the exhaust air dry-bulb temperature and relative humidity, heat exchanger type (e.g., sensible-only or total heat exchange, flat plate or rotary), and the heat exchanger effectiveness. Consult manufacturer's literature to obtain specific frost threshold temperatures for the heat exchanger being modeled.

Four frost control strategies can be modeled for this air-to-air heat exchanger unit. Each of these four strategies is discussed in detail below.

Preheat

One method to control frost formation is to preheat the cold outdoor (supply) air entering the heat exchanger. When a preheat coil is used for frost control, a separate heating coil object must be placed in the supply air stream at the inlet to the heat exchanger (Coil:Heating:Water, Coil:Heating:Electric or Coil:Heating:Gas). The preheat coil should be controlled to maintain a minimum supply air inlet temperature thereby eliminating frost buildup on the heat exchanger core. When modeling preheat frost control, specify "None" as the frost control method in the heat exchanger object. When modeling this heat exchanger as part of an air loop, refer to the objects AirLoopHVAC:OutdoorAirSystem and SetpointManager:Scheduled the EnergyPlus Input Output Reference for additional information on specifying a preheat coil and controlling its supply air temperature. This frost control method is not currently available when this heat exchanger is being used as part of the compound object ZoneHVAC:EnergyRecoveryVentilator.

Exhaust Only

This method of frost control bypasses the incoming supply air around the heat exchanger core thereby warming the core using the exiting exhaust air. This method is similar to 'supply air off' frost control where the supply air fan is turned off for a predetermined period of time while the exhaust air fan continues to operate. For the 'supply air off' method, the supply air flow is stopped for a period of time thereby reducing the ventilation air supplied to the zone(s). In addition, the building may be negatively pressurized for the period of time that the supply air flow is stopped since the exhaust air fan continues to operate. On the other hand, the 'exhaust only' method of frost control modeled by EnergyPlus continues to provide outdoor ventilation air to the zone(s), but this air is simply bypassed around the heat exchanger core for a portion of the time and the potential problem with negatively pressurizing the building is avoided. Since the supply airflow rate through the heat exchanger core is purposely reduced to control frost formation, average volumetric airflow rates below 50% of nominal are allowed when this frost control is active and no warning message is issued.

The user enters a threshold temperature, an initial defrost time fraction, and a rate of defrost time fraction increase. When the temperature of the supply air (e.g., outdoor air) entering the heat exchanger is equal to or below the specified threshold temperature, the fractional amount of time that the supply air is bypassed around the heat exchanger core is determined from the following equation:

$$X_{DefrostTime} = X_{Initial} + X_{RateofIncrease} (T_{Threshold} - T_{SupAirIn})$$

where:

$$X_{DefrostTime} = \text{Fractional time period for frost control } (0 \leq X_{DefrostTime} \leq 1)$$

$$X_{Initial} = \text{Initial defrost time fraction}$$

$$X_{RateofIncrease} = \text{Rate of defrost time fraction increase } (\text{K}^{-1})$$

$$T_{Threshold} = \text{Threshold temperature } (^{\circ}\text{C})$$

$$T_{SupAirIn} = \text{Supply air inlet temperature } (^{\circ}\text{C})$$

During the defrost time, supply air flow is fully bypassed around the heat exchanger core and no heat transfer takes place. For the remainder of the time period, no air is bypassed and full heat exchange is achieved. The average supply air flow bypassed around the heat exchanger core is calculated as follows:

$$\dot{m}_{SupAirBypass} = (X_{DefrostTime}) \dot{m}_{SupAir}$$

To determine the average heat transfer rates for the simulation time step, the supply air outlet conditions are first calculated as if the heat exchanger were not in defrost mode (see previous section, Model Description). The sensible and total heat transfer rates are then calculated and multiplied by the fractional time period that the heat exchanger is not in defrost mode ($1 - X_{DefrostTime}$).

$$\dot{Q}_{Sensible} = (1 - X_{DefrostTime}) \left(\dot{m} c_{p, sup} \right) (T_{SupAirIn} - T_{SupAirOut})$$

$$\dot{Q}_{Total} = (1 - X_{DefrostTime}) \dot{m}_{SupAir} (h_{SupAirIn} - h_{SupAirOut})$$

Once the average heat transfer rates are determined, the average conditions of the supply air exiting the overall heat exchanger unit are calculated as follows:

$$T_{SupAirOut} = T_{SupAirIn} - \frac{\dot{Q}_{Sensible}}{\dot{m} c_{p, sup}}$$

$$h_{SupAirOut} = h_{SupAirIn} - \frac{\dot{Q}_{Total}}{\dot{m}_{SupAir}}$$

$$\omega_{SupAirOut} = PsyWFnTdbH(T_{SupAirOut}, h_{SupAirOut})$$

As described previously, if the predicted conditions of the exiting supply air exceed the saturation curve ($>100\%$ RH), then the temperature and humidity ratio of the air are reset to saturated conditions ($= 100\%$ RH) at the enthalpy condition calculated above ($h_{SupAirOut}$). If the supply air temperature is reset, the average sensible heat transfer rate is recalculated before the exhaust air outlet conditions are determined:

$$T_{ExhAirOut} = T_{ExhAirIn} + \frac{\dot{Q}_{Sensible}}{\dot{m} C_{p, exh}}$$

$$h_{ExhAirOut} = h_{ExhAirIn} + \frac{\dot{Q}_{Total}}{\dot{m}_{ExhAir}}$$

$$\omega_{ExhAirOut} = PsyWFnTdbH(T_{ExhAirOut}, h_{ExhAirOut})$$

Exhaust Air Recirculation

This method of frost control routes exhaust (outlet) air back through the supply side of the heat exchanger to warm the core. Since this method routes exhaust air back into the building, the building is typically not depressurized when this frost control is active. However, the incoming supply (outdoor ventilation) air flow is stopped for the fractional period of time that frost control is active. If significant periods of time exist when outdoor temperatures are below the selected threshold temperature and outdoor ventilation air is continuously required, an alternative method of frost control should be considered.

The user enters a threshold temperature, an initial defrost time fraction, and a rate of defrost time fraction increase. When the temperature of the inlet supply air (e.g., outdoor air) is equal to or below the specified threshold temperature, the fractional amount of time that this heat exchanger frost control strategy is active is determined from the following equation:

$$X_{DefrostTime} = X_{Initial} + X_{RateofIncrease} (T_{Threshold} - T_{SupAirIn})$$

The air mass flow rate of the supply air leaving the heat exchanger unit is then calculated using the defrost time fraction calculated above the mass flow rates of supply and exhaust air entering the unit.

$$\dot{m}_{SupAirOut} = (1 - X_{DefrostTime}) \dot{m}_{SupAirIn} + X_{DefrostTime} \dot{m}_{ExhAirIn}$$

The model assumes that no heat exchange occurs during defrost, and the average supply air conditions are simply a blend of the conditions when the unit is not in defrost and the exhaust air inlet conditions during defrost operation:

$$T_{SupAirOut} = \frac{(1 - X_{DefrostTime}) \dot{m}_{SupAirIn} T_{SupAirOut, NoDefrost} + X_{DefrostTime} \dot{m}_{ExhAirIn} T_{ExhAirIn}}{\dot{m}_{SupAirOut}}$$

$$\omega_{SupAirOut} = \frac{(1 - X_{DefrostTime}) \dot{m}_{SupAirIn} \omega_{SupAirOut, NoDefrost} + X_{DefrostTime} \dot{m}_{ExhAirIn} \omega_{ExhAirIn}}{\dot{m}_{SupAirOut}}$$

$$h_{SupAirOut} = PsyHFnTdbW(T_{SupAirOut}, \omega_{SupAirOut})$$

The operating effectivenesses of the heat exchanger are initially calculated according to Equations (449) and (450) assuming no defrost operation. Since the supply air flow across

the heat exchanger core is not reduced during defrost operation, the sensible and latent effectiveness are therefore derated (for reporting purposes) in direct proportion to the fraction of time that frost control is not active.

$$\mathcal{E}_{operating,sensible} = (1 - X_{DefrostTime}) \mathcal{E}_{operating,sensible}$$

$$\mathcal{E}_{operating,latent} = (1 - X_{DefrostTime}) \mathcal{E}_{operating,latent}$$

Since the exhaust outlet air is recirculated through the supply side of the heat exchanger core, the incoming supply air and exiting exhaust air flows are stopped for the fraction of the time when frost control is active. The average air mass flow rate at the supply air inlet and the exhaust air outlet nodes are therefore reduced accordingly.

$$\dot{m}_{SupAirIn} = (1 - X_{DefrostTime}) \dot{m}_{SupAirIn}$$

$$\dot{m}_{ExhAirOut} = (1 - X_{DefrostTime}) \dot{m}_{ExhAirOut}$$

The conditions of the exiting (outlet) exhaust air (temperature, humidity ratio and enthalpy) are reported as the values when frost control is not active (i.e., the conditions when exhaust air is actually leaving the unit).

Minimum Exhaust Temperature

With this frost control method, frost formation is avoided by continuously maintaining the temperature of the exhaust air leaving the heat exchanger core above a specified setpoint. The minimum exhaust air temperature is maintained by modulating heat exchanger rotational speed or by bypassing supply air around a plate heat exchanger. For this frost control method, the user must only enter the threshold (minimum) temperature.

For the case of modulating heat exchanger rotation, the operating effectivenesses and outlet air conditions are first calculated as if the heat exchanger is not in defrost mode (see Model Description). If the resulting temperature of the exhaust air leaving the heat exchanger core is below the specified threshold temperature, then the operating effectivenesses are reduced as follows:

$$X_{DefrostTime} = \frac{(T_{Threshold} - T_{ExhAirOut})}{(T_{ExhAirIn} - T_{ExhAirOut})}$$

$$\mathcal{E}_{operating,sensible} = (1 - X_{DefrostTime}) \mathcal{E}_{operating,sensible}$$

$$\mathcal{E}_{operating,latent} = (1 - X_{DefrostTime}) \mathcal{E}_{operating,latent}$$

The supply air and exhaust air outlet conditions are then recalculated using these reduced effectiveness values. Finally the sensible, latent and total heat recovery rates are calculated along with the unit's electric power and electric consumption.

The calculation procedure is slightly different for the case of a plate heat exchanger where the supply air is bypassed around the heat exchanger core. Since the volumetric air flow rate through the heat exchanger core is reduced when frost control is active, an iterative process is used to determine the operating effectiveness of the heat exchanger. The operating effectivenesses and outlet air conditions are first calculated as if the heat exchanger is not in defrost mode (see Model Description). If the resulting temperature of the exhaust air leaving

the heat exchanger core is below the specified threshold temperature, then the fractional defrost time is calculated as follows:

$$X_{DefrostTime} = \frac{(T_{Threshold} - T_{ExhAirOut})}{(T_{ExhAirIn} - T_{ExhAirOut})}$$

The iteration process then begins to determine the heat exchanger effectiveness and the exhaust air outlet temperature as if frost control were active. The operating mass flow rate through the supply side of the heat exchanger core is calculated.

Beginning of iteration process:

$$\dot{m}_{SupAirCore} = (1 - X_{DefrostTime}) \dot{m}_{SupAirIn}$$

$$\dot{m}_{SupAirBypass} = (X_{DefrostTime}) \dot{m}_{SupAirIn}$$

The ratio of average volumetric flow rate through the heat exchanger core to heat exchanger's nominal volumetric flow rate ($HX_{flowratio}$) is then determined and used to calculate the operating effectiveness of the heat exchanger using Equations (449) and (450). Since the supply airflow rate through the heat exchanger core is purposely reduced to control frost formation, average volumetric airflow rates below 50% of nominal are allowed and no warning message is issued. Supply air outlet temperature (leaving the heat exchanger core), sensible heat transfer, and exhaust air outlet temperature are then calculated using the revised heat exchanger effectiveness.

$$T_{SupAirOut} = T_{SupAirIn} + \mathcal{E}_{operating,sensible} \left(\frac{\dot{m} C_{p,min}}{\dot{m} C_{p,sup}} \right) (T_{ExhAirIn} - T_{SupAirIn})$$

$$\dot{Q}_{Sensible} = (1 - X_{DefrostTime}) \left(\dot{m} C_{p,sup} \right) (T_{SupAirInlet} - T_{SupAirOutlet})$$

$$T_{ExhAirOut} = T_{ExhAirIn} + \frac{\dot{Q}_{Sensible}}{\dot{m} C_{p,exh}}$$

The error between the exhaust outlet temperature and the threshold temperature for frost control and a new defrost time fraction are subsequently calculated.

$$Error = T_{ExhAirOut} - T_{Threshold}$$

$$X_{DefrostTime} = X_{DefrostTime} \left(\frac{T_{ExhAirIn} - T_{ExhAirOut}}{T_{ExhAirIn} - T_{Threshold}} \right)$$

End of iteration process:

The iteration process ends when the calculated error is within an error tolerance of 0.001. The air streams passing through the heat exchanger core and bypassing the core through the bypass damper are then blended together to provide the air conditions leaving the heat

exchanger unit. Finally the sensible, latent and total heat recovery rates are calculated along with the unit's electric power and electric consumption.

Economizer Operation

A conventional air-side economizer may be used in conjunction with this heat exchanger object. The air-side economizer is specified through the use of an outside air controller (see object: Controller:OutdoorAir). Specify the appropriate economizer control type, and provide the required control points and air flow rates as defined in the outside air controller object. Energy transfer provided by the heat exchanger will be suspended whenever free cooling is available (i.e., when the air-side economizer is activated) and *the user specified economizer lockout input is specified as Yes*. For plate heat exchangers, heat transfer is suspended by fully bypassing the supply and exhaust air around the heat exchanger core. For rotary heat exchangers, air flows continue through the core but it is assumed that heat exchanger rotation is stopped.

Heat exchange can also be suspended for the purposes of providing free cooling operation in the absence of a conventional air-side economizer. In this case specify "MinimumFlowWithBypass" as the economizer choice and again provide the required control points as defined in the outside air controller object. Energy transfer provided by the heat exchanger will be suspended whenever free cooling is available and *the user specified economizer lockout input is specified as Yes*, however the supply air flow rate will remain at the minimum value specified in the outside air controller object.

If economizer operation is not required, specify "NoEconomizer" as the economizer control type in the outside air controller object. The heat exchanger will operate according to its availability schedule and free cooling will not be provided. . If economizer operation *is* required and the heat exchanger *should not* provide free cooling, specify the input for the heat exchanger's economizer lockout as No and heat recovery will remain active during economizer mode.

Heat recovery for this heat exchanger may also be suspended during a high humidity control event (see object Controller:OutdoorAir) in a similar manner. Specifying Yes for economizer lockout will also suspend heat recovery when high humidity control is activated. The default value for economizer lockout is Yes and must be specifically entered as No to disable the economizer or high humidity control lockout feature.

Supply Air Outlet Temperature Control

This heat exchanger object can also control the supply air outlet temperature to a setpoint to avoid overheating. This temperature control is accomplished through wheel speed modulation or bypassing supply air around the heat exchanger. To model this temperature control, the user must specify 'Yes' for the Supply Air Outlet Temperature Control field in this heat exchanger object, and a separate setpoint manager (see object: SetpointManager:Scheduled) and temperature schedule must be specified for the heat exchanger unit's supply air outlet node.

This control strategy is typically used in conjunction with economizer operation (see object Controller:OutdoorAir), and an example control profile is shown in the figure below. When the outdoor air temperature falls to the specified maximum limit for economizer operation, heat exchange is suspended (air is fully bypassed around the heat exchanger core or heat exchanger rotation is stopped). The figure below shows economizer operation being initiated based on outdoor temperature but other triggers can be used (e.g. differential temperature [outdoor temperature with respect to exhaust air temperature], single point enthalpy or differential enthalpy). Heat exchange remains suspended until the outdoor temperature falls to the minimum temperature (temperature lower limit) for economizer control. The setpoint for the supply air outlet temperature control should match the economizer temperature lower limit.

As the outdoor air temperature falls further below the setpoint for the supply air outlet temperature (same as the economizer lower temperature limit), the heat exchanger bypass

dampers will modulate closed to maintain the desired supply air temperature for a plate heat exchanger. For a rotary heat exchanger the rotary heat exchanger speed will gradually increase to maintain the desired supply air temperature. Modulation of heat exchanger performance will continue until the supply air temperature setpoint can no longer be maintained

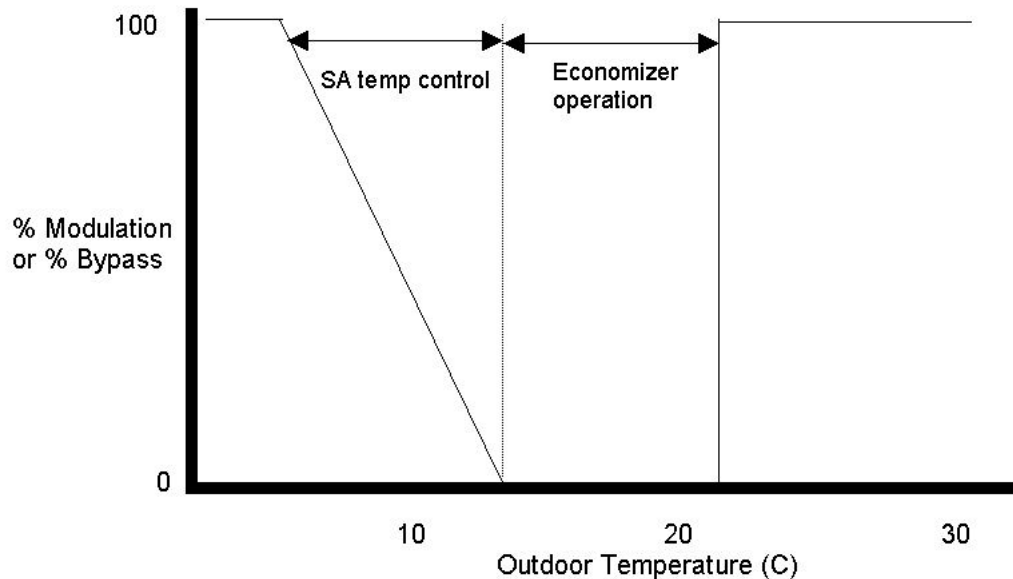


Figure 186. Air to Air Heat Exchanger with Supply Air Temperature Control

References

- ARI. 2001. Rating Air-to-Air Heat Exchangers for Energy Recovery Ventilation Equipment. Arlington, Virginia: Air-Conditioning & Refrigeration Institute.
- ARI. 2003. Certified Product Directory for Air-to-Air Energy Recovery Ventilation Equipment. Arlington, Virginia: Air-Conditioning & Refrigeration Institute.

Air System Air-To-Air Flat Plate Heat Exchanger

Overview

The input object HeatExchanger:AirToAir:FlatPlate provides an NTU – effectiveness model of a static flat plate air-to-air heat exchanger. Humidity issues are ignored in this model. A full, detailed description of the model can be found in the reference.

The inputs for the model are the design values for the primary air flow rate, primary air inlet and outlet temperature, secondary air flow rate, and secondary air inlet temperature. No design UAs are required: instead, the ratio (at design conditions) of the primary hA to the secondary hA (r_{hA}) is input. The flow configuration is also an input (counter, parallel, or crossflow).

- An important feature of this model is that the heat exchanger total UA is a time varying function of the primary and secondary mass flow rates and inlet temperatures.

Model Description

The design inlet / outlet conditions determine a design effectiveness eff_{des} . From the design capacity flow ratio, the flow arrangement, and eff_{des} , the NTU – effectiveness formulas give the NTU_{des} and UA_{des} .

The time varying calculations proceed as follows. First the UA is determined:

$$UA = UA_{des} (r_{hA} + 1) / ((\dot{m}_{p,des} T_{p,des} / \dot{m}_p T_p)^{78} + r_{hA} (\dot{m}_{s,des} T_{s,des} / \dot{m}_s T_s)^{78})$$

where *des* means *design*, *p* means *primary*, *s* means *secondary*, T is air stream temperature, and \dot{m} is air stream mass flow rate. From the UA and the capacity flow ratio the NTU is determined: $NTU = UA / C_{min}$. Then the NTU – effectiveness formulas are used to calculate the effectiveness. From the effectiveness and the inlet conditions, outlet conditions are determined.

Economizer Operation

A conventional air-side economizer may be used in conjunction with this heat exchanger object. The air-side economizer is specified through the use of an outside air controller (see object: Controller:OutdoorAir). Specify the appropriate economizer control type, and provide the required control points and air flow rates as defined in the outside air controller object. Energy transfer provided by the heat exchanger will be suspended whenever free cooling is available (i.e., when the air-side economizer is activated) or high humidity control is active and *the user specified economizer lockout input is specified as Yes*. For this flat plate heat exchanger, heat transfer is suspended by fully bypassing the supply and exhaust air around the heat exchanger core. If the economizer lockout is specified as No, the flat plate heat exchanger is operational even when economizer or high humidity mode is active. The default value for economizer lockout is Yes and must be specifically entered as No to disable the economizer or high humidity control lockout feature.

References

M. Wetter. 1999. *Simulation Model: Air-To-Air Plate Heat Exchanger*, LBNL-42354. This document can be downloaded from <http://simulationresearch.lbl.gov>.

Air System Air-To-Air Balanced Flow Desiccant Heat Exchanger

Overview

The input object HeatExchanger:Desiccant:BalancedFlow provides a model for a desiccant heat exchanger that is an HVAC component used to model both temperature (sensible) and moisture (latent) heat exchange between two air streams (Figure 187). The model assumes balanced air flow through the regeneration and process sides of the heat exchanger (i.e., regeneration and process air volume flow rates and face velocities are the same). Heat exchanger performance is specified through a performance data type object (e.g., Heat Exchanger:Desiccant:BalancedFlow:Performance Data Type 1). Refer to the EnergyPlus Input Output Reference for the specific input syntax for this desiccant heat exchanger object.

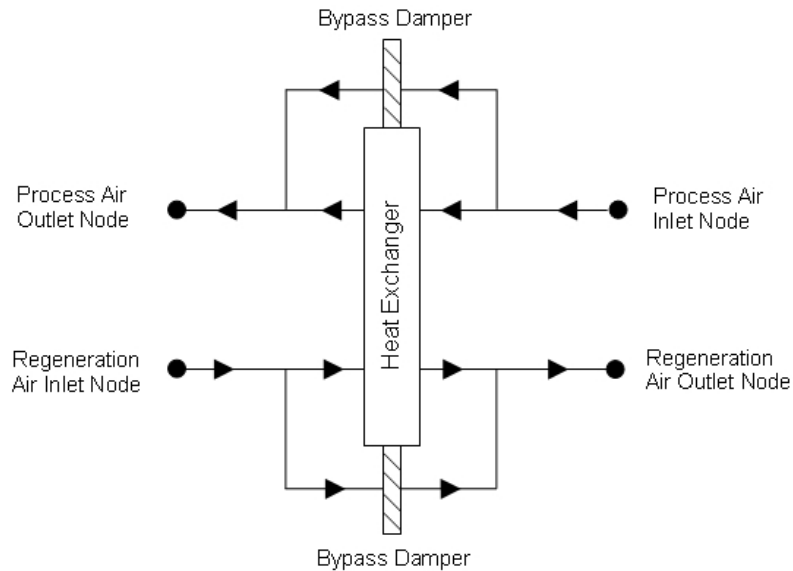


Figure 187. Schematic of the Balanced Flow Desiccant Heat Exchanger

EnergyPlus has another air-to-air heat exchanger object for modeling sensible and latent heat transfer between two air streams that uses the input object `HeatExchanger:AirToAir:SensibleAndLatent`. That heat exchanger model uses effectiveness values specified by the user to determine exiting air conditions from each side of the heat exchanger. In contrast, the balanced flow desiccant heat exchanger references a performance data type object (e.g., `HeatExchanger:Desiccant:BalancedFlow:PerformanceDataType1`) which defines the model equations, user-specified model coefficients, and min/max limits for the model's independent and dependent variables.

Model Description

This heat exchanger is a simple object that performs three basic functions:

- Informs the associated performance data type object (e.g., `HeatExchanger:Desiccant:BalancedFlow:PerformanceDataType1`) if it should calculate heat exchange for a simulation time step, or if heat exchange is suspended for the time step (i.e., bypass air flow around a fixed-plate heat exchanger or stop the rotation of a rotary heat exchanger),
- Passes the process and regeneration air inlet conditions (e.g., dry-bulb temperature, humidity ratio, air mass flow rate) to the associated performance data type object, and
- Reports the total, sensible and latent cooling/heating rates and electric power based on the results from the performance data type model that is specified.

While the desiccant heat exchanger object's availability to provide heat exchange between the two air streams is determined by the user-specified availability schedule, other objects that call this heat exchanger object can also control its heat exchange during a simulation time step. Currently, this desiccant heat exchanger model can be referenced by two compound objects: `CoilSystem:Cooling:DX:HeatExchangerAssisted` and `Dehumidifier:Desiccant:System`, both of which are used to provide enhanced dehumidification over conventional systems. If this heat exchanger is referenced by a compound object, the compound object will control heat exchanger operation (i.e., tell the heat exchanger if heat exchange is needed or not for each simulation time step). Details for how the compound objects control exchanger operation are described elsewhere in this document (ref. `CoilSystem:Cooling:DX:HeatExchangerAssisted` and `Dehumidifier:Desiccant:System`).

This desiccant heat exchanger object may also be specified directly in a AirLoopVHAC (air loop BranchList) or in an AirLoopHVAC:OutdoorAirSystem:EquipmentList without being referenced by a compound object. If specified directly in a AirLoopHVAC loop or AirLoopHVAC:OutdoorAirSystem:EquipmentList, then the heat exchanger can be controlled to provide heat exchange based on a maximum and/or minimum humidity setpoint placed on the process air outlet node (ref. SetpointManagers). If no humidity setpoints are provided on this node, then heat exchange will be provided whenever the heat exchanger is available to operate (via its availability schedule) and there is a temperature and/or humidity ratio difference between the two air streams. Further details regarding heat exchanger control via humidity setpoints on the process air outlet node are described in the section for the associated data type object (e.g., HeatExchanger:Desiccant:BalancedFlow:PerformanceDataType1).

The balanced flow desiccant heat exchanger model first checks for three conditions to be true before calling the specified performance data type model to calculate heat exchanger performance:

- Non-zero air mass flow rates on the process and regeneration inlet air nodes,
- Desiccant heat exchanger is available to operate based on its availability schedule,
- If a compound object is calling this desiccant heat exchanger, it is requesting that heat exchange be provided.

If any of these conditions is false, then heat exchange is suspended and the model simply passes the air conditions on the process and regeneration air inlet nodes to the respective outlet air nodes. In addition, the heat exchanger electric power is set to zero.

If all of the above conditions are true, then heat exchange is active and the specified performance data type model is called to calculate the process and regeneration outlet air conditions and heat exchanger electric power. Immediately before this call to the performance data type model, a check is made to determine if the operating air flow rates through the heat exchanger (i.e., the actual air flow rates for the simulation time step) are equal (balanced flow). If a difference of more than 2% exists between the process and regeneration air flow rates at any time during the simulation, a warning is issued.

After the specified performance data type model calculates the process/regeneration air outlet conditions and heat exchanger electric power, the balanced flow desiccant heat exchanger model uses that information to report overall performance. Specifically, the heat exchanger's sensible, latent and total heating and cooling rates are calculated for the process air side of the heat exchanger. Since energy must be conserved, the same heating or cooling rates apply to the regeneration air stream although they are opposite in heat transfer direction (e.g., a sensible cooling rate on the process air side of the heat exchanger would indicate an equivalent sensible heating rate on the regeneration air side).

$$\dot{Q}_{Sensible} = \dot{m}_{Proc} C_{p,Proc,in} (T_{Proc,out} - T_{Proc,in})$$

$$\dot{Q}_{Total} = \dot{m}_{Proc} (h_{Proc,out} - h_{Proc,in})$$

$$\dot{Q}_{Latent} = \dot{Q}_{Total} - \dot{Q}_{Sensible}$$

where:

$$\dot{Q}_{Sensible} = \text{sensible heat transfer rate to the process air stream (W)}$$

$$\dot{Q}_{Total} = \text{total heat transfer rate to the process air stream (W)}$$

\dot{Q}_{Latent}	= latent heat transfer rate to the process air stream (W)
\dot{m}_{Proc}	= process air mass flow rate (kg/s)
$C_{P_{Proc, in}}$	= specific heat of inlet process air (J/kg-K)
$T_{Proc, out}$	= process air outlet temperature (°C)
$T_{Proc, in}$	= process air inlet temperature (°C)
$h_{Proc, out}$	= process air outlet enthalpy (J/kg)
$h_{Proc, in}$	= process air inlet enthalpy (J/kg)

To simplify the accounting of heat exchanger performance, the sensible, latent, and total heat transfer rates for the process side of the heat exchanger are stored in cooling and heating report variables. For example, if the sensible heat transfer rate is negative, the absolute value is stored in a “cooling” rate report variable. Conversely, if the sensible heat transfer rate is positive, the value is stored in a “heating” rate report variable. Similar accounting is performed for the latent and total heat transfer rate variables as follows:

$$IF\left(\dot{Q}_{Sensible} \leq 0\right)\dot{Q}_{SensibleCooling} = ABS\left(\dot{Q}_{Sensible}\right)$$

$$IF\left(\dot{Q}_{Sensible} > 0\right)\dot{Q}_{SensibleHeating} = \dot{Q}_{Sensible}$$

$$IF\left(\dot{Q}_{Latent} \leq 0\right)\dot{Q}_{LatentCooling} = ABS\left(\dot{Q}_{Latent}\right)$$

$$IF\left(\dot{Q}_{Latent} > 0\right)\dot{Q}_{LatentHeating} = \dot{Q}_{Latent}$$

$$IF\left(\dot{Q}_{Total} \leq 0\right)\dot{Q}_{TotalCooling} = ABS\left(\dot{Q}_{Total}\right)$$

$$IF\left(\dot{Q}_{Total} > 0\right)\dot{Q}_{TotalHeating} = \dot{Q}_{Total}$$

At the end of each HVAC simulation time step, this object reports the sensible, latent and total cooling/heating energy and electric consumption for the heat exchanger as follows:

$$Q_{SensibleCooling} = \dot{Q}_{SensibleCooling} * TimeStepSys * 3600.$$

$$Q_{LatentCooling} = \dot{Q}_{LatentCooling} * TimeStepSys * 3600.$$

$$Q_{TotalCooling} = \dot{Q}_{TotalCooling} * TimeStepSys * 3600.$$

$$Q_{SensibleHeating} = \dot{Q}_{SensibleHeating} * TimeStepSys * 3600.$$

$$Q_{LatentHeating} = \dot{Q}_{LatentHeating} * TimeStepSys * 3600.$$

$$Q_{TotalHeating} = \dot{Q}_{TotalHeating} * TimeStepSys * 3600.$$

$$E_{HXUnit} = P_{HXUnit} * TimeStepSys * 3600.$$

where:

$Q_{SensibleCooling}$ = output variable 'Heat Exchanger Sensible Cooling Energy, J'

$\dot{Q}_{SensibleCooling}$ = output variable 'Heat Exchanger Sensible Cooling Rate, W' = $\dot{Q}_{Sensible}$ during times when the process air is cooled

$TimeStepSys$ = HVAC system simulation time step, hr

$Q_{LatentCooling}$ = output variable 'Heat Exchanger Latent Cooling Energy, J'

$\dot{Q}_{LatentCooling}$ = output variable 'Heat Exchanger Latent Cooling Rate, W' = \dot{Q}_{Latent} during times when the process air is dehumidified

$Q_{TotalCooling}$ = output variable 'Heat Exchanger Total Cooling Energy, J'

$\dot{Q}_{TotalCooling}$ = output variable 'Heat Exchanger Total Cooling Rate, W' = \dot{Q}_{Total} during times when the process air enthalpy is reduced

$Q_{SensibleHeating}$ = output variable 'Heat Exchanger Sensible Heating Energy, J'

$\dot{Q}_{SensibleHeating}$ = output variable 'Heat Exchanger Sensible Heating Rate, W' = $\dot{Q}_{Sensible}$ during times when the process air is heated

$Q_{LatentHeating}$ = output variable 'Heat Exchanger Latent Heating Energy, J'

$\dot{Q}_{LatentHeating}$ = output variable 'Heat Exchanger Latent Heating Rate, W' = \dot{Q}_{Latent} during times when the process air is humidified

$Q_{TotalHeating}$ = output variable 'Heat Exchanger Total Heating Energy, J'

$\dot{Q}_{TotalHeating}$ = output variable 'Heat Exchanger Total Heating Rate, W' = \dot{Q}_{Total} during times when the process air enthalpy is increased

E_{HXUnit} = output variable 'Heat Recovery Electric Consumption, J'

P_{HXUnit} = output variable 'Heat Recovery Electric Power, W'

Economizer Operation

A conventional air-side economizer may be used in conjunction with this heat exchanger object. The air-side economizer is specified through the use of an outside air controller (see

object: Controller:OutdoorAir). Specify the appropriate economizer control type, and provide the required control points and air flow rates as defined in the outside air controller object. Energy transfer provided by the heat exchanger will be suspended whenever free cooling is available (i.e., when the air-side economizer is activated) or high humidity control is active and *the user specified economizer lockout input is specified as Yes*. For the desiccant heat exchanger, heat transfer is assumed to be provided by a rotating heat exchanger core. For this reason, air continues to flow through the core when the outdoor air economizer is active but it is assumed that the rotation of the heat exchanger core is stopped. If the economizer lockout is specified as No, the desiccant heat exchanger is operational even when economizer or high humidity mode is active. This specific type of heat exchanger is typically *not* controlled by an outdoor air economizer. For this reason, the default value for economizer lockout is No and must be specifically entered as Yes to enable the economizer or high humidity control lockout feature.

Desiccant Heat Exchanger Performance Data

Overview

The input object HeatExchanger:Desiccant:BalancedFlow:PerformanceDataType1 specifies a performance model and model coefficients for a balanced flow desiccant heat exchanger. The HeatExchanger:Desiccant:BalancedFlow:PerformanceDataType1 object is referenced by a HeatExchanger:Desiccant:BalancedFlow object. This performance data object is used to specify the thermal performance and electric consumption of the heat exchanger. Some representative inputs for this object are provided in the EnergyPlus Reference DataSets (PerfCurves.idf).

This model predicts the regeneration air stream outlet temperature and humidity ratio values based on the entering regeneration and process air stream temperature, humidity ratio and face velocity. The process air stream outlet humidity ratio and temperatures are calculated based on a simple heat and moisture balance. The model requires that the user enter the nominal volumetric flow rate and a nominal face velocity, electric power consumption, empirical model coefficients for the regeneration outlet air temperature and humidity ratio equations, and the applicable minimum and maximum values for both the independent and dependent variables for the empirical model coefficients provided. Refer to the EnergyPlus Input Output Reference for details regarding the input syntax for this object.

Heat Exchanger Face Area and Air Velocity

The user is required to enter a nominal volumetric air flow rate and a nominal face velocity. From these inputs, a heat exchanger face area (applicable for both the regeneration and process sides of the heat exchanger) is calculated and used to determine the operating face velocity during the simulation.

$$A_{face} = \frac{V_{face,nom}}{\dot{v}_{face,nom}}$$

$$RFV = \frac{\dot{m}_{Reg,in}}{(\rho_{Std} A_{face})}$$

where:

A_{face} = heat exchanger face area (m²)

$V_{face,nom}$ = nominal air volume flow rate specified for the heat exchanger (m³/s)

$\dot{v}_{face,nom}$ = nominal air face velocity specified for the heat exchanger (m/s)

- RFV = face velocity of the regeneration (and process) air stream (m/s)
- $\dot{m}_{Reg,in}$ = mass flow rate of the regeneration air stream (kg/s)
- ρ_{Std} = density of air at standard temperature and pressure [dry air at 20°C] (m³/kg)

The face velocity calculated each simulation time step is used in the empirical equations (see 'Model Calculations' below) and should be within the minimum and maximum velocity boundaries specified for the model coefficients (see 'Empirical Model Boundaries (Minimum and Maximum)' below). When the calculated air velocity exceeds one of the boundaries, a warning is issued and the velocity is reset to the appropriate boundary value before being passed to the empirical equations for calculating regeneration air outlet temperature and humidity ratio. If the user is confident in their empirical model coefficients, the minimum and maximum velocity boundaries may be expanded slightly (caution should be used here) to allow extrapolation of the empirical equations during the simulation.

Model Calculations

The model coefficients may be obtained by curve fitting field measurements or the results from other computer models that accurately reflect the performance of a balanced flow desiccant heat exchanger. A wide range of data is necessary to properly define the performance of the desiccant heat exchanger such that all operating conditions expected during a simulation are included in the range of data used to obtain the model coefficients. The minimum and maximum boundaries for the independent variables used to generate the empirical model coefficients are used by this model to ensure that model extrapolation does not occur at any point during the simulation. For this reason, it is recommended that the widest possible range of data be used to determine the model coefficients.

The dry-bulb temperature of the regeneration outlet air is determined using the equation shown below.

$$RTO = B1 + B2 * RWI + B3 * RTI + B4 * \left(\frac{RWI}{RTI} \right) + B5 * PWI + B6 * PTI + B7 * \left(\frac{PWI}{PTI} \right) + B8 * RFV$$

where:

RTO = regeneration outlet air dry-bulb temperature (°C)

RWI = regeneration inlet air humidity ratio (kg/kg)

RTI = regeneration inlet air dry-bulb temperature (°C)

PWI = process inlet air humidity ratio (kg/kg)

PTI = process inlet air dry-bulb temperature (°C)

RFV = regeneration (and process) face velocity (m/s)

Similarly, the humidity ratio of the regeneration outlet air is defined using the same empirical equation form; however, different coefficients are used as follows:

$$RWO = C1 + C2 * RWI + C3 * RTI + C4 * \left(\frac{RWI}{RTI} \right) + C5 * PWI + C6 * PTI + C7 * \left(\frac{PWI}{PTI} \right) + C8 * RFV$$

where:

RWO = regeneration outlet air humidity ratio (kg/kg)

If the regeneration outlet air conditions exceed the saturation curve, RTO and RWO are reset to saturated conditions (100% RH) at the enthalpy calculated based on the original RTO and RWO values.

Once the regeneration outlet air conditions are determined as described above, the dry-bulb temperature and humidity ratio differences across the regeneration side of the heat exchanger are calculated.

$$T_{diff} = RTO - T_{Reg, in}$$

$$w_{diff} = RWO - w_{Reg, in}$$

where:

$T_{Reg, in}$ = actual regeneration inlet air dry-bulb temperature (°C)

$w_{Reg, in}$ = actual regeneration inlet air humidity ratio (kg/kg)

T_{diff} = regeneration air temperature difference based on empirical model (°C)

w_{diff} = regeneration air humidity ratio difference based on empirical model (kg/kg)

The regeneration outlet air conditions described above represent the full load outlet conditions under steady-state operation. However, there are times when the heat exchanger may not operate for the entire simulation time step (i.e., bypass air flow around a fixed-plate heat exchanger or stop the rotation of a rotary heat exchanger for a portion of the time step). For example, the parent object calling this heat exchanger model may request that it only provide heat exchange for a portion of the time step (*HXPartLoadRatio*). Another example would be if this heat exchanger is not called by a parent object but is instead placed directly in a AirLoopHVAC or AirLoopHVAC:OutdoorAirSystem and a setpoint manager is used to place a minimum and/or maximum humidity ratio setpoint on the process air outlet node. For this case the humidity setpoints, if present, are used to calculate a part-load ratio for the heat exchanger assuming the full-load process air humidity ratio difference is equivalent (but opposite in sign) to the regeneration air humidity ratio difference (w_{diff}):

IF ($w_{diff} > 0.0$) *THEN* ! *Regen air humidified, process air dehumidified*

$$HXPartLoadRatio = \frac{(w_{Proc, in} - w_{max, set point})}{w_{diff}}$$

ELSE IF ($w_{diff} < 0.0$) *THEN* ! *Regen air dehumidified, process air humidified*

$$HXPartLoadRatio = \frac{(w_{Proc, in} - w_{min, set point})}{w_{diff}}$$

END IF

$HXPartLoadRatio = MAX(0.0, HXPartLoadRatio)$

$HXPartLoadRatio = MIN(1.0, HXPartLoadRatio)$

If this heat exchanger is not being called by a parent object and no humidity setpoints are placed on the process air outlet node, then the model assumes that the heat exchanger operates for the entire simulation time step when it is available to operate (based on its

availability schedule) and there is a temperature and/or humidity ratio difference between the two air streams.

EnergyPlus has an established convention for placing information on outlet air nodes. If the air flow rate is continuous but the device only operates for a portion of the simulation time step, then the average outlet conditions (temperature, humidity and enthalpy) are placed on the outlet air node. If the air flow rate cycles on and off during the simulation time step, then the full load outlet conditions are placed on the outlet air node along with the average air mass flow rate for the simulation time step. To account for these cases, this model uses the following logic:

IF (FanOpMode .EQ. CycFanCycComp .OR. RegAirInletIsOANode) THEN

$$T_{Reg, out} = T_{Reg, in} + T_{diff}$$

$$W_{Reg, out} = W_{Reg, in} + W_{diff}$$

ELSE

$$T_{Reg, out} = T_{Reg, in} + (T_{diff} * HXPartLoadRatio)$$

$$W_{Reg, out} = W_{Reg, in} + (W_{diff} * HXPartLoadRatio)$$

ENDIF

where:

$$T_{Reg, out} = \text{regeneration outlet air dry-bulb temperature (}^{\circ}\text{C)}$$

$$W_{Reg, out} = \text{regeneration outlet air humidity ratio (kg/kg)}$$

HXPartLoadRatio = heat exchanger part-load ratio (determined by parent object calling this model, or calculated based on a minimum/maximum humidity ratio setpoints as shown above).

In the logic shown above, *RegAirInletIsOANode* is true if the regeneration air inlet node is an outside air node. If so, the model assumes that the air flow through the regeneration side of the heat exchanger cycles on and off as required during the simulation time step. Also, the regeneration outlet air humidity ratio is limited to be between 1E-5 and 1.0.

The regeneration outlet air enthalpy is then calculated using the regeneration outlet air temperature and humidity ratio.

$$h_{Reg, out} = PsyHFdTdbW(T_{Reg, out}, W_{Reg, out})$$

Using the regeneration outlet air conditions, the heat transfer on the regeneration side of the heat exchanger is then calculated:

$$\dot{Q}_{Sensible, Reg} = \dot{m}_{Reg} C_{p, Reg, in} (T_{Reg, out} - T_{Reg, in})$$

$$\dot{Q}_{Total, Reg} = \dot{m}_{Reg} (h_{Reg, out} - h_{Reg, in})$$

where:

$$\dot{Q}_{Sensible, Reg} = \text{sensible heat transfer rate to the regeneration air stream (W)}$$

$\dot{Q}_{Total,Reg}$ = total heat transfer rate to the regeneration air stream (W)

\dot{m}_{Reg} = regeneration air mass flow rate (kg/s)

$Cp_{Reg,in}$ = specific heat of inlet regeneration air (J/kg-K)

Since the model assumes that total and sensible heat transfer is conserved, these heat transfer rates are then used to calculate the process air outlet conditions:

$$h_{Proc,out} = h_{Proc,in} - \left(\frac{\dot{Q}_{Total,Reg}}{\dot{m}_{Proc}} \right)$$

$$T_{Proc,out} = T_{Proc,in} - \left(\frac{\dot{Q}_{Sensible,Reg}}{\dot{m}_{Proc} Cp_{Proc,in}} \right)$$

$$w_{Proc,out} = PsyWFnTdbH(T_{Proc,out}, h_{Proc,out})$$

where:

$h_{Proc,out}$ = process outlet air enthalpy (J/kg)

$T_{Proc,out}$ = process outlet air dry-bulb temperature (°C)

$w_{Proc,out}$ = process outlet air humidity ratio (kg/kg)

\dot{m}_{Proc} = process air mass flow rate (kg/s)

$Cp_{Proc,in}$ = specific heat of inlet process air (J/kg-K)

Like the regeneration outlet air conditions, the process outlet air conditions are also checked for exceeding saturated conditions and, if detected, the temperature and humidity ratio are reset assuming constant enthalpy.

$$T_{sat} = PsyTsatFnHPb(h_{Proc,out}, PB)$$

IF ($T_{sat} > T_{Proc,out}$) THEN

$$T_{Proc,out} = T_{sat}$$

$$w_{Proc,out} = PsyWFnTdbH(T_{Proc,out}, h_{Proc,out})$$

END IF

Heat recovery electric power is the electric consumption rate of the unit in watts. The nominal electric power for the heat exchanger is specified in the input data file, and can be used to model controls (transformers, relays, etc.) and/or a motor for a rotary heat exchanger. The model assumes that this electric power is consumed whenever the heat exchanger operates. The electric power is assumed to be zero for all other times. None of this electric power is assumed to contribute thermal load to either of the heat exchanger air streams. As with the

thermal performance of the heat exchanger, the power used by the heat exchanger is also proportional to the heat exchanger part load ratio.

$$P_{HXUnit} = P_{HX,nom} (HXPartLoadRatio)$$

where:

P_{HXUnit} = output variable 'Heat Recovery Electric Power, W' reported by the HeatExchanger:Desiccant:BalancedFlow object.

$P_{HX,nom}$ = user specified 'Nominal Electric Power, W'

Empirical Model Boundaries (Minimum and Maximum)

Since this model uses empirical equations and extrapolation can cause unexpected results, the entering air conditions to both the regeneration and process sides of the heat exchanger are checked during each simulation time step to make sure they are within the valid range defined by the user. If any of the independent variables used in the equations for RTO and RWO above are outside the minimum and maximum limits specified for each independent variable, the values for these specific independent variables are limited to the appropriate boundary (min/max) value for use by the empirical equations and a warning is issued.

Additional model checks occur by testing the regeneration and process inlet air relative humidities. In this case, the process and regeneration inlet air relative humidities are calculated and compared to the minimum/maximum values specified by the user. If the calculated relative humidity exceeds the specified range at any point during the simulation when the heat exchanger is operating, a warning message is issued but the simulation proceeds.

If the user is confident in their empirical model coefficients, the minimum and maximum independent variable boundaries may be expanded (caution should be used here) to allow extrapolation of the empirical equations during the simulation.

In addition to checking the inlet air conditions, the model allows the user to specify limits for the regeneration air outlet conditions (RTO and RWO). If the calculated value for RTO or RWO exceeds the minimum/maximum limits specified by the user, then the calculated value is reset to the appropriate boundary (min/max) value and a warning message is issued.

This model also assumes that the performance of the desiccant heat exchanger (i.e., the output of the empirical equations) conforms to the following two guidelines.

- The regeneration outlet air temperature (RTO) is always less than or equal to the regeneration inlet air temperature (T_{RegIn})
- The regeneration outlet air humidity ratio (RWO) is always greater than or equal to the regeneration inlet air humidity ratio (w_{RegIn})

When the calculated regeneration outlet air conditions do not follow these guidelines, a warning is issued and the simulation proceeds.

Plant Loop Deep-Ground-To-Water Vertical U-Tube Field Heat Exchanger

This model (Object: GroundHeatExchanger:Vertical) documentation is derived from the M.S. thesis of Arunachalam Murugappan, which is available on the Oklahoma State University web site <http://www.hvac.okstate.edu/>. Eskilson (1987) Yavuzturk and Spitler (1999) developed the long and short time response factors respectively, which are used in determining the borehole temperature responses. Response factors are infinite series of numbers, which relate the current value of a variable to past values of other variables at discrete time intervals. These response factors are referred as *g*-functions. The variable

time-step model presented here uses both long time-step g -functions and short time-step g -functions to predict the boreholes response to short term fluctuations in the load.

Long Time-Step Response Factors

Eskilson developed g -functions for various borehole configurations. He plotted the g function curves for different sets of borehole spacing to borehole length ratio B/H typically for .05, 0.1, 0.15, 0.2, 0.3 and ∞ (∞ represents the single borehole configuration). All the plots were for the ratio of 0.0005 between the borehole radius and the borehole length r_b/H . For any other radius a simple relation between the two radii as given by Eskilson (1987) can be used. Eskilson gives the g -function curves for 38 different configurations.

Figure 188 shows the g -function plotted against the non-dimensional time defined as $\ln(t/t_s)$, $t_s = H^2/9\alpha$ for various configurations of vertical boreholes with B/H ratio of .1 along with single borehole. It is seen from this figure that the thermal interaction between boreholes increases with time and with the number of boreholes in field.

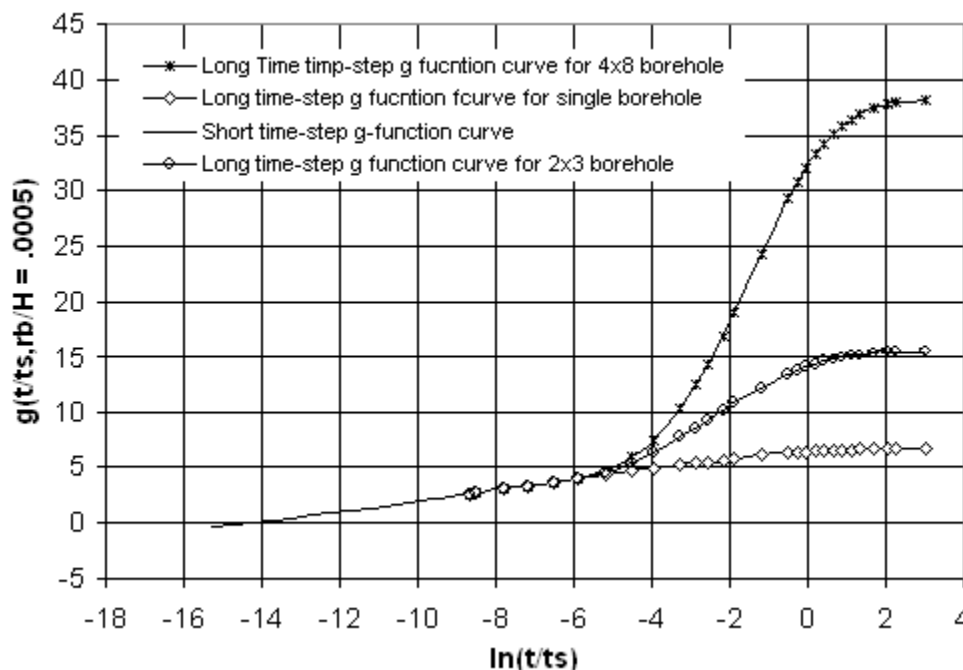


Figure 188. Short Time Step g Function Curve as an Extension of Long Time Step g Function Curves for Different Configuration of Boreholes (Eskilson 1987, Yavuzturk 1999).

The g -functions developed by Eskilson are valid only after time estimated by Eskilson as $5r_b^2/\alpha$. This time varies from 3-6 hours for a typical borehole field. This is because the analytical line source model, based on which the Eskilson's model was developed, does not give a prompt increase in borehole wall temperature at $r = r_b$. It gives acceptable results only after the non-dimensional times of $\alpha t/r_b^2 > 5$. But to model short time responses of a borehole we need response factors, which can give accurate results down to minutes.

Short Time-Step Response Factors

Yavuzturk and Spitler (1999) developed short time step response factors using a transient, two-dimensional, implicit finite volume model on a polar grid.

The circular u-tube pipe in the ground loop heat exchanger was approximated as a pie sector of equivalent perimeter. A constant heat flux for the heat transfer from/to U-tube, a zero heat

flux in the angular direction and a constant far field temperature in the radial axis makes up the three boundary condition and undisturbed far field temperature as the initial condition. The numerical model accounts for the thermal resistance due to individual borehole elements; such as resistance of the pipe and grout material and the convection resistance due to the heat transfer fluid in the pipes. The long time step g -functions discussed in the previous section do not account for such effects.

The short time-step g -functions are the same for different borehole configurations. This is because there is no thermal interaction between the boreholes for times less than 200 hrs during which the short time-step g -functions apply. So it is suggested to use the short time-step g function for time steps in the range of 2.5 min and 200 hours and the long time-step g -functions for time steps longer than 200 hours. The g function for any time can be found by linear interpolation between the bounding known values.

Development of the Variable Short Time Step Model

The variable time step model was developed as an extension of the model presented by Yavuzturk and Spitler (1999). The variable, short time step model uses a similar algorithm and extends it to accommodate sub-hourly responses, variable time steps and explicit equations to calculate the outlet fluid temperature of the ground loop heat exchanger.

The uniform time-steps model developed by Yavuzturk and Spitler (1999) is able to pre-calculate all the g -functions at the beginning of the simulation. The variable time-step model on the other hand must calculate the g -functions when the borehole response calculation for each time step is carried out. For every time step a different set of g -functions is needed in the variable time step model as the time at which the g -function is to be applied for the past loads changes for each time-steps.

This is made clear from the illustration in Figure 189, which shows a simulation in progress. The boxes with numbers represent the sub-hourly loads. The time (in hrs) at which these loads occurred are shown by solid arrows above the respective load boxes. The right-most solid arrow gives the current simulation time, which is 3.31 hrs. The times given below the boxes, pointed by dashed arrows, are the time at which the g -functions are to be estimated and applied to the respective sub hourly loads (boxes) for the current time.

For example, let us take the sub hourly loads 1,2 & 3. These loads occurred at 0 hrs, 0.16 hrs & 0.28 hrs. The response of the borehole temperature for the current time step is calculated by applying the g -functions at 3.15 hrs, 3.03 hrs & 2.5 hrs respectively. Thus to calculate the present borehole temperature, the sub hourly loads 1-12 are superposed using the corresponding g functions at times given by the dashed lines. This gives the borehole temperature at hr 3.31. However, for the previous time step, which occurred at 3.15 hrs, the g -functions for the loads 1, 2 & 3 are at 2.99 hrs 2.87 hrs and 2.42 hrs, and the over all response is obtained by superposing the loads 1-11.

Thus for each time step since the time step increments are not uniform we need to store the simulation times at which these time-steps occurred, and calculate corresponding g -functions at each time-step.

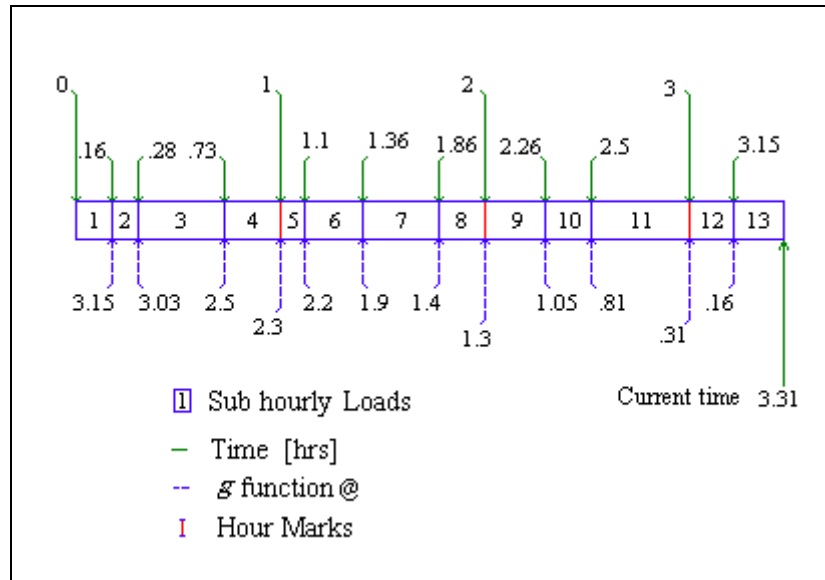


Figure 189. Variable Timestep Ground Loop Heat Exchanger Model Schematic Explaining the g Function Estimation.

Yavuzturk model calculates the outlet fluid temperature by iteration beginning with the undisturbed mean temperature of the surrounding ground as an initial guess. This aggravates the time taken by an already computationally intensive algorithm. To circumvent this a set of explicit equations were formulated to estimate the outlet fluid temperature.

Description of the Load Aggregation Scheme

A load aggregation scheme was developed for energy simulation software with variable short time steps down to one minute. A major issue in the development was the calculation of the g -functions. As discussed previously in the variable time step environments, the time step increments are not uniform. As a result, g -functions cannot be pre-calculated. Figure 190 shows a schematic of the variable time step ground loop heat exchanger model. The figure shows the larger monthly block loads, hourly loads and sub hourly loads and along with the time of occurrence of those loads. The figure also shows the time at which the g -functions are applied to different load blocks.

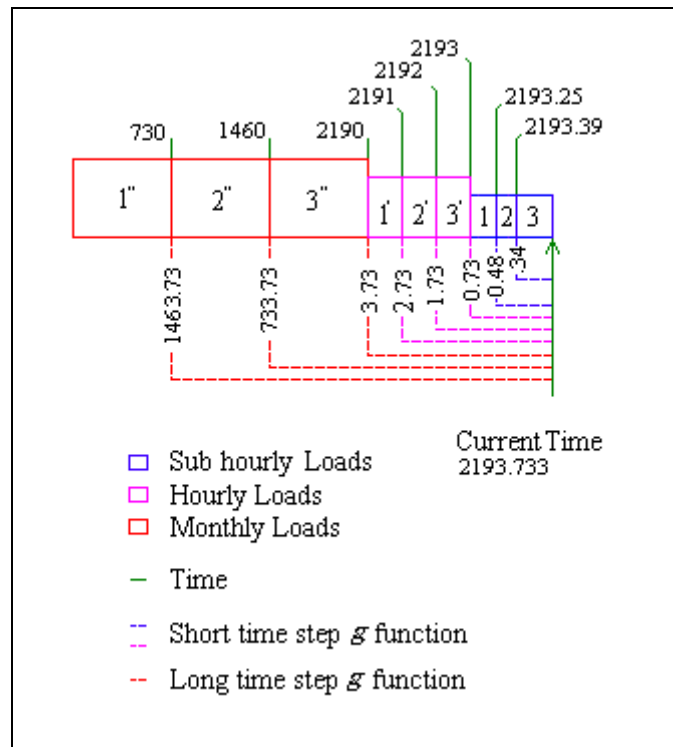


Figure 190. Schematic of Variable Time Step Model g Function Calculation.

To calculate the response of a past load on the borehole temperature we apply the g -function corresponding to the time elapsed since the load was applied. This is easily understood from the schematic. For example, to calculate the response of the aggregated load 1" (at the end of 730hrs.) for the current time step (2193.733hrs) we apply a g -function at 1463.73hrs. The g -function for the same block 1" at the previous time step, which occurred at 2193.25 hrs, would be at 1463.25hrs. From the schematic it is also seen that for the other two aggregated monthly loads 2", 3" the g -function are applied at 733.73 hrs and 3.73 hrs for the current time-step and at 733.25 hrs and 3.25 hrs respectively for the previous time-step. The same scheme applies to hourly and sub-hourly. Thus to estimate the time at which the past monthly, hourly or sub-hourly loads occur, we might be tempted to store the simulation times at each time step for the entire simulation, storing load times for the whole length of simulation for a multi year simulation with a variable short time step would require a large amount of memory. But little consideration shows that it is not necessary. Since the monthly and hourly loads occur at equal intervals of time 730hrs and 1hr respectively, the g -functions can be estimated with the current simulation time and the time at which the load block ends, which is a multiple of the monthly duration of the block size. Only the sub-hourly loads require storage of simulation times.

For example from the schematic (Figure 191), for the sub hourly load 1, which occurred at the end of 2193.25 a g -function at .48 hrs has to be applied and for the next load 2 a g -function at 0.34 hrs has to be applied. Since the time intervals are not even for the sub hourly loads, we need to store the time steps at which those loads occurred. These times are required to estimate the time elapsed between the current simulation time and the time at which the sub hourly loads occurred.

Thus, the algorithm keeps track of the sub hourly loads along with their time of occurrence for a user-defined length of time during which the sub hourly calculations are made. The algorithm also estimates the time weighted hourly load from their corresponding sub hourly loads as each hour passes. The sub-hourly loads are time weighted because of the irregular intervals at which the time-step occurs. This is also illustrated in Figure 191. The sub hourly loads 1,2 &3 occur for varying length of time. The load 3 occurs for a longer duration than 1

and 2 in that order. This implies that the load 3 has to be given more weight than 1 and 2. So the sub hourly loads for a particular hour are multiplied by the length of their respective period of occurrence and averaged over the hour. This is further explained by the schematic in Figure 5.

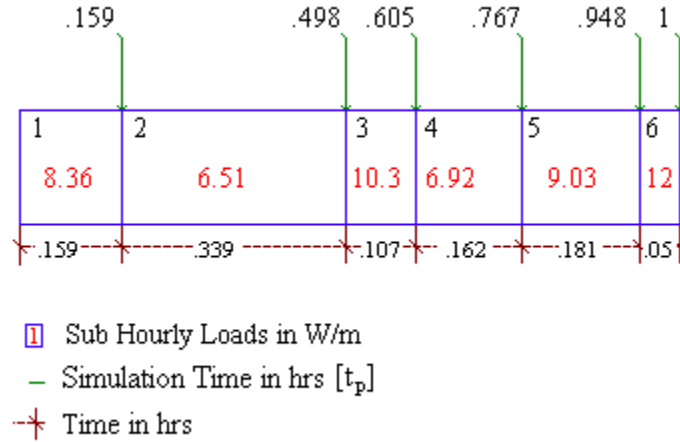


Figure 191. Schematic Showing the Calculation of Hourly Load from the Sub Hourly Loads.

The bottom text in the boxes represents the magnitude of the sub hourly loads in W/m for each time step. The duration of the occurrence of each time-step for the each block is shown below the respective block. The first hourly load is given by the expression (453)

$$\bar{q}_1 = \left[\begin{array}{l} (8.36 \times .159) + (6.51 \times .339) + (10.3 \times .107) \\ + (6.92 \times .162) + (9.03 \times .181) + (12 \times .05) \end{array} \right] = 7.993W / m \quad (453)$$

Where \bar{q}_1 = the first hourly load in W/m

The algorithm keeps track of enough of these past hourly loads to calculate the monthly load. As each month or user defined time passes, hourly loads over the entire month or user defined time “blocks” are averaged and stored in arrays for the respective monthly user defined block of time.

The borehole temperature for any time step is computed by superposing the monthly (larger time block loads) hourly and sub-hourly loads for each time step. To understand more clearly consider the schematic in Figure 191 where the borehole temperature at 2193.733 hour is to be estimated. Here the monthly block time is 730 hrs. We have three monthly aggregated load blocks for 730 hrs, 1460 hrs and 2190 hrs and hourly loads from 2191st hr to 2193rd hour. For the remaining 0.733 hours a sub hourly calculation is done. The three monthly aggregated load blocks when superposed using long time g -functions, yields the borehole temperature at the end of 2190th hour. Then the hourly loads from 2191st to 2193rd hrs are superposed using the corresponding short time step g functions values yielding the borehole temperature at the end of 2193rd hour. The sub-hourly variations for the current hour are obtained, by superposing the sub-hourly loads. From the schematic, we see there are two sub-hourly loads, 1 and 2. Thus the borehole temperature at the end of 2193.733 is expressed as:

$$\begin{aligned}
T_{2193.733} = & T_{ground} + \sum_{m=1}^3 \left[\frac{\bar{q}_m - \bar{q}_{m-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_{730(m-1)}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{n=2190}^{2193} \left[\frac{\bar{q}_n - \bar{q}_{n-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_{n-1}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{p=2193}^{2193.733} \left[\frac{q_p - q_{p-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \tag{454}
\end{aligned}$$

Where

\bar{q} = The average monthly loads

\bar{q} = The average hourly loads

Q = the sub-hourly loads

m = index for monthly aggregated blocks

p = array index for sub hourly loads

t = time

t_p = the sub hourly time steps over the history period. (here increment is not always unity)

Superposing the temperature responses of monthly (larger) blocks over the shorter, namely the hourly and sub hourly, introduces some error in the borehole temperature calculation at the beginning of every month. Yavuzturk and Spitler suggest a method to reduce the error in borehole temperature prediction by using a minimum hourly history period during which only the short time step superposition is carried out. In our model this idea is extended to sub hourly loads as well. Thus a user specified minimum sub-hourly history period is included along with the minimum hourly history period to model the sub-hourly variations. During this period only sub-hourly and hourly superpositions are made. This guarantees that at any given time step the superposition of temperature responses involves a minimum period of short time responses, which ensures a better estimation of borehole temperature. For example, a minimum hourly history period of 96 hrs and a minimum sub hourly history period of 5 hours would result in only 2 monthly aggregation blocks (1" and 2"). The last monthly aggregation does not occur because neither of the minimum hourly history period of 96 hours or sub-hourly history period of five hrs is met. So an hourly superposition of the load is carried out for the third month until the minimum sub-hourly history period after which sub hourly superposition is carried out. The equation becomes

$$\begin{aligned}
T_{2193.733} = & T_{ground} + \sum_{m=1}^2 \left[\frac{\bar{q}_m - \bar{q}_{m-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_{730(m-1)}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{n=1460}^{2188} \left[\frac{\bar{q}_n - \bar{q}_{n-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_{n-1}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{p=2188}^{2193.733} \left[\frac{q_p - q_{p-1}}{2\pi k_{ground}} g \left(\frac{t_{2193.733} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \tag{455}
\end{aligned}$$

Yavuzturk and Spitler have done a detailed analysis on the effect of minimum hourly history period. They found that a minimum hourly history period of 192 hrs for an annual simulation would reduce the running time by 90%. They also found that for a 20year simulation, the computation time of the aggregated load scheme is just 1% of the non-aggregated load scheme.

Summary of Variable Short Time Step Response Factor Model

The load aggregation scheme developed in line with the above example is summarized eight steps as follows:

1. Define monthly load blocks duration (mb) in hrs (generally 730 hrs) and the minimum hourly history period and minimum sub hourly history period.
2. Read Borehole Geometry Parameters: number of boreholes, borehole length radius thickness of the pipe etc. Read Ground and Fluid thermal properties: Ground conductivity, volumetric specific heat capacity of the ground and heat carrier fluid. Read the short and long time-step g -functions into arrays with their respective non-dimensionalized times.
3. Start Simulation from $p=1$ to nts . Here “nts” is the number of time steps that have occurred since the start of simulation. (Note that P is not a count of number of hour elapsed in the simulation)
4. Compute the hourly loads as each hour passes. This is done by averaging the sub hourly loads during the past hour. The monthly loads are calculated by averaging the hourly loads during that month. This is done by summing the hourly loads during that monthly period and dividing the sum by 730 hours NumMonths(the number of months used in aggregation calculations) is set to the number of months of simulation (current number of aggregated load blocks)
5. If the simulation time is less than the minimum sub hourly history period the borehole temperature is estimated with no aggregation. Only sub hourly loads are superposed as given by the following equation.

$$T_{nts} = T_{ground} + \sum_{p=1}^{nts} \left[\frac{q_p - q_{p-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \quad (456)$$

6. . If the simulation time is less than sum of minimum hourly history and sub hourly history periods, then decomposed hourly aggregated loads are superposed using their corresponding g function until the sub hourly history required. Then the sub hourly temperature differences are found by superposing the decomposed sub hourly loads with their short time step g functions. Average borehole temperature is found by superposing the hourly and sub hourly temperature differences with the following equation

$$T_{nts} = \sum_{n=1}^{nh-sh} \left[\frac{\bar{q}_n - \bar{q}_{n-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_{n-1}}{t_s}, \frac{r_b}{H} \right) \right] + \sum_{p=nts-sh}^{nts} \left[\frac{q_p - q_{p-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \quad (457)$$

7. If the simulation time is greater then the sum of a monthly period, sub hourly history and the hourly history period. Then monthly load aggregation is performed. Here if the difference between the simulation time and product of a monthly block period and the current number of monthly blocks is greater than the sum of the minimum hourly history and sub hourly history periods then the average borehole temperature is found by the following equation.

$$\begin{aligned}
T_{nts} = & T_{ground} + \sum_{m=1}^{calb} \left[\frac{\bar{q}_m - \bar{q}_{m-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_{mb[m-1]}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{n=nh-[calb(mb)+sh]}^{nh-sh} \left[\frac{\bar{q}_n - \bar{q}_{n-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_{n-1}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{p=nts-sh}^{nts} \left[\frac{q_p - q_{p-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \tag{458}
\end{aligned}$$

8. If the difference between the simulation time and product of a monthly block period and the current number of monthly blocks is less than the sum of the minimum hourly history and sub hourly history periods, then NumMonths is set to one month less than the actual number of months of simulation completed. The average borehole temperature is calculated by superposing the long and time step temperature differences using the following equation.

$$\begin{aligned}
T_{nts} = & T_{ground} + \sum_{m=1}^{calb-1} \left[\frac{\bar{q}_m - \bar{q}_{m-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_{mb[m-1]}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{n=nh-[(calb-1)(mb)+sh]}^{nh-sh} \left[\frac{\bar{q}_n - \bar{q}_{n-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_{n-1}}{t_s}, \frac{r_b}{H} \right) \right] \\
& + \sum_{p=nts-sh}^{nts} \left[\frac{q_p - q_{p-1}}{2\pi\kappa_{ground}} g \left(\frac{t_{nts} - t_p}{t_s}, \frac{r_b}{H} \right) \right] \tag{459}
\end{aligned}$$

Define Monthly block duration, minimum hourly history period and minimum sub hourly history period (step 1)

Read Borehole Geometry parameters, Ground and carrier fluid properties and read short and long time-step g functions. (Step 2)

Do until $p = 1$ to number of time-steps (nts) (Step 3)

Compute the Ground Load for the current time step using Entering Fluid temperature (For the first time-step $T_{FarField}$ may be used)

Compute the hourly loads as each hour and monthly loads as each month passes. Calculate the number of monthly blocks (NumMonths) (**Step 4**)

If (Current time less than minimum sub hourly history) (**Step 5**)
use Equation (456)

Else If (Current Simulation Time less than sum of minimum hourly history and minimum sub hourly history) (**step 6**)
use Equation (457)

Else

If (Difference between current simulation time and duration of the total number months is Greater than sum of minimum hourly and sub hourly histories) (**step 7**)
use Equation (458)

Else (step 8)

use Equation (459)

End if

```

End if
End do

```

References

Eskilson, P. 1987. Thermal Analysis of Heat Extraction Boreholes. Ph.D. Thesis, Department of Mathematical Physics, University of Lund, Lund, Sweden.

Yavuzturk, C. 1999. Modeling of Vertical Ground Loop Heat Exchangers for Ground Source Heat Pump Systems. Ph.D. Thesis, Department of Mechanical and Aerospace Engineering, Oklahoma State University, Stillwater, Oklahoma.

Yavuzturk, C., J.D. Spitler. 1999. [A Short Time Step Response Factor Model for Vertical Ground Loop Heat Exchangers](#). *ASHRAE Transactions*. 105(2):475-485.

Plant Loop Pond-To-Water Heat Exchanger

The pond model (Object: GroundHeatExchanger:Pond) is a 'lumped parameter' model where the pond is represented by a single node with thermal mass. The pond surface temperature is the same as the temperature at this node, i.e. the surface temperature is the same as the bulk temperature. A first order differential equation is solved in the model to calculate the pond temperature at each time step. This type of heat rejecter is modeled as several circuits connected in parallel.

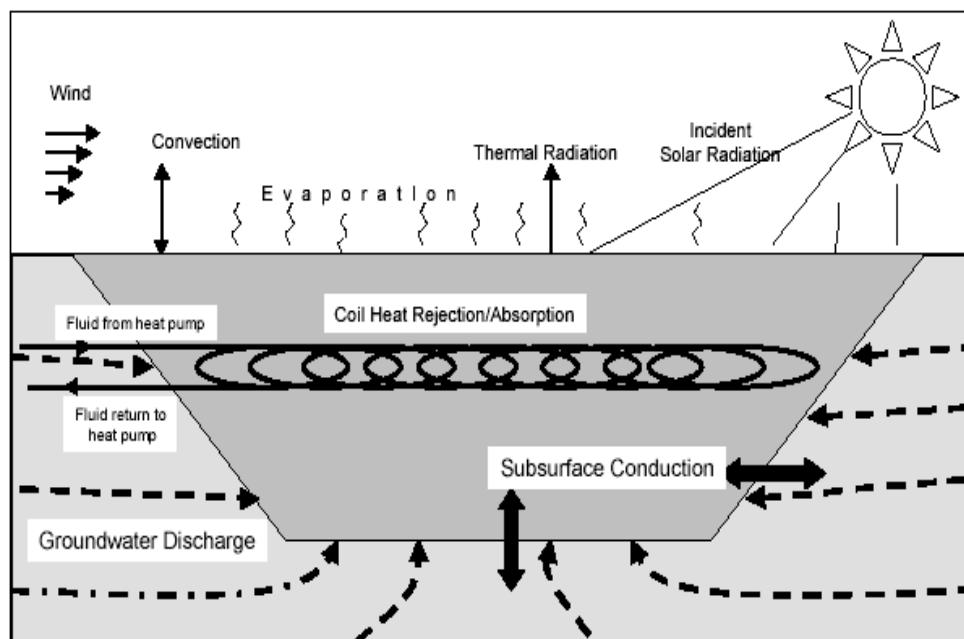


Figure 192. Heat Transfer Mechanisms in a Pond (Chiasson 1999)

Rees(2002) implemented the model developed by Chiasson(1999) for the shallow ponds. The model is based on the assumption that thermal gradients in shallow ponds are negligible, especially during times of heat rejection. Using the lumped parameter approach, an overall energy balance for the pond may be devised as

$$q_{in} - q_{out} = V \rho C_p \frac{dT}{dt}$$

Where

q_{in} = Heat transfer to the pond

q_{out} = Heat transfer from the pond

V = Pond Volume

ρ = Density of pond water

q_{in} = Heat transfer to the pond

C_p = Specific heat capacity of pond water

$\frac{dT}{dt}$ = rate of change of temperature of the pond water

Rate of change in average pond temperature is given as

$$\frac{dT}{dt} = \frac{q_{solar} + q_{thermal} + q_{convection} + q_{groundwater} + q_{evaporation} + q_{fluid}}{V \rho C_p}$$

Where

q_{solar} = Solar radiation heat gain to the pond

$q_{thermal}$ = Thermal radiation heat transfer at the pond surface.

$q_{convection}$ = Convection heat transfer at the pond surface

q_{ground} = Heat transfer to/from ground to the pond

$q_{groundwater}$ = Heat transfer due to ground water inflow/outflow

$q_{evaporation}$ = Heat transfer due to evaporation at the pond surface

q_{fluid} = Total heat transfer to/from the heat exchanging fluid flowing in all spools or coils in the pond

A heat balance is calculated at a single node that represents the pond. Heat transfer takes place by surface convection, long-wave radiation to the sky, absorption of solar energy, ground heat transfer and heat exchange with the fluid. A heat exchanger analogy is used to calculate the heat transfer between the heat transfer fluid and the pond. The differential equation defined by the heat balance is solved using a fourth order Runge-Kutta numerical integration method. The implementation along with the model equations are summarized in the figure below.

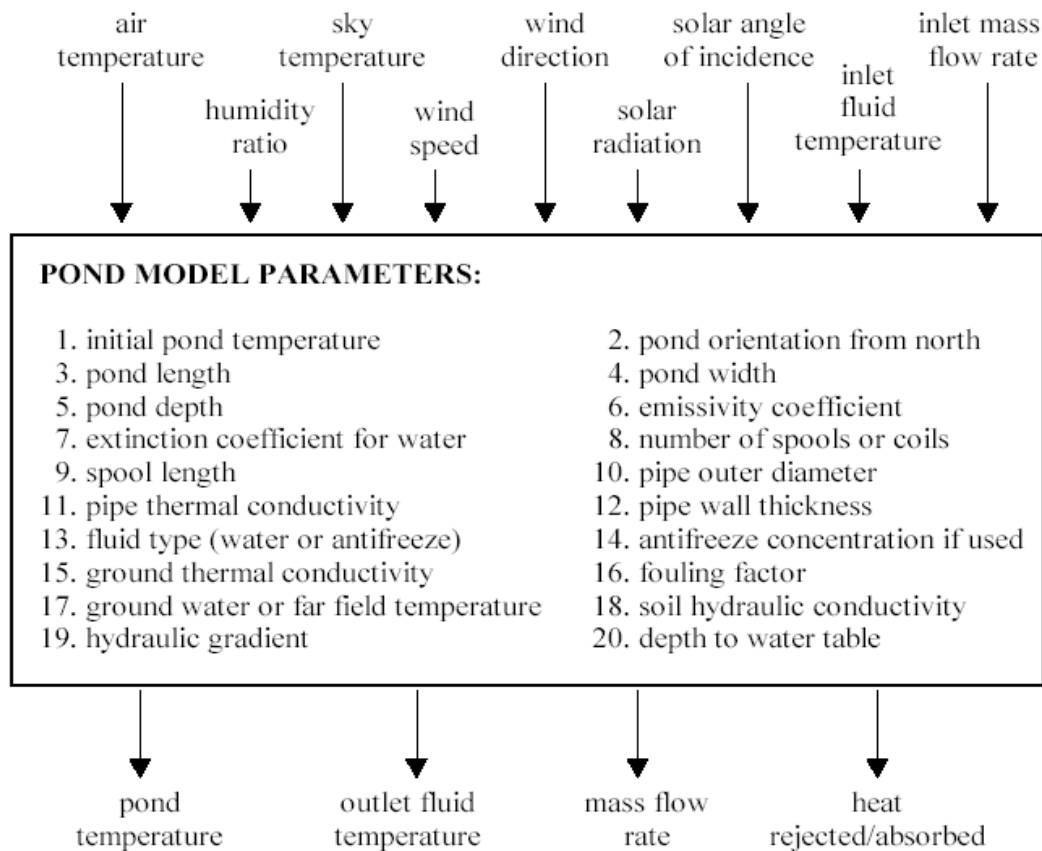


Figure 193. Pond Model Component Configuration (Chiasson 1999)

The model overall pond model consists in a number of sub-models which are enumerated below.

Solar radiation heat gain to the pond

$$\rho' = \tau_a - \tau$$

where

ρ' is the reflectance

τ is the transmittance of solar radiation by the pond surface and the subscript 'a' refers to the absorbed component.

$$\tau_a = e^{\frac{-\mu'd}{\cos\theta_r}}$$

and

$$\tau = \frac{1}{2} \left(\frac{1 - r_{par}}{1 + r_{par}} + \frac{1 - r_{\perp}}{1 + r_{\perp}} \right) e^{\frac{-\mu'd}{\cos\theta_r}}$$

where

μ' is the extinction coefficient for water

d is the pond depth

r_{par} represents the parallel component of unpolarized radiation and

r_{\perp} represents the perpendicular component of unpolarized radiation which are computed by Duffie and Beckman (1991) as:

$$r_{par} = \frac{\tan^2(\theta_r - \theta)}{\tan^2(\theta_r + \theta)}$$

$$r_{\perp} = \frac{\sin^2(\theta_r - \theta)}{\sin^2(\theta_r + \theta)}$$

Finally, the amount of solar radiation absorbed by the pond (q_{solar}) is expressed as:

$$q_{solar} = I(1 - \rho') A_{pond}$$

where

I is the solar radiation flux incident on the pond surface (here, the total reflectance is approximated by the beam reflectance)

A_{pond} is the area of the pond surface.

The model also accepts solar radiation in the form of beam (I_b) and diffuse (I_d) components, in which case I is computed from:

$$I = I_b \cos \theta + I_d$$

Thermal radiation heat transfer at the pond surface

$$q_{thermal} = \sigma \alpha (T_{surface}^4 - T_{sky}^4)$$

Where

α = Thermal absorptivity

σ = Stefan Boltzman constant

Convection heat transfer at the pond surface

$$q_{convection} = h_c (T_{pond} - T_{db})$$

ASHRAE simple convection coefficient model is used to obtain the convection coefficient for external surfaces. Initializing of pond temps to mean of dry-bulb temperature T_{db} and ground temperatures is useful because repeated warm up days tend to drive the initial pond temperature toward the dry-bulb temperature. Hence, for each environment the simulation starts the pond temperature T_{pond} midway between the dry-bulb and ground temperature.

Heat transfer to/from ground to the pond

Hull et al (1984) expressed the following equation for ground heat losses from any pond. The equations are also based on the assumption that, for all practical purposes, the constant temperature sink is the ground water table (Kishore and Joshi 1984)

$$q_{ground} = U_{ground} A_{ground} (T_{groundwater} - T_{pond})$$

$$U_{ground} = 0.999 \left(\frac{k_{ground}}{d_{groundwater} - d_{pond}} \right) + 1.37 \left(\frac{k_{ground} P_{pond}}{A_{pond}} \right)$$

Where

k_{ground} = Thermal conductivity of the ground

$d_{groundwater}$ = depth of water table

d_{pond} = Pond depth

P_{pond} = Pond perimeter

Heat transfer due to evaporation at the pond surface

Evaporation is calculated assuming a fixed Lewis number unlike the Chaisson model.

$$q_{evaporation} = \left(\frac{Pr}{Sc} \right)^{2/3} \left(\frac{h_c}{Cp_{air}} \right) (HR_f - HR_a) (q_l)$$

Where

Pr = Prandtl number for air

Sc = Schmidt number for air

Hc = convection coefficient

HRf = humidity ratio at pond surface/film temperature

HRa = humidity ratio of air

ql = Latent heat of air

Air properties are obtained by applying the existing the psychometric functions of EnergyPlus.

Total heat transfer to/from the heat exchanging fluid flowing in all spools or coils in the pond

$$q_{fluid} = UA_{pipe} (T_{fluid} - T_{pond}) N_{circuit}$$

Where

UA_{pipe} = overall heat transfer coefficient expressed in terms of inside pipe area

$N_{circuit}$ = number of spools installed in the pond

The fluid temperature Tfluid is computed as the average fluid temperatures of the inlet and the outlet node at a given temperature.

$$UA_{pipe} = \frac{2\pi r_i \cdot L_{spool}}{\sum r_t}$$

where

r_i = inner pipe radius

L_{spool} = length of one spool

$\sum r_t$ = composite thermal resistance defined as

$\sum r_t$ = Resistance due to fluid flow in the pipe + external pipe thermal resistance + fouling factor

References

Chiasson, A.D. 1999. *Advances in modeling of groundsource heat pump systems*. Master's thesis, Oklahoma State University, Stillwater, OK

Duffie, J.A., and W.A. Beckman. 1991. *Solar engineering of thermal processes*, 2d ed. New York: John Wiley & Sons.

Hull, J.R., K.V. Liu, W.T. Sha, J. Kamal, and C.E. Nielsen. 1984. Dependence of ground heat losses upon solar pond size and perimeter insulation—Calculated and experimental results. *Solar Energy* 33(1): 25-33

Kishore, V.V.N., and V. Joshi. 1984. A practical collector efficiency equation for nonconvecting solar ponds. *Solar Energy* 33(5): 391-395.

Rees, S.J., J.D. Spitler and X. Xiao, X 2002. [Transient Analysis of Snow-melting System Performance](#). *ASHRAE Transactions*. 108(2):406-423.

Plant Loop Surface-Ground-To-Water Heat Exchanger

This model (Object: GroundHeatExchanger:Surface) is based on the QTF formulation of heat transfer through building elements with embedded heat sources/sinks. The model uses a heat exchanger analogy to relate the inlet fluid temperature to the net heat transfer rate and consequently outlet temperature. The model is entirely passive, i.e. it does not set any flow rates or incorporate any controls. In order to deal with the non-linear boundary conditions at the top surface due to the presence of ice/snow fluxes have to be calculated by the QTF model and temperature calculated from the surface heat balance. This requires some iteration. Note, top surface variables correspond to 'outside' variables in standard CTF/QTF definition. Bottom surface variables correspond to 'inside' variables.

For given current surface temperatures the terms of the QTF equations can be grouped into constant terms, and those depending on the current source flux. The surface heat balance may be given by the following equation (Strand, 1997)

$$QS = \sum_{m=1}^M X_{k,m} T_{i,t-m+1} - \sum_{m=1}^M Y_{k,m} T_{o,t-m+1} + \sum_{m=1}^k F_m q_{i,t-m} + \sum_{m=1}^M W_m q''_{source,t-m+1}$$

$$T_s = \sum_{m=1}^M X_{k,m} T_{i,t-m+1} - \sum_{m=1}^M Y_{k,m} T_{o,t-m+1} + \sum_{m=1}^k F_m q_{i,t-m} + \sum_{m=1}^M W_m q''_{source,t-m+1}$$

Where

T_s = temperature of the node where the heat source or sink is present

QS = Surface heat balance

q'' = Heat flux

T = Temperature

i = inside of the building element

o = outside of the building element

t = current time step

X Y F = Conduction transfer functions

The surface balance equation includes terms for incident solar energy, radiation heat transfer from internal sources, linearized radiation between surfaces using the mean radiation temperature concept and convection to the surrounding air.

The heat balance on the water loop is given by

$$q = \dot{m} c_p (T_{wi} - T_{wo})$$

Where

q = heat transferred between the water loop and the building elements.

\dot{m} = mass flow rate of water

T_{wi} = Inlet water temperature

T_{wo} = Outlet water temperature

From the second law of thermodynamics the maximum amount of heat transfer is

$$q_{\max} = (\dot{m} c_p)_{\text{water}} (T_{wi} - T_s)$$

Where

T_s = temperature at the source location

The effectiveness of the heat exchanger is given by

$$\varepsilon = 1 - e^{-NTU}$$

Where NTU is defined by

$$NTU = \frac{UA}{(\dot{m} c_p)_{\text{water}}}$$

$$UA = h\pi DL$$

h is the convection coefficient, D is the interior tube diameter and L is the total length of the tube.

The Colburn equation is used to define the Nusselt number Nu

$$Nu = \frac{hD}{k} = 0.023 Re_D^{4/5} Pr^{1/3}$$

Pr is the Prandtl number, Re is the Reynolds number and k is the thermal conductivity of the fluid

$$\text{Re}_D = \frac{4\dot{m}}{\pi\mu D}$$

with μ being the absolute viscosity of water

References

Strand, R.K. and C.O. Pedersen, 1997. Implementation of a Radiant Heating and Cooling Model Into an Integrated Building Energy Analysis Program, ASHRAE Transactions v. 103, n. 1, pp 949-958, 1997.

Plant Loop Water-To-Water Free Cooling Hydronic Heat Exchanger

This component (Object: HeatExchanger:Hydronic) is a simple hydronic heat exchanger that can be used to couple plant side and condenser side hydronic loops. The purpose is primarily to enable implementation of free cooling. In other words, it would include coupling the zone plant directly to the heat rejection equipment. For example, coupling a radiant hydronic system to a cooling tower and bypassing the heat pump or chiller. Another component (chiller or heat pump) on the loop is switched on or off by this component. When the other component is operating the loops are 'uncoupled' and no heat exchange takes place. When the control conditions are satisfied this heat exchanger switches the other component off and heat exchange between the loops is calculated. This is to emulate a bypass arrangement. The heat exchanger can be operated in 'ideal' mode, that is the condition where maximum possible heat exchanger occurs. If the loop fluids, and the flow rates are the same, complete coupling occurs and the condenser fluid flows through the plant loop.

From the second law of thermodynamics the maximum amount of heat transfer is

$$q_{\max} = \varepsilon(\dot{m}c_p)_{\text{Minimum}}(T_p - T_c)$$

T_p = Plant inlet temperature

T_c = Condenser inlet temperature

ε = effectiveness

$$\varepsilon = (1 - e^{-NTU^{0.22}/Cratio})((-Cratio * NTU^{0.78}) - 1)$$

$$Cratio = \frac{\text{Min}(C1, C2)}{\text{Max}(C1, C2)}$$

C1 is the plant capacity rate

C2 is the condenser capacity rate

$$NTU = \frac{UA}{\text{Min}(C1, C2)}$$

The model is an NTU-effectiveness heat exchanger model. Effectiveness is calculated from the user supplied UA value. If 'ideal' mode has been set, effectiveness is set to 1.0.

References

No specific references.

Plant Loop Water Side Economizer

The input object HeatExchanger:WatersideEconomizer provides a model for water side economizer that is used in a plant loop to reduce the load or to completely turn off the primary equipment. The economizer operation results in reduced chilled water return temperature thereby reducing the load. In situations where the loop demand is low and conditions favorable the primary equipment can be turned off completely and the whole load met by the economizer alone. Water side economizer consists of a heat exchanger and related controller. The purpose of the controller is to determine if the conditions are favorable to turn on the heat exchanger. In a typical economizer application the decision is made based on the temperature difference between the inlet temperatures on both sides of the heat exchanger.

In cooling applications, heat exchanger is turned on when the plant side inlet temperature (chilled water return temperature) is more than the condenser side inlet temperature (cooling tower return temperature). In heating applications, the temperature difference is calculated as difference between the condenser side inlet temperature and plant side inlet temperature of heat exchanger. The user specifies the threshold for the temperature difference.

Water side economizers are configured in two different ways viz. Integrated and Non-integrated economizers. A brief description of both configurations is given in following paragraphs.

In an integrated water side economizer, heat exchanger is placed in series with the chillers. A typical integrated water side economizer installation is shown in Figure 194. In such applications, the heat exchanger pre-cools the chiller supply temperature. The net effect of the economizer is to meet the chiller load fully or partly. The heat exchanger is turned ON/OFF by monitoring the condenser return water temperature. A schematic of the integrated water side economizer in EnergyPlus Plant Loop convention is shown in Figure 195. The plant loop in this case can be either a primary-secondary pumping system or a primary only pumping system.

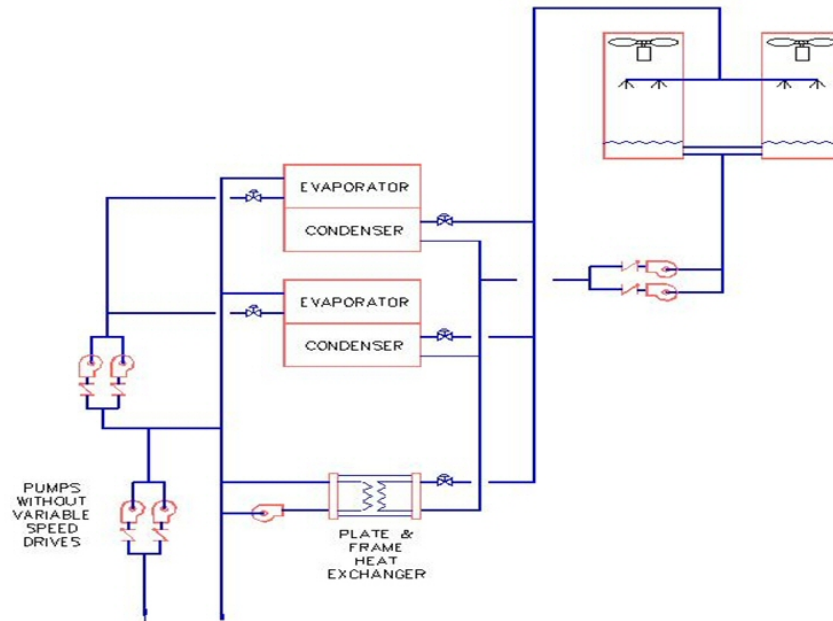


Figure 194. Typical Integrated Water side Economizer Installation

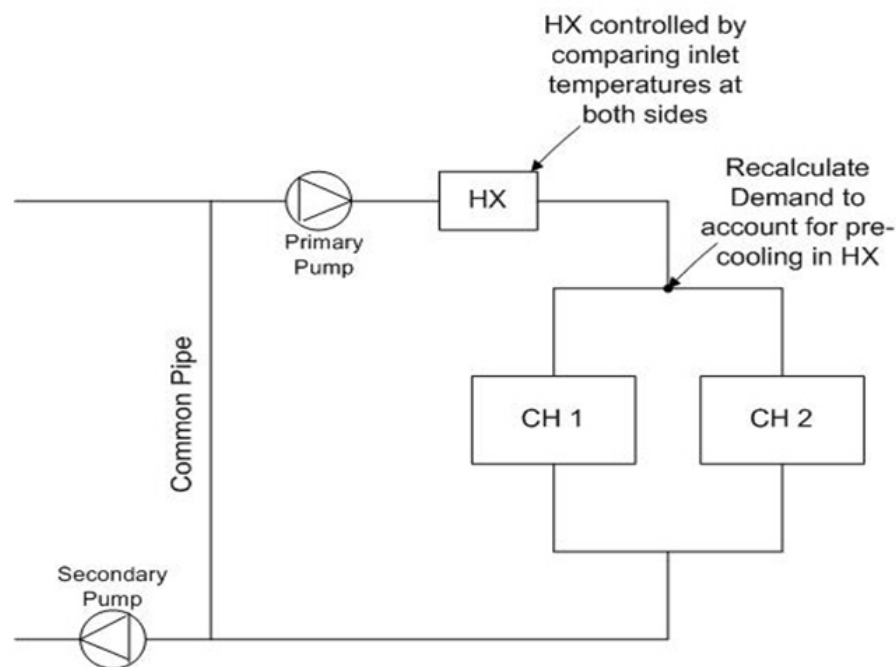


Figure 195. Schematic of Integrated Water side Economizer

In a non-integrated economizer, the heat exchanger is placed in parallel with the chillers and is supplied by the same pump system as the chillers. The immediate effect of the heat exchanger in this case is not to pre-cool the chiller inlet water temperature. In this configuration, the purpose of heat exchanger is to reduce the load on the chillers by reducing the chiller run-time. A typical non-integrated water side economizer installation is shown in Figure 196. The corresponding EnergyPlus Loop schematic is shown in Figure 197. In EnergyPlus achieving complete unloading of the chillers when the heat exchangers are ON

could be challenging with a simple control algorithm used in the integrated economizer case. An alternative method implemented is to allow the loop to subcool with both heat exchanger and chillers running concurrently. This control scheme allows the chillers to be gradually unloaded as the heat exchanger picks up the loop load, and it allows the heat exchanger to continue operating after the loop load has been met to subcool the loop if desired. Under these conditions, if the return water from the coil is less than the setpoint and favorable conditions exist at heat exchanger, the chillers would remain OFF and the load would be completely met by the heat exchanger.

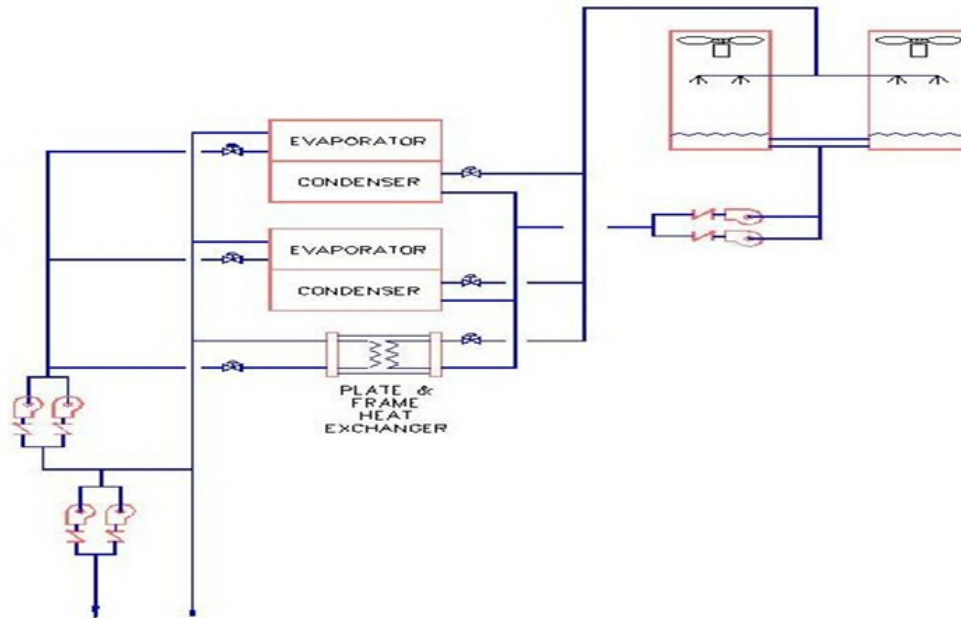


Figure 196. Typical Non-Integrated Water side Economizer Installation

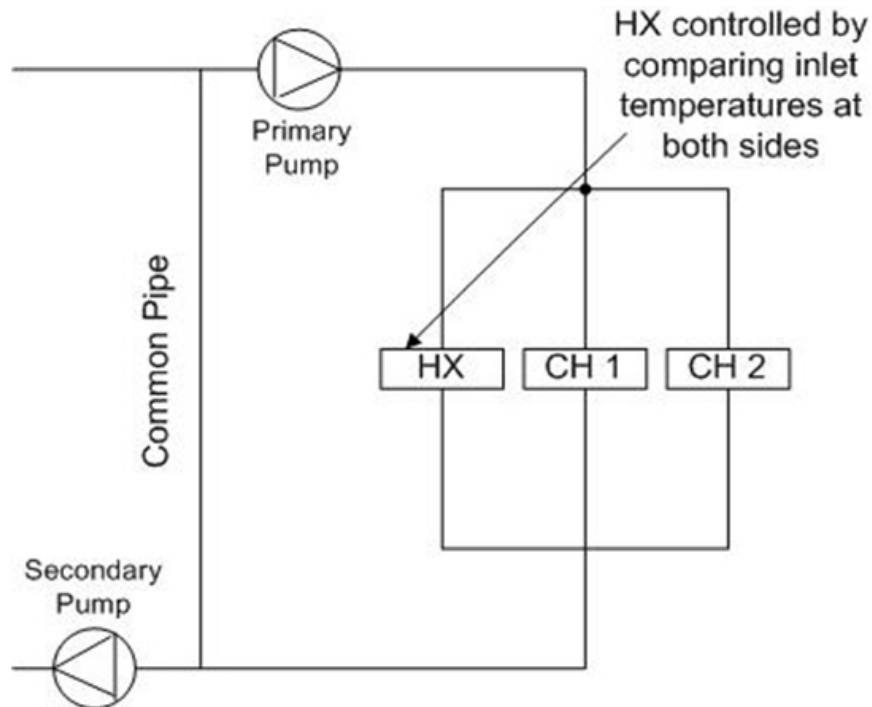


Figure 197. Schematic of Non-Integrated Water side Economizer

EnergyPlus economizer specification consists of a heat exchanger object and a controller object. Heat exchanger simulation is based on the effectiveness-NTU model. Effectiveness of heat exchanger is calculated based on the mode of heat exchange occurring, which is user specified. Heat exchanger mode can be specified as a Plate-frame, Counter flow, Parallel flow or an Ideal heat exchanger. Correlations corresponding to different types of heat exchanger are used for counter flow and parallel flow, while a plate frame heat exchanger is approximated as a cross flow heat exchanger with both fluid streams unmixed. The effectiveness of an ideal heat exchanger is assumed to be 1.0.

For counter flow heat exchanger the effectiveness is calculated from UA and NTU as:

$$\varepsilon = \frac{1 - \exp[-NTU(1-C)]}{1 - C \cdot \exp[-NTU(1-C)]}$$

Effectiveness for a parallel flow heat exchanger is calculated as:

$$\varepsilon = \frac{1 - \exp[-NTU(1-C)]}{1 + C}$$

Plate Frame heat exchanger is approximated as a cross flow heat exchanger with both streams unmixed and effectiveness is calculated as:

$$\varepsilon = 1 - \exp\left[\frac{NTU^{0.22} \left\{ \exp(-C \cdot NTU^{0.78}) - 1 \right\}}{C}\right]$$

Where,

ε = Effectiveness [-]
 NTU = Number of Transfer Units [-]
 C = Ratio of Capacities, C_{min}/C_{max} [-]

The heat exchanger is always passive from a load perspective. From a flow request perspective it acts as a passive component when configured as an integrated economizer and active or passive when configured as a non-integrated economizer. An active heat exchanger in non-integrated configuration, will allow a variable speed pump to give enough flow so that heat exchanger could have flow through it. When configured as passive in a non-integrated configuration, variable speed pump will not give flow to heat exchanger.

References

Incropera, F.P. and D.P. DeWitt. 1981. Fundamentals of Heat Transfer. New York: John Wiley & Sons.
 Presentation by Taylor Engineering.

Air System Humidifiers

Overview

Air system humidifiers are components that add moisture to the supply air stream. They fall into 2 broad categories: spray type humidifiers which act like direct evaporative coolers, cooling the supply air as well as humidifying it; and dry steam humidifiers, which humidify the supply air stream while causing almost no change to the supply air stream temperature. The EnergyPlus electric steam humidifier uses electrical energy to convert ordinary tap water to steam which it then injects into the supply air stream by means of a blower fan. The actual unit might be an electrode-type humidifier or a resistance-type humidifier.

Electric Steam Humidifier

The electric steam humidifier model (object name: Humidifier:Steam:Electric) is based on moisture and enthalpy balance equations plus standard psychrometric relationships. The approach is similar to that taken in the ASHRAE HVAC 2 Toolkit, page 4-112 (ASHRAE 1993). EnergyPlus contains its own module of psychrometric routines; the psychrometric theory and relations are given in the 2001 edition of ASHRAE Fundamentals, Chapter 6 (ASHRAE 2001). The model contains both an ideal controller and the component. The control model assumes that there is a minimum humidity setpoint on the component air outlet node. This setpoint is established by a setpoint manager described elsewhere.

Model

The component model is a forward model: its inputs are its inlet conditions; its outputs are its outlet conditions and its energy consumption. The inputs are the temperature, humidity ratio, and mass flow rate of the inlet air stream, which are known; and the water addition rate (kg/s) which is determined by the controller.

Controller

The controller first decides whether the humidifier is on or off. For the humidifier to be on:

- the humidifier schedule value must be nonzero;
- the inlet air mass flow must be greater than zero;
- the inlet air humidity ratio must be less than the minimum humidity ratio setpoint.

If the humidifier is off, the water addition rate is set to zero. If the humidifier is on, the water addition rate needed to meet the humidity setpoint is calculated.

$$\dot{m}_a \cdot w_{in} + \dot{m}_{w,add,needed} = \dot{m}_a \cdot w_{set} \quad (460)$$

where

\dot{m}_a = the air mass flow rate [kg/s]

w_{in} = the inlet air humidity ratio [kg/kg]

$\dot{m}_{w,add,needed}$ = water addition rate needed to meet the setpoint [kg/s]

w_{set} = the humidity ratio setpoint [kg/kg]

Equation (460) is the moisture balance equation for the component. It is solved for $\dot{m}_{w,add,needed}$ (the other variables are known) which is passed to the humidifier component model as its desired inlet water addition rate.

Component

The inputs to the component model are the air inlet conditions and mass flow rate and the water addition rate set by the controller. The outputs are the air outlet conditions. First the desired water addition rate is checked against component capacity.

$$\dot{m}_{w,add,needed,max} = \text{Min}(\dot{m}_{w,add}, \text{Cap}_{nom})$$

where

Cap_{nom} = the humidifier nominal capacity [kg/s], a user input.

If $\dot{m}_{w,add,needed,max}$ is zero, the outlet conditions are set to the inlet conditions and the water addition rate is set to zero. If the humidifier is scheduled on the component power consumption is set to the standby power consumption: $W_{hum} = W_{stby}$. Otherwise $W_{hum} = 0$.

If $\dot{m}_{w,add,needed,max} > 0$, then the moisture and enthalpy balance equations

$$\dot{m}_a \cdot w_{in} + \dot{m}_w = \dot{m}_a \cdot w_{out}$$

$$\dot{m}_a \cdot h_{in} + \dot{m}_w \cdot h_w = \dot{m}_a \cdot h_{out}$$

with \dot{m}_w set equal to $\dot{m}_{w,add,needed,max}$ are solved for w_{out} and h_{out} . Here

\dot{m}_a = the air mass flow rate [kg/s]

w_{in} = the inlet air humidity ratio [kg/kg]

\dot{m}_w = the inlet water addition rate [kg/s]

w_{out} = the outlet air humidity ratio [kg/kg]

h_{in} = the inlet air specific enthalpy [J/kg]

h_w = the steam specific enthalpy = 2676125. [J/kg] at 100 °C

h_{out} = the outlet air specific enthalpy [J/kg]

The outlet temperature is obtained from

$$T_{out} = \text{PsyHFnTdbW}(h_{out}, w_{out})$$

where

T_{out} = outlet air temperature [°C],

$PsyHFnTdbW(h_{out}, w_{out})$ is an EnergyPlus psychrometric function.

The humidity ratio at saturation at the outlet temperature is

$$w_{out,sat} = PsyWFnTdbRhPb(T_{out}, 1.0, P_{atmo})$$

where

P_{atmo} = the barometric pressure [Pa],

1.0 is the relative humidity at saturation,

$PsyWFnTdbRhPb$ is an EnergyPlus psychrometric function.

IF $w_{out} \leq w_{out,sat}$ then the outlet condition is below the saturation curve and the desired moisture addition rate can be met. $\dot{m}_{w,add}$ is set to $\dot{m}_{w,add,needed,max}$ and the calculation of outlet conditions is done. But if $w_{out} > w_{out,sat}$ then it is assumed that this condition will be detected and the steam addition rate throttled back to bring the outlet conditions back to the saturation condition. We need to find the point where the line drawn between state 1 (inlet) and state 2 (our desired outlet) crosses the saturation curve. This will be the new outlet condition. Rather than iterate to obtain this point, we find it approximately by solving for the point where 2 lines cross: the first drawn from state 1 to state 2, the second from $T_1, w_{1,sat}$ to $T_2, w_{2,sat}$; where

T_1 is the inlet temperature [°C],

$w_{1,sat}$ is the humidity ratio at saturation at temperature T_1 [kg/kg],

T_2 is the desired outlet temperature [°C],

$w_{2,sat}$ is the humidity ratio at saturation at temperature T_2 [kg/kg].

The 2 lines are given by the equations:

$$w = w_1 + ((w_2 - w_1)/(T_2 - T_1)) \cdot (T - T_1)$$

$$w = w_{1,sat} + ((w_{2,sat} - w_{1,sat})/(T_2 - T_1)) \cdot (T - T_1)$$

Solving for the point (state 3) where the lines cross:

$$w_3 = w_1 + ((w_2 - w_1) \cdot (w_{1,sat} - w_1)) / (w_2 - w_{2,sat} + w_{1,sat} - w_1)$$

$$T_3 = T_1 + (w_3 - w_1) \cdot ((T_2 - T_1)/(w_2 - w_1))$$

This point isn't quite on the saturation curve since we made a linear approximation of the curve, but the temperature should be very close to the correct outlet temperature. We will use this temperature as the outlet temperature and move to the saturation curve for the outlet humidity and enthalpy. Thus we set $T_{out} = T_3$ and

$$w_{out} = PsyWFnTdbRhPb(T_{out}, 1.0, P_{atmo})$$

$$h_{out} = PsyHFnTdbW(T_{out}, w_{out})$$

where $PsyHFnTdbW$ is an EnergyPlus psychrometric function. The water addition rate is set to

$$\dot{m}_{w,add} = \dot{m}_a \cdot (w_{out} - w_{in})$$

We now have the outlet conditions and the adjusted steam addition rate for the case where the desired outlet humidity results in an outlet state above the saturation curve.

Finally, the electricity consumption is given by

$$W_{hum} = (\dot{m}_{w,add} / Cap_{nom}) \cdot W_{nom} + W_{fan} + W_{stby}$$

where

W_{fan} = nominal fan power [W], a user input,

W_{stby} = standby power [W], a user input.

and the water consumption rate is

$$\dot{V}_{cons} = \dot{m}_{w,add} / \rho_w$$

where

\dot{V}_{cons} = the water consumption rate [m³],

ρ_w = water density (998.2 kg/m³).

References

- ASHRAE. 1993. HVAC 2 Toolkit: A Toolkit for Secondary HVAC System Energy Calculations. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- ASHRAE. 2001. 2001 ASHRAE Handbook Fundamentals. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Zone Internal Gains

Sources and Types of Gains

Internal heat gains from lights, people, and equipment of various types are often significant elements in the zone thermal balance. EnergyPlus allows the user to specify heat gains for several equipment types including people, lights, gas/electric equipment, and several other types. The total heat gain is comprised of convective, radiant and latent gains in various proportions from these sources. Convective gains are instantaneous additions of heat to the zone air. Radiant gains are distributed on the surfaces of the zone, where they are first absorbed and then released back into the room (with some fraction conducted through the surface) according to the surface heat balances. {See Surface Heat Balance Manager / Processes in this document}. Latent gains must be handled by ventilation or air conditioning equipment. Recommended heat gains are given by ASHRAE [1]. These recommendations include the sensible (convective plus radiative) and latent proportions. Sensible gains from

equipment are primarily radiant. The user can specify the heat gains and proportions for any type of equipment. Determining the gains from lights, people and baseboard heat are slightly more complicated.

Heat Gain from Lights

The input object Lights provides a model for internal gains from lights. Radiant gains from lights must be handled differently from other radiant gains for reasons described here (long wavelength description). The total radiant gains from lights must be divided into visible and thermal portions. For example, the total electric input to typical incandescent lights is converted to 10% visible radiation, 80% thermal radiation, and 10% convective gain. In contrast, the electric input to typical fluorescent lights is converted to 20% visible radiation, 20% thermal radiation, and 60% convective gain [2]. These percentage splits are under user control with the Lights input object.

Heat Gain from People

The input object People provides a model for internal gains from occupants. Heat is generated in the human body by oxidation at a rate called the metabolic rate (see Thermal Comfort discussion for more details). This heat is dissipated from the body surface and respiratory tract by a combination of radiation, convection, and evaporation. The relative proportions of sensible (radiation plus convection) and latent (evaporation) heat from people is a complex function of the metabolic rate and the environmental conditions. EnergyPlus uses a polynomial function to divide the total metabolic heat gain into sensible and latent portions. That function is based on a fit to data [3] at average adjusted metabolic rates of 350, 400, 450, 500, 750, 850, 1000 and 1450 Btu/h each at temperatures of 70, 75, 78, 80, 82 degrees Fahrenheit. Sensible gains of 0 at 96 F and sensible gains equal to the metabolic rate at 30 F were assumed in order to give reasonable values beyond the reported temperature range.

Average adjusted metabolic rate [3] is the metabolic rate to be applied to a mixed group of people with a typical percent composition based on the following factors:

Metabolic rate, adult female = Metabolic rate, adult male X 0.85

Metabolic rate, children = Metabolic rate, adult male X 0.75

The original data was in I-P (Inch-Pound) units, but the following correlation is in SI (Systems-International) units.

$$S = 6.461927 + .946892 \cdot M + .0000255737 \cdot M^2 \\ + 7.139322 \cdot T - .0627909 \cdot T \cdot M + .0000589172 \cdot T \cdot M^2 \\ - .198550 \cdot T^2 + .000940018 \cdot T^2 \cdot M - .00000149532 \cdot T^2 \cdot M^2$$

where

M=Metabolic Rate (W)

T=Air Temperature (C)

S=Sensible Gain (W)

Latent Gain is simply the total gain (metabolic rate) – sensible gain:

$$LatentGain = MetabolicRate - SensibleGain$$

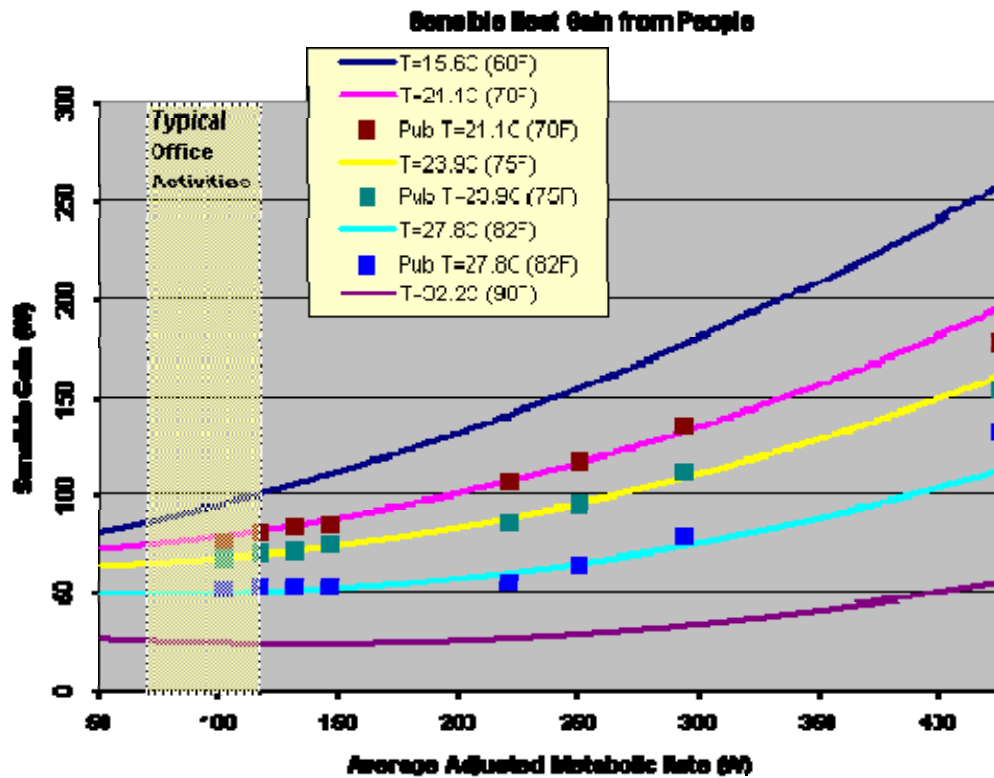


Figure 198. Sensible Heat Gain from People Correlation

The function for sensible gain calculation is compared to the original data points in the following figure. The radiant fraction of the sensible gain is a user input on the People object.

Heat Gain from Baseboard Heat

The input object ZoneBaseboard:OutdoorTemperatureControlled provides a model for an outdoor temperature controlled baseboard heater that adds energy to the zone according a control profile as shown in the following figure. At $T_A = T_2$, the baseboard heat gain is Q_2 . For $T_A > T_2$, there is no heat gain. For $T_A < T_1$, a maximum amount of energy, Q_1 , is added to the zone. There is proportional control between those two temperatures:

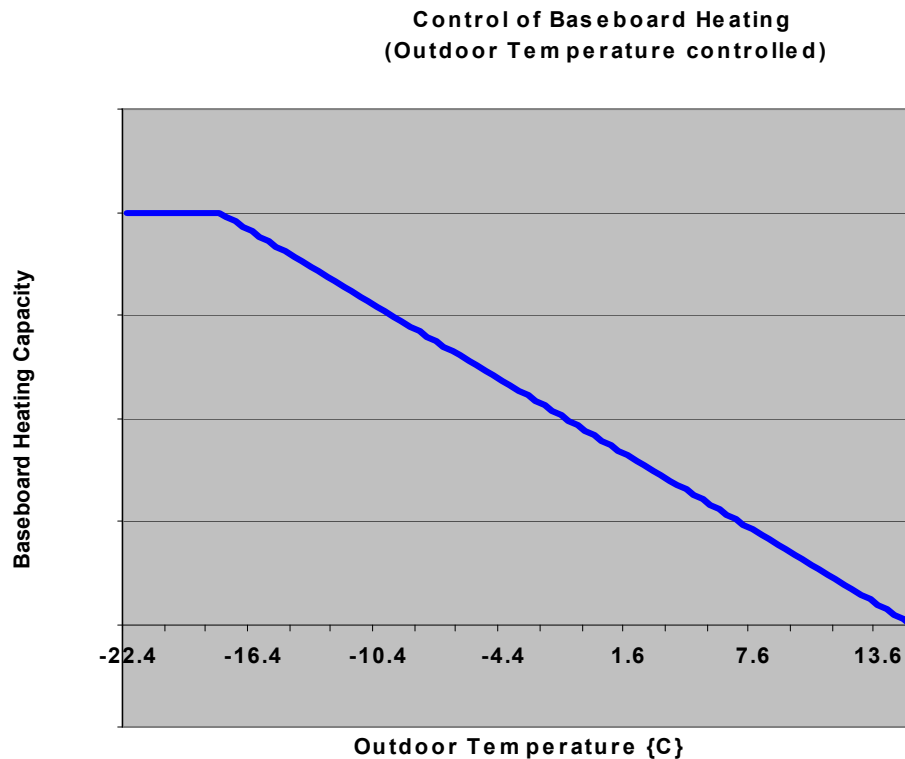


Figure 199. Control of Outdoor Temperature Controlled Baseboard Heat

$$Q = Q2 - \frac{(Q2 - Q1) \cdot (T2 - TA)}{(T2 - T1)}$$

Distribution of Radiant Gains

It is useful to consider the distribution of short wavelength (including visible) radiant energy separate from long wavelength (thermal) radiant energy because many materials have different optical properties at different wavelengths. An extreme example is glass that is opaque to the long wavelengths and transparent to the short. Properties of materials vary across the entire spectrum of wavelengths. In EnergyPlus, all radiant interactions are represented in terms of only two wavelengths: “short” and “long”. Short wavelength refers to the distribution given by a ~6000K black body source such as the sun. Long wavelengths refer to radiation from ~300K sources such as walls or people. There is negligible overlap between these two distributions. Some sources, such as lights, must be considered as emitting both long and short wavelength radiation in proportions that approximate their actual effects on room surfaces.

Long wavelength radiation from all internal sources, such as people, lights and equipment, is combined and then distributed over surfaces. (see Internal Long-Wave Radiation Exchange).

Some fraction of the beam solar radiation transmitted into the zone is directly absorbed by the interior surfaces according to the solar distribution algorithm (see Solar Distribution) selected by the user. The beam radiation not directly absorbed, plus the diffuse sky and ground-reflected radiation, plus the short wavelength radiation from lights are combined and distributed over the surfaces of the zone according to:

$$QSI_i = QS_n \cdot \alpha_i / \sum_{i=1}^{NS} S_i \cdot (1 - \rho_i)$$

If all surfaces in the room are opaque, the radiation is distributed in proportion to the area*absorptance product of each surface. For surfaces which are transparent,

$$\rho_i = 1 - \alpha_i - \tau_i$$

That fraction of radiation represented by τ_i is lost from the zone.

The transmittance and absorptance of transparent surfaces (windows or glass doors) are calculated as in section Window Calculation Module based on the optical properties of the window material layers. The total absorptance of the window is computed for the interior shading device, the inside surface, and the outside surface for diffuse solar radiation incident from outside the zone. Those absorptances are used for short wavelength radiation incident from inside the zone. In most cases, this should not cause significant error. When movable insulation covers the window, the radiation that would have been transmitted is absorbed at the outer surface of the window (thermally equal to the inside surface of the insulation).

References

ASHRAE. 2001. Handbook of Fundamentals, pp 29.8-29.13, Atlanta: ASHRAE.

Carrier Air Conditioning Company. 1965a. Handbook of Air Conditioning System Design, pp 1-99 to 1-100. New York: McGraw Hill.

Carrier Air Conditioning Company. 1965b. Handbook of Air Conditioning System Design, pp 1-100, Table 48. New York: McGraw Hill.

Pipes

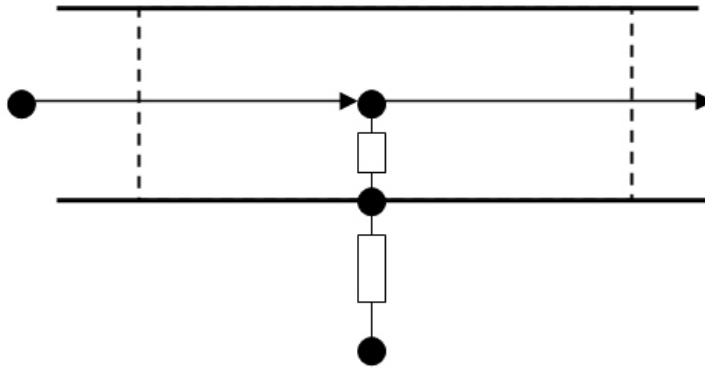
Heat Transfer Pipes (Objects: Pipe:Indoor & Pipe:Outdoor)

Heat Loss and Time Delay in Pipes

The effects of heat loss and time delay in plant loop pipes exposed to air (Pipe:Indoor and Pipe:Outdoor) can be modeled explicitly in EnergyPlus. Users can select the environment with which the pipe transfers heat. Currently users have three options: 'OutdoorAir', 'Zone' and 'Schedule'. Simulation for each of the environments is similar except the way in which the heat transfer between the pipe outer wall and the surrounding environment is calculated. When using the 'OutdoorAir' option, the current outdoor dry-bulb temperature and wind velocity from the weather file (or design day input) are used. When the environment is specified as 'Zone', the mean air temperature and room air velocity of the corresponding zone are used.

In the case of a pipe in a zone, the heat loss or gain is accounted for in the pipe heat transfer calculation and is also included in the zone air heat balance calculations. When the environment is specified as 'Schedule', the user specifies a temperature and velocity schedule which will be used to calculate the heat transfer.

Pipe heat transfer in EnergyPlus is simulated by discretizing the pipe length into a number of nodes (20) and is an implementation of the model by Hanby et al. (2002). A control volume drawn around a node in the pipe is shown in Figure 200. Three nodes are defined at each discrete section of the pipe and represent the fluid, pipe wall and external environment. The fluid and pipe have defined thermal capacitance (mass). There is one-dimensional flow from each upstream node.

Figure 200. Control Volume drawn around node i

The model is formulated from the heat balances on the fluid and wall nodes.

$$M_{f,i} C_{p,f} \frac{dT_{f,i}}{dt} = \dot{m} C_{p,f} (T_{f,i-1} - T_{f,i}) - h_f A_i (T_{f,i} - T_{w,i}) \quad (461)$$

$$M_{w,i} C_{p,w} \frac{dT_{w,i}}{dt} = h_f A_i (T_{f,i} - T_{w,i}) - h_e A_i (T_{w,i} - T_e) \quad (462)$$

Where, subscript w , f and e denote the values for pipe wall, fluid and environment respectively. The current node is represented by a subscript of i , while the previous node is represented by $i-1$.

M = Mass

C_p = Specific Heat

\dot{m} = Mass flow rate of fluid in pipe

T = Temperature

A = Heat Transfer Area

h = Film convective resistance

t = time

The exterior film convective resistance is calculated based on either wind speed, room air velocity, or a scheduled value based on the type of pipe heat transfer object. However, when the velocity gets too low, natural convection must be modeled. This is handled within the program by having a lower limit on the Nusselt number. For natural convection from a horizontal cylinder, a constant Nusselt number is assumed based on information from Spang (referenced below). This Nusselt number is 0.36. The Nusselt number used in calculating the exterior convection coefficient (Incropera and Dewitt 1996) is the maximum of the Nusselt number from the forced convection coefficient correlation and this natural convection Nusselt number (0.36).

Approximating the derivatives using backward differencing enables these equations to be represented as simultaneous algebraic equations. For the fluid, at time step n , the heat balance is:

$$\frac{M_{f,i} C_{p,f}}{\Delta t} (T_{f,i}^n - T_{f,i}^{n-1}) = \dot{m} C_{p,f} (T_{f,i-1}^n - T_{f,i}^n) - h_f A_i (T_{f,i}^n - T_{w,i}^n) \quad (463)$$

Rearranging gives,

$$T_{f,i}^n \left[M_{f,i} C_{P,f} + \dot{m} C_{P,f} \Delta t + h_f A_i \Delta t \right] = \dot{m} C_{P,f} \Delta t T_{f,i-1}^n + h_f A_i \Delta t T_{w,i}^n + M_{f,i} C_{P,f} T_{f,i}^{n-1} \quad (464)$$

or,

$$a_1 T_{f,i}^n = a_2 T_{f,i-1}^n + a_3 T_{w,i}^n + a_4 T_{f,i}^{n-1} \quad (465)$$

where,

$$a_1 = M_{f,i} C_{P,f} + \dot{m} C_{P,f} \Delta t + h_f A_i \Delta t$$

$$a_2 = \dot{m} C_{P,f} \Delta t$$

$$a_3 = h_f A_i \Delta t$$

$$a_4 = M_{f,i} C_{P,f}$$

Similarly, taking backwards differences for the conduit wall at time step n, the heat balance becomes,

$$\frac{M_{w,i} C_{P,w}}{\Delta t} (T_{w,i}^n - T_{w,i}^{n-1}) = h_f A_i (T_{f,i}^n - T_{w,i}^n) - h_e A_i (T_{w,i}^n - T_e^n) \quad (466)$$

Rearranging gives,

$$T_{w,i}^n \left[M_{w,i} C_{P,w} + h_f A_i \Delta t + h_e A_i \Delta t \right] = h_f A_i \Delta t T_{f,i}^n + h_e A_i \Delta t T_e^n + M_{w,i} C_{P,w} T_{w,i}^{n-1} \quad (467)$$

or,

$$b_1 T_{w,i}^n = b_2 T_{f,i}^n + b_3 T_e^n + b_4 T_{w,i}^{n-1} \quad (468)$$

where,

$$b_1 = M_{w,i} C_{P,w} + h_f A_i \Delta t + h_e A_i \Delta t$$

$$b_2 = h_f A_i \Delta t$$

$$b_3 = h_e A_i \Delta t$$

$$b_4 = M_{w,i} C_{P,w}$$

Substituting (468) into (465) gives an equation for the current fluid temperature:

$$a_1 T_{f,i}^n = a_2 T_{f,i-1}^n + a_3 \left(b_2 T_{f,i}^n + b_3 T_e^n + b_4 T_{w,i}^{n-1} \right) / b_1 + a_4 T_{f,i}^{n-1} \quad (469)$$

$$T_{f,i}^n = \frac{1}{(a_1 - a_3 b_2 / b_1)} \left[a_2 T_{f,i-1}^n + a_3 \left(b_3 T_e^n + b_4 T_{w,i}^{n-1} \right) / b_1 + a_4 T_{f,i}^{n-1} \right] \quad (470)$$

The conduit is simulated by solving Eq.(470) followed by Eq. (465) for each of the twenty cells in the model and incrementing the time step. The fluid temperature of the last node is taken to be the pipe outlet temperature.

References

Hanby, V.I., Wright, J.A., Fletcher, D.W and Jones, D.N.T. 2002. Modeling the Dynamic Response of Conduits. Int J. of HVACR&R, Vol.8, No.1. pp. 1-12.

Incropera, F.P. and Dewitt, D.P. 1996. Fundamentals of Heat Transfer, 4th Edition, pp. 369-370.

Spang, Bernhard. Correlations for Convective Heat Transfer. Chemical Engineers' Resource Page: <http://www.cheresources.com/convection.pdf>

Underground Pipe (Object: Pipe:Underground)

Description of Model

The buried pipe model in EnergyPlus is similar to the other pipe heat transfer objects (i.e., Pipe:Indoor and Pipe:Outdoor) except for the way in which the pipe boundary condition is developed. For a buried pipe the ground between the pipe and the surface must be modeled. For a shallow buried pipe, the GroundHeatExchanger:Surface object may be used, which uses modified conduction transfer functions to model the ground. However, beyond a certain thickness, the transfer function method fails, and EnergyPlus will respond with a fatal error due to convergence problems. Therefore, when a pipe is buried deeper than about one meter, this new buried pipe model should be used. When the pipe is buried shallower than one meter, either model may be used. Due to the finite difference nature of the Pipe:Underground model, the GroundHeatExchanger:Surface may be slightly faster and therefore more desirable.

The buried model develops a grid around the pipe. The grid was based originally on a model by Piechowski (1999), and still carries the model nomenclature. The grid extends from the ground surface down to a calculated distance below the pipe. The domain extends sideways from the symmetric center of the pipe to a calculated distance from the pipe. The grid stretches along the full length of the pipe. At each cross section, transient 2D Cartesian finite difference equations are used, updating each node except the node centered on the pipe. Axial heat transfer is not modeled in the soil. The large view of the outer Cartesian grid system is shown in Figure 201.

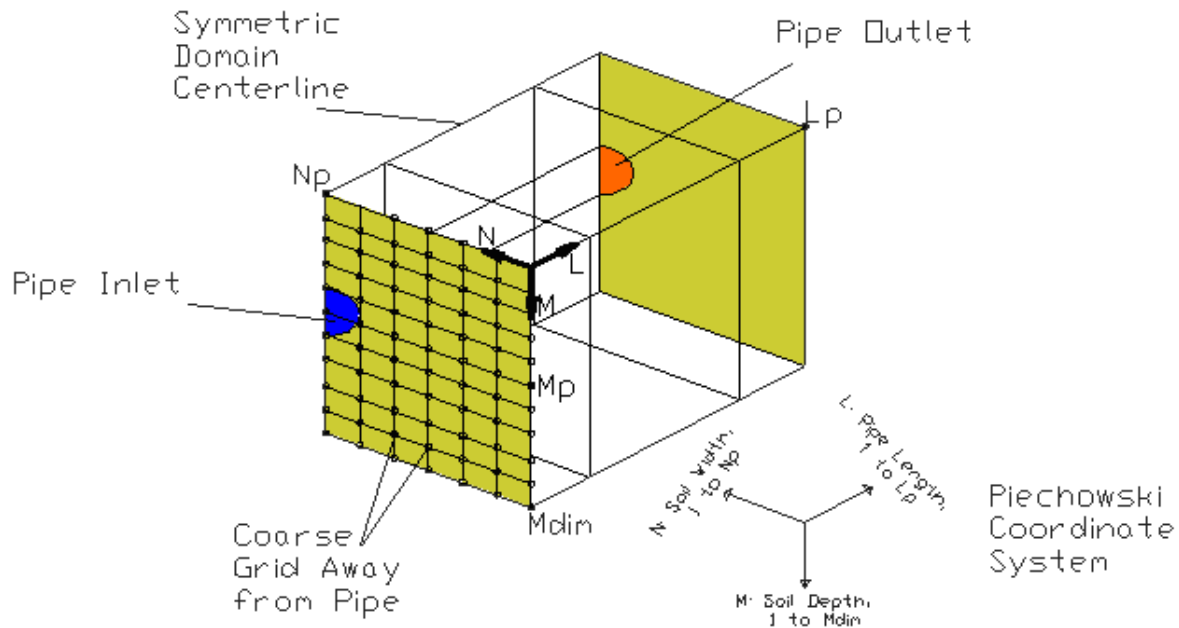


Figure 201. Pipe:Underground Outer Finite Difference Grid

When the model encounters the pipe node, the existing model for Pipe:Interior and Pipe:Exterior pipes is used. The finite difference temperatures neighboring the pipe, grid spacing and soil properties are used to create an average boundary temperature for the pipe along with a conductance value. With a boundary temperature available and a conductance value mimicking the convection coefficient, the simulation continues exactly as with the other pipe heat transfer objects. To avoid redundancy, see the Pipe:Indoor or Pipe:Outdoor objects for a detailed description of the pipe model.

Boundary Conditions

The boundary conditions for this model include a symmetric vertical boundary centered on the pipe, the ground surface, a 'far-field', and a 'deep ground'. The ground surface boundary uses current simulation outdoor dry-bulb temperature, and the user-entered convection coefficient. The 'far-field' and 'deep ground' use a correlation by Kusuda and Achenbach (1965). This correlation requires surface temperature data. Either the Site:GroundTemperature:Shallow object must be given, or the three optional ground surface temperature inputs must be given.

Currently, the model is set up to be exposed to open soil above the pipe. If the user intends on simulating this buried pipe under a foundation slab, the effects can be approximating by use of the basement/slab heat transfer preprocessor program. This program takes in general building information, and performs a simulation which generates ground temperatures. Typically these effects are used to generate boundary conditions for the floor zone, but they may also be used in generating the ground surface temperatures for this pipe model. The data from the slab program will be monthly temperatures, so the user can use these as a surface ground temperature object which provides boundary data to the Pipe:Underground model.

Geometry

The model develops the pipe depth and ground thickness from the user-entered construction information. The soil, pipe wall, and optional pipe insulation are entered as materials (with inherent thicknesses). The soil is entered as a standalone material, while the pipe insulation (if applicable) and the pipe wall should be given as a construction containing one or two

materials. With knowledge of each individual thickness, the pipe geometry is obtained. The pipe length and inside diameter are the only additional geometry inputs.

Model Assumptions

- Constant properties throughout domain
- Moisture is not directly involved with the model operation, so careful selection of soil thermal conductivity is a priority
- Negligible axial heat transfer compared to radial heat transfer
- Axisymmetric heat transfer in near pipe region
- Surface convection coefficient is constant throughout simulation (does not vary with wind speed)

References

Kusuda, T. & Achenbach, P. (1965), 'Earth Temperature and Thermal Diffusivity at Selected Stations in the United States', ASHRAE Transactions 71(1), 61–75.

Piechowski, M. (1999), 'Heat and Mass Transfer Model of a Ground Heat Exchanger: Theoretical Development', Energy research 23(7), 571–588.

Pumps

The water pump is quite simply the component that drives the flow in plant and condenser loops. How it reacts depends on several different conditions. In total, there are three different decision variables, two of which are defined by user input. These three deciding factors are whether the pump is constant or variable speed, whether the pump operation is continuous or intermittent, and whether or not there is a load on the loop. The pump is simulated first on the supply side loop after the demand side loop has determined what the demand on the loop will be. For further reference look at sections Pump Control for Plant and Condenser Loops, Plant/Condenser Supply Side, and Plant/Condenser Demand Side in the Plant Flow Resolver of the Engineering Document.

Summary of Pump Rules

- Pumps in Plant Loop can be on the supply side or demand side
- A Pump, if present, in the demand side of plant loop must be the first component of the inlet branch.
- Pumps in Condenser loop must be on supply side
- Pumps can operate as constant or variable flow.
- Pumps can run continuously or intermittently.
- Single boiler/chiller with NO bypass, use Pump:ConstantSpeed
 - Boiler/chiller should be constant flow
 - Pump should be intermittent
- Single boiler/chiller with NO bypass, Pump:VariableSpeed
 - Boiler/chiller should be variable flow, regardless of whether pump is intermittent or continuous (runs at the minimum if demand is less than minimum, this includes zero.)
- Single boiler/chiller with bypass, Pump:ConstantSpeed
 - Boiler/chiller can be constant or variable flow
 - Pump may be intermittent or continuous as long as the bypass can handle the entire pump volume when the boiler is not operating

Multiple branches add more complexity, but it is nothing more than continuity. If the pump is putting out flow then it has to have a branch to flow down whether it is a chiller or a bypass. It can be safer to add the bypass for a simulation. If the active machines require the flow the

bypass will be dry. Thermodynamically it does not make any difference if the flow goes through a machine that is OFF or it flows down the bypass. There is no pressure simulation and flow losses are not accounted for.

If the user designates a pump that is operating continuously, the pump will run regardless of whether or not there is a load. This may have the net effect of adding heat to the loop if no equipment is turned on. If the pump operates intermittently, the pump will run at its capacity if a load is sensed and will shut off if there is no load on the loop. If the pump is scheduled, the schedule modifies the Rated Volumetric Flow Rate of the pump on a time basis. The default is that the pump is ON and runs according to its other operational requirements.

Shown below is the calculation of the total efficiency of the pump and the actual pumping efficiency when the motor efficiency is accounted for either the variable or constant volume pumps.

```
! Total_Efficiency % = Rated_Volume_Flow_Rate * Rated_Pump_Head / Rated_Power_Use
TotalEffic = PumpEquip(PumpNum)%NomVolFlowRate * PumpEquip(PumpNum)%NomPumpHead &
            / PumpEquip(PumpNum)%NomPowerUse

! Calculated Pump_Efficiency % =Total_Efficiency % / Motor_Efficiency %
PumpEquip(PumpNum)%PumpEffic = TotalEffic / PumpEquip(PumpNum)%MotorEffic
```

Variable Speed Pump

A variable speed pump (object name: Pump:VariableSpeed) is defined with maximum and minimum flow rates that are the physical limits of the device. The pump will operate and select a flow somewhere between the minimum and maximum limits. In the case where the pump is running, the pump will try to meet the flow request made by demand side components.

All of the pump rules and efficiency and power calculations are applicable from the introduction in the pump group section. The main difference between the variable volume pump and the constant volume pump is the Part Load Performance Curve. The fraction of full load power is determined by the cubic equation:

$$FractionFullLoadPower = C_1 + C_2PLR + C_3PLR^2 + C_4PLR^3$$

where $C_1, C_2, C_3,$ and C_4 are Coefficients 1 – 4 and PLR is the Part Load Ratio. In the pseudo code below the `FracFullLoadPower` modifies the `NomPowerUse` for the total pump “Power” and shows the “ShaftPower” and the “PumpHeattoFluid”.

```

VolFlowRate = PumpMassFlowRate / LoopDensity

PartLoadRatio = VolFlowRate / PumpEquip(PumpNum)%NomVolFlowRate

FracFullLoadPower = PumpEquip(PumpNum)%PartLoadCoef(1)      &
                    + PumpEquip(PumpNum)%PartLoadCoef(2) * PartLoadRatio &
                    + PumpEquip(PumpNum)%PartLoadCoef(3) * PartLoadRatio**2 &
                    + PumpEquip(PumpNum)%PartLoadCoef(4) * PartLoadRatio**3

Power = FracFullLoadPower * PumpEquip(PumpNum)%NomPowerUse

ShaftPower = Power * PumpEquip(PumpNum)%MotorEffic

! This adds the pump heat based on User input for the pump
! We assume that all of the heat ends up in the fluid eventually since this is a closed loop
! PumpHeattoFluid = ShaftPower*(1-PumpEquip(PumpNum)%PumpEffic) &

PumpHeattoFluid = ShaftPower + (Power - ShaftPower)*PumpEquip(PumpNum)%FracMotorLossToFluid

Node(OutletNode)%Temp = Node(InletNode)%Temp+PumpHeattoFluid/(PumpMassFlowRate * LoopCp)

PumpEquip(PumpNum)%Power = Power

```

Constant Speed Pump

The operation of a constant speed pump (object name: Pump:ConstantSpeed) is fairly straightforward. The user designates a maximum flow rate and when this pump operates it will run at that capacity. The main difference between the constant speed pump and the variable speed pump is that the fraction of full load power is always = 1. In the pseudo code below the FracFullLoadPower is = 1.0, therefore the Power is always the full power.

```

VolFlowRate = PumpMassFlowRate / LoopDensity

PartLoadRatio = VolFlowRate / PumpEquip(PumpNum)%NomVolFlowRate

FracFullLoadPower = 1.0

Power = FracFullLoadPower * PumpEquip(PumpNum)%NomPowerUse

ShaftPower = Power * PumpEquip(PumpNum)%MotorEffic

! This adds the pump heat based on User input for the pump
! We assume that all of the heat ends up in the fluid eventually since this is a closed loop
! PumpHeattoFluid = ShaftPower*(1-PumpEquip(PumpNum)%PumpEffic) &

PumpHeattoFluid = ShaftPower + (Power - ShaftPower)*PumpEquip(PumpNum)%FracMotorLossToFluid

Node(OutletNode)%Temp = Node(InletNode)%Temp+PumpHeattoFluid/(PumpMassFlowRate * LoopCp)

PumpEquip(PumpNum)%Power = Power

```

Pump Heat Addition to the Loop

Due to the fact that a pump is a mechanical device that acts on the fluid it is circulating, it causes the fluid to increase in temperature. The EnergyPlus model assumes that all pressure increase caused by the pump will eventually be lost due to friction, and that friction will be added as heat to the fluid. Since the plant and condenser loops are not yet true pressure-based models, EnergyPlus assumes that all of the heat resulting from the pump itself and from friction throughout the loop is added at the pump component (at the outlet node of the pump). The amount of heat added to the fluid is calculated using the following two equations:

$$\text{ShaftPower} = \text{PumpPower} * \text{PumpMotorEfficiency}$$

$$\text{PumpHeatToFluid} = \text{ShaftPower} + (\text{PumpPower} - \text{ShaftPower}) * \text{FracMotorLossToFluid}$$

where the pump motor efficiency is defined by the user input and the `FracMotorLossToFluid` is the amount of heat generated by the pump motor that is added to the fluid loop (as opposed to being lost to the environment where the pump is located). `FracMotorLossToFluid` is also a user input.

Note that the shaft power relates to the increase in head through the pump. Since all of this head is lost through the piping network due to frictional heat, this represents a heat gain by the fluid throughout the network. For simplicity, this heat is added along with the heat resulting from the pump motor. The difference between the pump power and the shaft power is the inefficiency of the pump—or the amount of energy input into the pump that the motor converts to heat rather than mechanical energy. Some of this heat is added to the fluid being pumped. These two terms are shown in the `PumpHeatToFluid` equation shown above. A simple energy balance over the pump based on the pump inlet conditions and flow rate is used to arrive at the pump outlet temperature.

Headered Pumps

The input objects `HeaderedPumps:ConstantSpeed` and `HeaderedPumps:VariableSpeed` provide models for headered pumps that consist of two or more pumps connected in parallel. The headered pump is simulated as a single component, and it is specified as an integer number of a specific pump. The flow rate provided by the headered pump is determined by the number of pumps in operation and the flow rate of the individual pump. The total flow rate is calculated as

$$FlowProvided = NumPumpsON * IndividualPumpFlowRate$$

The simulation starts by turning ON all pumps in the group. The pumps are then turned OFF one at a time until the flow provided is less than the flow requested. Finally the last pump is turned back ON to meet the remaining flow (`FlowDifference`) requested. The flow rate of the last pump depends on the pump bank type. For constant speed headered pumps, the last pump runs at the nominal flow rate, thereby giving a final headered pump flow which is equal to or greater than the flow requested. In a variable speed headered pump the last pump runs at part load so that the flow provided matches the flow requested. The power of the headered pump is then calculated as

$$Power = (PR_{FL} * N_{FL} + PR_{PL} * N_{PL}) * P_{Nom}$$

Where;

$Power$ = Power consumed by the pump bank

PR_{FL} = Power ratio at full load (Generally equal to 1)

N_{FL} = Number of pumps running at full load

PR_{PL} = Power ratio at part load

N_{PL} = Number of pumps running at part load

P_{Nom} = Nominal power consumption of individual pumps

For a constant speed headered pump N_{PL} is zero. For a variable speed headered pump N_{PL} is equal to one.

Condensate Pumps

The input object `Pump:VariableSpeed:Condensate` provides a model for steam condensate pumps, see the discussion for steam loops, reference: `Condensate Pump`.

Ideal Loads Air System

Overview

The input object ZoneHVAC:IdealLoadsAirSystem provides a model for an ideal HVAC system. It occupies a place in the program hierarchy corresponding to a terminal unit. It is not connected to a central air system – instead each ZoneHVAC:IdealLoadsAirSystem object supplies cooling or heating air to a zone in sufficient quantity to meet the zone load. The supply air conditions are specified in the ZoneHVAC:IdealLoadsAirSystem input.

Model

The ZoneHVAC:IdealLoadsAirSystem object is modeled as an ideal VAV terminal unit. The supply air flow rate is varied between zero and the maximum in order to satisfy the zone heating or cooling load.

Inputs and Data

The user specifies the following data for each ZoneHVAC:IdealLoadsAirSystem object:

- name of the zone inlet node;
- supply air temperature when in heating mode $T_{s,heating}$ [C];
- supply air temperature when in cooling mode $T_{s,cooling}$ [C];
- supply air humidity ratio when in heating mode $W_{s,heating}$ [kg water/kg dry air];
- supply air humidity ratio when in cooling mode $W_{s,cooling}$ [kg water/kg dry air];
- heating limit flag (*Limit* or *No Limit*) *HeatFlowLimit*;
- maximum heating air flow rate [m³/s]
- cooling limit flag (*Limit* or *No Limit*) *CoolFlowLimit*;
- maximum cooling air flow rate [m³/s]
- outside air flag (*Outside Air* or *No Outside Air*)
- outside air flow rate [m³/s]
- name of heating availability schedule
- name of cooling availability schedule

All input data for the ZoneHVAC:IdealLoadsAirSystem is stored in the array *PurchAir*. The model and data are encapsulated in the module *PurchasedAirManager*.

Calculation

1. Set the unit on/off flag *UnitOn*. The unit is off (*UnitOn = False*) if:
 - a. the zone temperature is in the thermostat deadband;
 - b. there is a heating load ($\dot{Q}_z > 0$) and the current heating availability schedule value is 0;
 - c. there is a cooling load ($\dot{Q}_z < 0$) and the current cooling availability schedule value is 0;
 - d. the zone load is very small ($\dot{Q}_z \leq \dot{Q}_{small} = 1$ watt);

otherwise the unit is on (*UnitOn = True*). If the unit is on, the calculation proceeds through steps 2 – 6. If the unit is off, the zone inlet node conditions are set to the zone node condition, the inlet node mass flow rate is set to zero, and the unit outputs are set to zero.

2. Set the zone inlet node conditions.

If $\dot{Q}_z < 0$ then

$$T_{node, zin} = T_{s, cooling}$$

$$W_{node, zin} = W_{s, cooling}$$

otherwise

$$T_{node, zin} = T_{s, heating}$$

$$W_{node, zin} = W_{s, heating}$$

and $h_{node, zin} = \text{PsyHFnTdbW}(T_{node, zin}, W_{node, zin})$, where h is specific enthalpy and PsyHFnTdbW is the EnergyPlus psychrometric function for air enthalpy as a function of drybulb temperature and humidity ratio.

3. Calculate the supply air flow rate that will satisfy the load.

$$\dot{m}_s = \dot{Q}_z / (c_{p, air} \cdot (T_{node, zin} - T_z))$$

4. Set limits on supply mass flow rate. If $\dot{Q}_z < 0$ \dot{m}_s must be less than or equal to the cooling mass flow rate limit. If $\dot{Q}_z > 0$ \dot{m}_s must be less than or equal to the heating mass flow rate limit. \dot{m}_s must be greater than zero. Set

$$\dot{m}_{node, zin} = \dot{m}_s$$

5. Set the unit entering conditions, taking into account the outside air (oa).
- The recirculation air conditions are set equal to the zone return air node conditions; if there is no return air node the recirculation air conditions are set equal to the conditions at the zone node.
 - The unit entering conditions are then:

If $\dot{m}_{entering} > \dot{m}_{oa}$ then

$$h_{entering} = (\dot{m}_{oa} \cdot h_{oa} + (\dot{m}_s - \dot{m}_{oa}) \cdot h_{recirc}) / \dot{m}_s$$

$$W_{entering} = (\dot{m}_{oa} \cdot W_{oa} + (\dot{m}_s - \dot{m}_{oa}) \cdot W_{recirc}) / \dot{m}_s$$

$$T_{entering} = \text{PsyHFnTdbW}(h_{entering}, W_{entering})$$

Otherwise the entering air conditions are set equal to the outside air conditions.

6. Calculate the unit output. If the supply air flow rate (\dot{m}_s) is less than or equal to zero, the unit heating / cooling output is set to zero. Otherwise,

$$\dot{Q}_{purch air} = \dot{m}_s \cdot c_{p, air} \cdot (T_{node, zin} - T_{entering})$$

$$\dot{Q}_{purch air, heating} = \max(\dot{Q}_{purch air}, 0)$$

$$\dot{Q}_{purch air, sens cooling} = \left| \min(\dot{Q}_{purch air}, 0) \right|$$

$$\dot{Q}_{purch air, tot cooling} = \left| \min(\dot{m}_s \cdot (h_{node, zin} - h_{entering}), 0) \right|$$

References

No specific references.

District Cooling

When the user is not interested in a plant simulation or there is some centralized source of chilled water, the following model can be used in the input. This allows the user to achieve a simulation without specifying operating parameters or curve fits for chiller models. This model only needs the connections to the loop and the nominal capacity to simulate. See the InputOutput Reference for additional information (Object: DistrictCooling). This model calculates the output capacity necessary from the inlet temperature to the setpoint temperature for that loop with the given mass flow rate in Watts.

District Heating

When the user is not interested in a plant simulation or there is some centralized source of hot water, the following model can be used in the input. This allows the user to achieve a simulation without specifying operating parameters or curve fits for boiler models. This model only needs the connections to the loop and the nominal capacity to simulate. See the InputOutput Reference for additional information (Object: DistrictHeating). This model calculates the output capacity necessary from the inlet temperature to the setpoint temperature for that loop with the given mass flow rate in Watts.

Radiant System Models

Low Temperature Radiant System Model

The input objects ZoneHVAC:LowTemperatureRadiant:ConstantFlow, ZoneHVAC:LowTemperatureRadiant:VariableFlow, and ZoneHVAC:LowTemperatureRadiant:Electric provide models for low temperature radiant heating and cooling systems that appear, on the surface, to be relatively simple systems. The system circulates hot or cold fluid through tubes embedded in a wall, ceiling, or floor or runs current through electric resistance wires embedded in a surface or a panel. Energy is thus either added to or removed from the space, and zone occupants are conditioned by both radiation exchange with the system and convection from the surrounding air that is also affected by the system. Unless specifically required for indoor air quality considerations, fans, ductwork, dampers, etc. are not needed.

Despite the relative simplicity of the low temperature radiant systems, the integration of such a system within an energy analysis program requires one to overcome several challenges. First, for systems with significant thermal mass, the conduction transfer function method for modeling transient conduction must be extended to include embedded heat sources or sinks. Second, one must integrate this formulation within an energy analysis program like EnergyPlus. Finally, one must overcome the fact that the radiant system is both a zone heat balance element and a conditioning system. Each of these issues will be addressed in the next several subsections.

One Dimensional Heat Transfer Through Multilayered Slabs

One of the most important forms of heat transfer in energy analysis is heat conduction through building elements such as walls, floors, and roofs. While some thermally lightweight structures can be approximated by steady state heat conduction, a method that applies to all structures must account for the presence of thermal mass within the building elements. Transient one dimensional heat conduction through a homogeneous layer with constant thermal properties such as the one shown in Figure 202 is governed by the following equation:

$$\frac{\partial^2 T}{\partial x^2} = \frac{1}{\alpha} \frac{\partial T}{\partial t} \quad (471)$$

where: T is the temperature as a function of position and time,
 x is the position,
 t is the time,

$\alpha = \frac{k}{\rho c_p}$ is the thermal diffusivity of the layer material,

k is its thermal conductivity,

ρ is its density, and

c_p is its specific heat.

This equation is typically coupled with Fourier's law of conduction that relates the heat flux at any position and time to temperature as follows:

$$q''(x,t) = -k \frac{\partial T(x,t)}{\partial x} \quad (472)$$

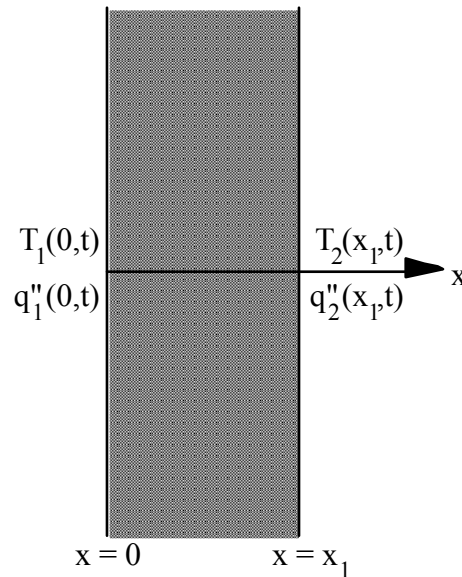


Figure 202. Single Layered Building Element

While analytical solutions exist for the single homogeneous layer shown in Figure 202, the solution becomes extremely tedious for the multiple layered slab shown in Figure 203.

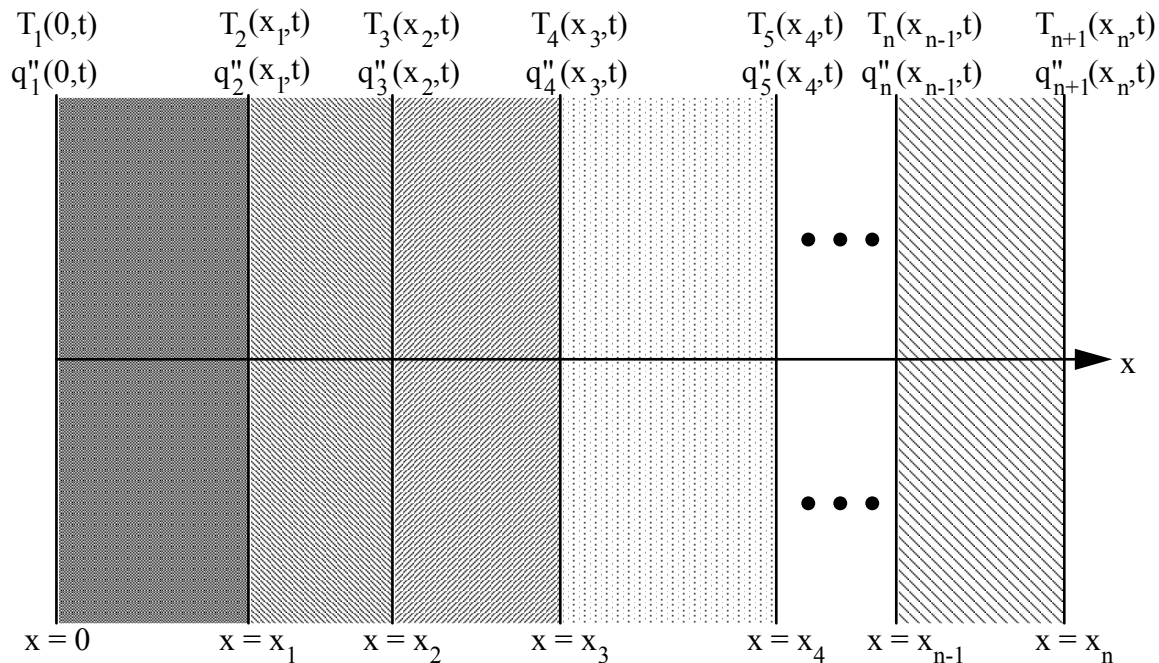


Figure 203. Multilayered Building Element

Time Series Solutions: Conduction Transfer Functions

Equations (471) and (472) can be solved numerically in a variety of ways. As mentioned in the previous section, other models have used control theory and numerical methods such as finite difference and finite element. However, each of these methods have drawbacks which render them inappropriate for use within an energy analysis program which requires both accuracy and efficiency from the simulation.

Another possible modeling method is a time series solution. Several of the detailed energy analysis programs such as EnergyPlus use a time series solution to transient heat conduction. The most basic time series solution is the response factor equation which relates the flux at one surface of an element to an infinite series of temperature histories at both sides as shown by:

$$q''_{i,t} = \sum_{m=1}^{\infty} X_m T_{i,t-m+1} - \sum_{m=1}^{\infty} Y_m T_{o,t-m+1} \quad (473)$$

where q'' is heat flux, T is temperature, i signifies the inside of the building element, o signifies the outside of the building element, and t represents the current time step.

While in most cases the terms in the series decay fairly rapidly, the infinite number of terms needed for an exact response factor solution makes it less than desirable. Fortunately, the similarity of higher order terms can be used to replace them with flux history terms. The new solution contains elements that are called conduction transfer functions (CTFs). The basic form of a conduction transfer function solution is shown by the following equation:

$$q''_{i,t} = \sum_{m=1}^M X_m T_{i,t-m+1} - \sum_{m=1}^M Y_m T_{o,t-m+1} + \sum_{m=1}^k F_m q''_{i,t-m} \quad (474)$$

where k is the order of the conduction transfer functions, M is a finite number defined by the order of the conduction transfer functions, and X , Y , and F are the conduction transfer functions. This equation states that the heat flux at the interior surface of any generic

building element for which the assumption of one dimensional conduction heat transfer is valid is linearly related to the current and some of the previous temperatures at both the interior and exterior surface as well as some of the previous flux values at the interior surface. A similar equation holds for the heat flux at the exterior surface.

The final CTF solution form reveals why it is so elegant and powerful. With a single, relatively simple equation, the conduction heat transfer through an element can be calculated. The coefficients (CTFs) in the equation are constants that only need to be determined once. The only storage of data required is the CTFs themselves and a limited number of temperature and flux terms. The formulation is valid for any surface type and does not require the calculation or storage of element interior temperatures.

As the next several sections will detail, there are two main methods for calculating conduction transfer functions: the Laplace Transform method and the State Space method. Both methods are well suited for the main focus of this research, the extension of conduction transfer functions to include heat sources or sinks.

Laplace Transform Formulation

The traditional method for calculating conduction transfer functions is described in detail by Hittle (1981). Beginning with the transient one dimensional heat conduction equation {Equation (471)} and Fourier's law of conduction {Equation (472)}, the Laplace transform method is used to convert the governing equations into the s-domain for a single layer such as the one shown in Figure 202.

$$\frac{d^2T(x,s)}{dx^2} = \frac{s}{\alpha} T(x,s) \quad (475)$$

$$q''(x,s) = -k \frac{dT(x,s)}{dx} \quad (476)$$

The transformed equations are solved and then put in matrix form as shown below:

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} T_2(s) \\ q_2(s) \end{bmatrix} \quad (477)$$

where: $T_1(s)$, $T_2(s)$, $q_1(s)$, and $q_2(s)$ are the temperature and flux terms in the Laplace domain,

$$A_1(s) = \cosh\left(\ell_1 \sqrt{s/\alpha_1}\right),$$

$$B_1(s) = \left(1/k_1 \sqrt{s/\alpha_1}\right) \sinh\left(\ell_1 \sqrt{s/\alpha_1}\right),$$

$$C_1(s) = k_1 \sqrt{s/\alpha_1} \sinh\left(\ell_1 \sqrt{s/\alpha_1}\right),$$

$$D_1(s) = \cosh\left(\ell_1 \sqrt{s/\alpha_1}\right),$$

k_1 is the thermal conductivity of the layer,

α_1 is the thermal diffusivity of the layer, and

ℓ_1 is the thickness of the layer.

The 2 x 2 matrix consisting of $A_1(s)$, $B_1(s)$, $C_1(s)$, and $D_1(s)$ is called the transmission matrix and contains all of the thermophysical properties of the layer necessary to calculate transient

conduction heat transfer through it. It can easily be shown that a second layer could be characterized in a similar way as:

$$\begin{bmatrix} T_2(s) \\ q_2(s) \end{bmatrix} = \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \begin{bmatrix} T_3(s) \\ q_3(s) \end{bmatrix} \quad (478)$$

where $A_2(s)$, $B_2(s)$, $C_2(s)$, and $D_2(s)$ are calculated using the properties of the second layer. This can be substituted into Equation (477) to provide insight how the extension to multilayered slabs is achieved.

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \begin{bmatrix} T_3(s) \\ q_3(s) \end{bmatrix} \quad (479)$$

Thus, for a multilayered element as shown in Figure 203, each separate layer has a transmission matrix of $A_i(s)$, $B_i(s)$, $C_i(s)$, and $D_i(s)$ associated with it. The form of the matrix equation for the multilayered element is the same as the equation for a single layer:

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A(s) & B(s) \\ C(s) & D(s) \end{bmatrix} \begin{bmatrix} T_{n+1}(s) \\ q_{n+1}(s) \end{bmatrix} \quad (480)$$

but the transmission matrix is replaced by:

$$\begin{bmatrix} A(s) & B(s) \\ C(s) & D(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \dots \begin{bmatrix} A_n(s) & B_n(s) \\ C_n(s) & D_n(s) \end{bmatrix} \quad (481)$$

Equation (480) is typically rearranged as follows:

$$\begin{bmatrix} q_1(s) \\ q_{n+1}(s) \end{bmatrix} = \begin{bmatrix} \frac{D(s)}{B(s)} & \frac{-1}{B(s)} \\ \frac{1}{B(s)} & \frac{-A(s)}{B(s)} \end{bmatrix} \begin{bmatrix} T_1(s) \\ T_{n+1}(s) \end{bmatrix} \quad (482)$$

which relates the flux at either surface of the element to the temperature histories at both surfaces. When the temperature histories are formulated as triangular pulses made up of simple ramp functions, the roots of this equation can be found and result in response factors. The response factors can be simplified as described above through the introduction of flux history terms to form conduction transfer functions. A simplified method of finding the roots of the Laplace domain equations is described by Hittle and Bishop (1983) and is used by the current version of BLAST.

State Space Formulation

Recently, another method of finding conduction transfer functions starting from a state space representation has begun receiving increased attention (Ceylan and Myers 1980; Seem 1987; Ouyang and Haghghat 1991). The basic state space system is defined by the following linear matrix equations:

$$\frac{d[x]}{dt} = [A][x] + [B][u] \quad (483)$$

$$[y] = [C][x] + [D][u] \quad (484)$$

where x is a vector of state variables, u is a vector of inputs, y is the output vector, t is time, and A , B , C , and D are coefficient matrices. Through the use of matrix algebra, the vector of state variables (x) can be eliminated from the system of equations, and the output vector (y) can be related directly to the input vector (u) and time histories of the input and output vectors.

This formulation can be used to solve the transient heat conduction equation by enforcing a finite difference grid over the various layers in the building element being analyzed. In this case, the state variables are the nodal temperatures, the environmental temperatures (interior and exterior) are the inputs, and the resulting heat fluxes at both surfaces are the outputs. Thus, the state space representation with finite difference variables would take the following form:

$$\frac{d \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix}}{dt} = [A] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [B] \begin{bmatrix} T_i \\ T_o \end{bmatrix} \quad (485)$$

$$\begin{bmatrix} q_i'' \\ q_o'' \end{bmatrix} = [C] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [D] \begin{bmatrix} T_i \\ T_o \end{bmatrix} \quad (486)$$

where $T_1, T_2, \dots, T_{n-1}, T_n$ are the finite difference nodal temperatures, n is the number of nodes, T_i and T_o are the interior and exterior environmental temperatures, and q_i'' and q_o'' are the heat fluxes (desired output).

Seem (1987) shows that for a simple one layer slab with two interior nodes as in Figure 204 and convection at both sides the resulting finite difference equations are given by:

$$C \frac{dT_1}{dt} = hA(T_o - T_1) + \frac{T_2 - T_1}{R} \quad (487)$$

$$C \frac{dT_2}{dt} = hA(T_i - T_2) + \frac{T_1 - T_2}{R} \quad (488)$$

$$q_i'' = h(T_i - T_2) \quad (489)$$

$$q_o'' = h(T_1 - T_o) \quad (490)$$

where: $R = \frac{\ell}{kA}$, thermal resistance

$$C = \frac{\rho c_p \ell A}{2}, \text{ thermal capacitance}$$

T_o = outside temperature

T_i = inside temperature

T_1 = temperature of node 1

T_2 = temperature of node 2

and

A is the area of the surface exposed to the environmental temperatures.

In matrix format:

$$\begin{bmatrix} \frac{dT_1}{dt} \\ \frac{dT_2}{dt} \end{bmatrix} = \begin{bmatrix} \frac{-1}{RC} - \frac{hA}{C} & \frac{1}{RC} \\ \frac{1}{RC} & \frac{-1}{RC} - \frac{hA}{C} \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} \frac{hA}{C} & 0 \\ 0 & \frac{hA}{C} \end{bmatrix} \begin{bmatrix} T_o \\ T_i \end{bmatrix} \quad (491)$$

$$\begin{bmatrix} q_i'' \\ q_o'' \end{bmatrix} = \begin{bmatrix} 0 & -h \\ h & 0 \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} 0 & h \\ -h & 0 \end{bmatrix} \begin{bmatrix} T_o \\ T_i \end{bmatrix} \quad (492)$$

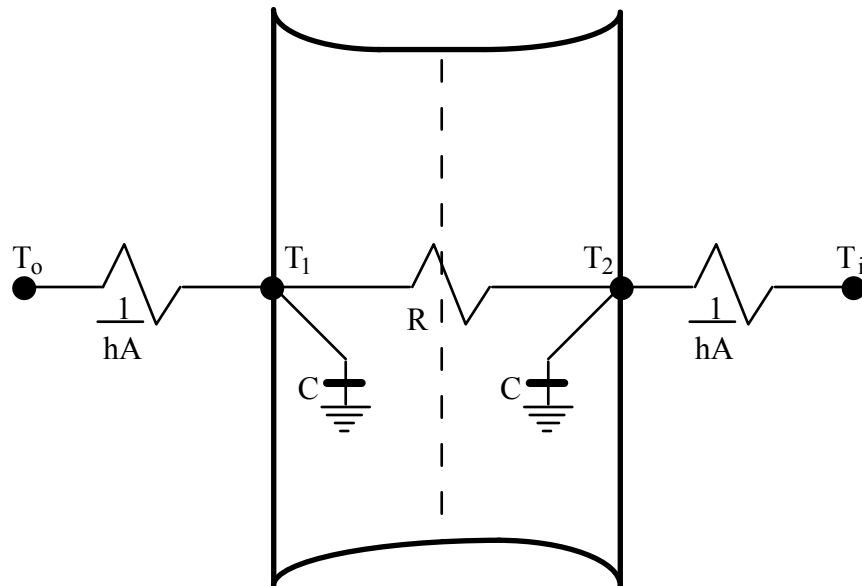


Figure 204. Two Node State Space Example

The important aspect of the state space technique is that through the use of relatively simple matrix algebra the state space variables (nodal temperatures) can be eliminated to arrive at a matrix equation that gives the outputs (heat fluxes) as a function of the inputs (environmental temperatures) only. This eliminates the need to solve for roots in the Laplace domain. In addition, the resulting matrix form has more physical meaning than complex functions required by the Laplace transform method. The current version of EnergyPlus uses the state space method for computing CTFs.

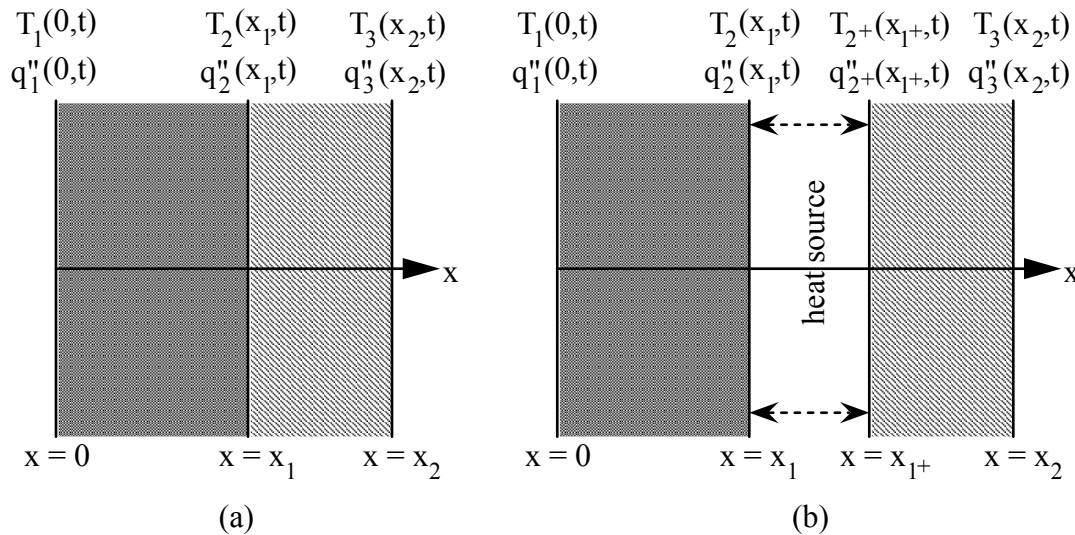
The accuracy of the state space method of calculating CTFs has been addressed in the literature. Ceylan and Myers (1980) compared the response predicted by the state space method to various other solution techniques including an analytical solution. Their results showed that for an adequate number of nodes the state space method computed a heat flux at the surface of a simple one layer slab within 1% of the analytical solution. Ouyang and Haghghat (1991) made a direct comparison between the Laplace and state space methods. For a wall composed of insulation between two layers of concrete, they found almost no difference in the response factors calculated by each method.

Extension of Time Series Solutions to Include Heat Sources and Obtain Internal Temperatures

Laplace Transform Formulation

Degiovanni (1988) proposed two methodologies for including sources or sinks in the Laplace Transform Formulation. The first method shows how a source that varies as a function of time and location can be incorporated. The resulting equations involve some fairly complicated terms including spatial derivatives.

The second method that will be analyzed in more detail involves the addition of a source or sink at the interface between two layers. The derivation of the necessary equations is begun by analyzing the simple two layer element shown in Figure 205.



$$\begin{bmatrix} T_2(s) \\ q_2(s) \end{bmatrix} = \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \begin{bmatrix} T_3(s) \\ q_3(s) \end{bmatrix}$$

Figure 205. Two Layer Example for Deriving the Laplace Transform Extension to Include Sources and Sinks

For the first layer, it was determined that in the Laplace domain

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} T_2(s) \\ q_2(s) \end{bmatrix} \tag{493}$$

For the second layer:

$$\tag{494}$$

To link the two layers and include the heat source between them, the following substitution is made:

$$\begin{bmatrix} T_2(s) \\ q_2(s) \end{bmatrix} = \begin{bmatrix} T_{2+}(s) \\ q_{2+}(s) \end{bmatrix} + \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \quad (495)$$

which results in:

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \left\{ \begin{bmatrix} T_{2+}(s) \\ q_{2+}(s) \end{bmatrix} + \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \right\} \quad (496)$$

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \left\{ \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \begin{bmatrix} T_3(s) \\ q_3(s) \end{bmatrix} + \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \right\} \quad (497)$$

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} A_2(s) & B_2(s) \\ C_2(s) & D_2(s) \end{bmatrix} \begin{bmatrix} T_3(s) \\ q_3(s) \end{bmatrix} + \begin{bmatrix} A_1(s) & B_1(s) \\ C_1(s) & D_1(s) \end{bmatrix} \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \quad (498)$$

While Degiovanni concludes with this formula, some insight into what the generic equation for an element that has n layers might look like is gained by working with Equation (498). If a layer is added to the left of the first layer, the entire right hand side of Equation (498) is multiplied by the transmission matrix of the new layer. Conversely, if a layer is added to the right of the second layer in Figure 205, the vector containing the Laplace transform of the temperature and heat flux at interface 3 is replaced by the product of the transmission matrix of the new layer and the vector for temperature and heat flux at the next interface, and the term dealing with the heat source is not affected. The general equation for a building element with n layers and m layers between the left hand surface and the heat source can be derived as:

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \left(\prod_{i=1}^n \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix} \right) \begin{bmatrix} T_{n+1}(s) \\ q_{n+1}(s) \end{bmatrix} + \left(\prod_{i=1}^m \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix} \right) \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \quad (499)$$

or in more compact form:

$$\begin{bmatrix} T_1(s) \\ q_1(s) \end{bmatrix} = \begin{bmatrix} A(s) & B(s) \\ C(s) & D(s) \end{bmatrix} \begin{bmatrix} T_{n+1}(s) \\ q_{n+1}(s) \end{bmatrix} + \begin{bmatrix} a(s) & b(s) \\ c(s) & d(s) \end{bmatrix} \begin{bmatrix} 0 \\ q_{source}(s) \end{bmatrix} \quad (500)$$

where: $\begin{bmatrix} A(s) & B(s) \\ C(s) & D(s) \end{bmatrix} = \prod_{i=1}^n \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix}$ and

$$\begin{bmatrix} a(s) & b(s) \\ c(s) & d(s) \end{bmatrix} = \prod_{i=1}^m \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix}.$$

Next, Equation (500) must be rearranged to match the form of Equation (482), which relates the heat flux at both sides of the element to the temperature at each side. The matrix equation that is obtained shows that:

$$\begin{bmatrix} q_1(s) \\ q_{n+1}(s) \end{bmatrix} = \begin{bmatrix} \frac{D(s)}{B(s)} & \frac{-1}{B(s)} \\ \frac{1}{B(s)} & \frac{-A(s)}{B(s)} \end{bmatrix} \begin{bmatrix} T_1(s) \\ T_{n+1}(s) \end{bmatrix} + \begin{bmatrix} d(s) - \frac{D(s)b(s)}{B(s)} \\ \frac{b(s)}{B(s)} \end{bmatrix} [q_{source}(s)] \quad (501)$$

This equation bears a striking resemblance to Equation (482). If the source term in Equation (501) is dropped, then the equation is identical to Equation (482). This result conforms with the superposition principle which was used to develop the conduction transfer functions from the summation of a series of triangular pulses or ramp sets. Now, the effect of the heat source is simply added to the response to the temperature inputs.

While Equation (501) is correct for any single or multilayered element, the first term in the heat source transmission matrix does not appear to match the compactness of the other terms in the matrix equation. It can be shown (see Strand 1995: equations 32 through 42 which detail this derivation) that the heat source transmission term for a two-layer problem reduces to

$$\begin{bmatrix} q_1(s) \\ q_3(s) \end{bmatrix} = \begin{bmatrix} \frac{D(s)}{B(s)} & \frac{-1}{B(s)} \\ \frac{1}{B(s)} & \frac{-A(s)}{B(s)} \end{bmatrix} \begin{bmatrix} T_1(s) \\ T_3(s) \end{bmatrix} + \begin{bmatrix} \frac{B_2(s)}{B(s)} \\ \frac{B_1(s)}{B(s)} \end{bmatrix} [q_{source}(s)] \quad (502)$$

If this is extended to a slab with n layers and a source between the m and $m+1$ layers, the general matrix equation for obtaining heat source transfer functions using the Laplace transform method is:

$$\begin{bmatrix} q_1(s) \\ q_{n+1}(s) \end{bmatrix} = \begin{bmatrix} \frac{D(s)}{B(s)} & \frac{-1}{B(s)} \\ \frac{1}{B(s)} & \frac{-A(s)}{B(s)} \end{bmatrix} \begin{bmatrix} T_1(s) \\ T_{n+1}(s) \end{bmatrix} + \begin{bmatrix} \bar{b}(s) \\ b(s) \\ \bar{b}(s) \end{bmatrix} [q_{source}(s)] \quad (503)$$

where: $\begin{bmatrix} A(s) & B(s) \\ C(s) & D(s) \end{bmatrix} = \prod_{i=1}^n \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix}$,

$$\begin{bmatrix} a(s) & b(s) \\ c(s) & d(s) \end{bmatrix} = \prod_{i=1}^m \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix}, \text{ and}$$

$$\begin{bmatrix} \bar{a}(s) & \bar{b}(s) \\ \bar{c}(s) & \bar{d}(s) \end{bmatrix} = \prod_{i=m+1}^n \begin{bmatrix} A_i(s) & B_i(s) \\ C_i(s) & D_i(s) \end{bmatrix}.$$

At first glance, the terms in the heat source transmission matrix may appear to be reversed. It is expected that only the layers to the left of the source will affect $q_1(s)$, but the presence of

$\bar{b}(s)$ in the element multiplied by $q_{\text{source}}(s)$ to obtain $q_1(s)$ seems to be contradictory. In fact, the entire term, $\bar{b}(s)/B(s)$, must be analyzed to determine the effect of $q_{\text{source}}(s)$ on $q_1(s)$. In essence, the appearance of $\bar{b}(s)$ removes the effects of the layers to the right of the source from $B(s)$ leaving only the influence of the layers to the left of the source. The form displayed by Equation (503) is, however, extremely convenient because the terms in the heat source transmission matrix have the same denominators, and thus roots, as the terms in the temperature transmission matrix. Thus, the same roots that are calculated for the CTFs can be used for the QTFs, saving a considerable amount of computer time during the calculation of the transfer functions.

Once Equation (503) is inverted from the Laplace domain back into the time domain, the combined CTF–QTF solution takes the following form:

$$q''_{i,t} = \sum_{m=1}^M X_m T_{i,t-m+1} - \sum_{m=1}^M Y_m T_{o,t-m+1} + \sum_{m=1}^k F_m q''_{i,t-m} + \sum_{m=1}^M W_m q_{\text{source},t-m+1} \quad (504)$$

This relation is identical to Equation (474) except for the presence of the QTF series that takes the heat source or sink into account.

State Space Formulation

The two-node example introduced by Seem (1987) can be utilized to examine the extension of the state space method to include heat sources or sinks. Figure 206 shows the simple two node network with a heat source added at node 1.

The nodal equations for the finite difference network shown in Figure 206 are:

$$C \frac{dT_1}{dt} = hA(T_o - T_1) + \frac{T_2 - T_1}{R} + q_{\text{source}}A \quad (505)$$

$$C \frac{dT_2}{dt} = hA(T_i - T_2) + \frac{T_1 - T_2}{R} \quad (506)$$

$$q''_i = h(T_i - T_2) \quad (507)$$

$$q''_o = h(T_1 - T_o) \quad (508)$$

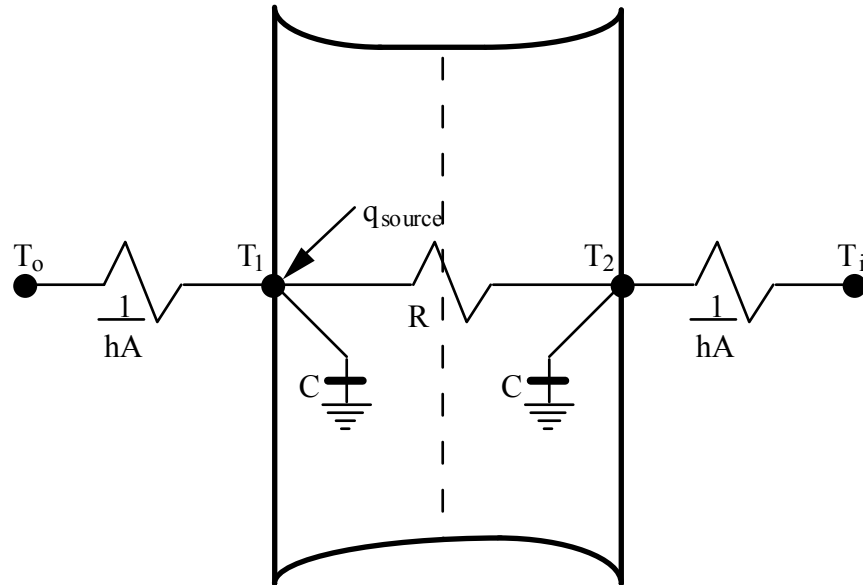


Figure 206. Two Node State Space Example with a Heat Source

In obtaining the matrix equivalent for this set of equations, it should be noted that the source term is not a constant but rather an input that varies with time. Thus, it must be grouped with the environmental temperatures as inputs. The resulting matrix equations take the following form:

$$\begin{bmatrix} \frac{dT_1}{dt} \\ \frac{dT_2}{dt} \end{bmatrix} = \begin{bmatrix} \frac{-1}{RC} - \frac{hA}{C} & \frac{1}{RC} \\ \frac{1}{RC} & \frac{-1}{RC} - \frac{hA}{C} \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} \frac{hA}{C} & 0 & \frac{A}{C} \\ 0 & \frac{hA}{C} & 0 \end{bmatrix} \begin{bmatrix} T_o \\ T_i \\ q_{source} \end{bmatrix} \quad (509)$$

$$\begin{bmatrix} q_1'' \\ q_2'' \end{bmatrix} = \begin{bmatrix} 0 & -h \\ h & 0 \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} 0 & h & 0 \\ -h & 0 & 0 \end{bmatrix} \begin{bmatrix} T_o \\ T_i \\ q_{source} \end{bmatrix} \quad (510)$$

Equation (510) appears to suggest that the source term has no direct effect on the heat flux at either side of the element because its coefficients are zero. This is not the case. Equation (510) only relates variables that have a direct influence on heat flux. So, while T_i has no direct influence on q_o'' , it does have an indirect influence through the nodal network. The same would hold for the influence of q_{source} .

If this analysis is extended to a finite difference network with n nodes, the corresponding matrix equations can be shown to be:

$$\frac{d \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix}}{dt} = [A] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [B] \begin{bmatrix} T_o \\ T_i \\ q_{source} \end{bmatrix} \quad (511)$$

$$\begin{bmatrix} q_i'' \\ q_o'' \end{bmatrix} = [C] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [D] \begin{bmatrix} T_o \\ T_i \\ q_{source} \end{bmatrix} \quad (512)$$

The influence of the heat source is also confirmed by the final solution form, which is identical to the Laplace transform result shown in Equation (504). As with the Laplace solution method, the state space method results in a set of QTFs that relate the heat source at the current time step and several previous time steps to the current heat flux at the surface of the element.

Other similarities between the two solution methods are evident. It is interesting to note that as with the Laplace method there is no alteration of the CTFs calculated by the state space method. Thus, the principle of superposition is still valid. Furthermore, the introduction of the source term did not substantially increase the computing effort required to calculate the additional transfer functions. In the Laplace method, this was shown by the common roots, $B(s)$, shared by both the CTFs and the QTFs. In the state space method, it can be noted that the A matrices in Equations (491) and (509) are identical. Since the state space method requires the inversion and the exponentiation of the A matrix only, the additional QTF terms will not require a substantial amount of additional computing time for their calculation.

Determination of Internal Temperatures

One aspect of low temperature radiant systems that has not been addressed to this point is the appropriateness of specifying the effect of the system on slab response via a heat source term. For a heating system that employs electrical resistance heating, the use of a heat source as the input variable is logical. The heat produced by such a system can easily be related to the current passing through the heating wire. However, for a hydronic heating or cooling system, the known quantity is not heat but rather the temperature of the water being sent to the building element.

The use of a temperature to simulate the presence of a heating or cooling system presents one major obstacle. When fluid is not being circulated, there is no readily available temperature value available for use as an input variable.

In a hydronic system, a link between the fluid temperature being sent to the slab and the heat delivered to the slab exist. The most effective way of relating these two variables is to consider the slab to be a heat exchanger. Using heat exchanger relationships, an equation could then be formulated to obtain the heat delivered to the slab based on the inlet fluid temperature.

Most heat exchangers are used to thermally link two fluids. In the case of a hydronic radiant system, there is only one fluid and a stationary solid. Presumably, if the inlet fluid temperature, the system geometry, and the solid temperature are known, then the outlet temperature and thus the heat transfer to the building element can be computed. This leads to an interesting question: what is the solid temperature?

By definition, for one dimensional conduction heat transfer, the solid temperature is the temperature of the building element at the depth where the hydronic loop is located. Typically, this temperature is not known because it is not needed. The goal of both methods of calculating CTFs was the elimination of internal temperatures that were not needed for the simulation. For a hydronic system, it is necessary to extract this information to solve for the heat source term. Two methods of accomplishing this are described below.

Returning to the two layer example shown in Figure 205, it can be shown that the final solution form in the time domain for the slab with a source at the interface between the two layers is:

$$q''_{1,t} = \sum_{m=1}^M X_{k,m} T_{1,t-m+1} - \sum_{m=1}^M Y_{k,m} T_{2,t-m+1} + \sum_{m=1}^k F_m q''_{1,t-m} + \sum_{m=1}^M W_m q_{source,t-m+1} \quad (513)$$

A similar equation could be written for the response of the first layer in absence of any source term and is given by:

$$q''_{1,t} = \sum_{m=1}^M x_{k,m} T_{1,t-m+1} - \sum_{m=1}^M y_{k,m} T_{2,t-m+1} + \sum_{m=1}^k f_m q''_{1,t-m} \quad (514)$$

While the current temperature at the interface is not known, presumably the previous values of this parameter will be known. In addition, the temperatures and the flux histories at surface 1 are also known. The unknowns in Equation (514) are the current heat flux at surface 1 and the temperature at surface 2. However, Equation (513) does define the current value of the heat flux at surface 1 based on temperature, heat flux, and heat source histories. Thus, if this value is used in Equation (514), the only remaining unknown in this equation is the current temperature at surface 2, the surface where the heat source or sink is present. Rearranging Equation (514) provides an equation from which the temperature at the source location may be calculated:

$$T_{2,t} = \sum_{m=1}^M \bar{X}_{k,m} T_{1,t-m+1} - \sum_{m=1}^{M-1} \bar{Y}_{k,m} T_{2,t-m} + \sum_{m=1}^{k+1} \bar{F}_m q''_{1,t-m+1} \quad (515)$$

where the new coefficients are obtained from the standard conduction transfer functions for the first layer via the following equations:

$$\bar{X}_{k,m} = \frac{x_{k,m}}{y_1} \quad (m = 1, \dots, M) \quad (516)$$

$$\bar{Y}_{k,m} = \frac{y_{k,m+1}}{y_1} \quad (m = 1, \dots, M-1) \quad (517)$$

$$\bar{F}_1 = \frac{1}{y_1} \quad (518)$$

$$\bar{F}_m = \frac{f_{m-1}}{y_1} \quad (m = 2, \dots, k+1) \quad (519)$$

This system for backing out an internal temperature through the use of a second, rearranged CTF equation is valid regardless of whether the Laplace transform or state space method is utilized to calculate the CTFs and QTFs. The state space method, however, offers a more direct method of obtaining an internal temperature through its definition as an additional output variable.

Consider again the state space example shown in Figure 206. Two output variables were defined for this example: q''_i and q''_o . The temperature of the node where the source is present can also be defined as an output variable through the identity equation:

$$T_1 = T_1 \quad (520)$$

When this equation for T_1 is added to Equation (510), the resulting output matrix equation for the heat flux at both surfaces and the internal temperature is:

$$\begin{bmatrix} q_i'' \\ q_o'' \\ T_1 \end{bmatrix} = \begin{bmatrix} 0 & -h \\ h & 0 \\ 1 & 0 \end{bmatrix} \begin{bmatrix} T_1 \\ T_2 \end{bmatrix} + \begin{bmatrix} 0 & h & 0 \\ -h & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} T_i \\ T_o \\ q_{source} \end{bmatrix} \quad (521)$$

The only difference between this relation and Equation (510) is the presence of T_1 on both the right and left hand side of the equation. The dual role of T_1 as a state variable and an output parameter may seem to contradict the goal of the state space method of eliminating the state variables. However, due to the flexibility of the formulation, nodal temperatures can be extracted in the same manner that any other output quantity would be obtained. For an element with n layers, Equation (521) becomes:

$$\begin{bmatrix} q_i'' \\ q_o'' \\ T_s \end{bmatrix} = [C] \begin{bmatrix} T_1 \\ \vdots \\ T_n \end{bmatrix} + [D] \begin{bmatrix} T_i \\ T_o \\ q_{source} \end{bmatrix} \quad (522)$$

where T_s is the temperature of the node where the heat source or sink is present. The transfer function equation for the calculation of T_s that results from Equation (522) is identical in form to Equation (504):

$$T_{s,t} = \sum_{m=1}^M x_{k,m} T_{i,t-m+1} - \sum_{m=1}^M y_{k,m} T_{o,t-m+1} + \sum_{m=1}^k f_m T_{s,t-m} + \sum_{m=1}^M w_m q_{source,t-m+1} \quad (523)$$

Instead of the flux at either side of the element characterized as a function of temperature, flux, and source history terms, the temperature at the source location is related to source and temperature histories including histories of T_s . The validity of these internal temperature calculation methods as well as heat source transfer functions in general will be discussed in the next chapter.

Low Temperature Radiant System Controls

The use of this equation allows the low temperature radiant system to be handled like any other surface within the heat balance framework. Heat balances at the inside and outside surfaces take on the same form as other surfaces, and the participation of the radiant system in the radiation balance within the space and thermal comfort models is automatically included. Thus, the radiant system model is fully integrated into the heat balance, and any improvements that are made in areas such as convection coefficients, shading models, etc. are immediately available to the radiant system as part of the overall heat balance solution.

Once the transient nature of the system is accounted for, one must then turn to the next difficult issue: controls. Controls are problematic for almost any simulation program. The problem is not whether something can be simulated because typically a simulation program offers the ability to experiment with many different control strategies. Rather, the problem is typically the diversity of controls that are implemented and keeping the controls that can be simulated up to date. EnergyPlus offers two different control schemes: variable flow (ZoneHVAC:LowTemperatureRadiant:VariableFlow) and variable temperature (ZoneHVAC:LowTemperatureRadiant:ConstantFlow). The control strategies are different enough that they were developed as separate system types. More details of the controls are described below.

The controls for variable flow low temperature radiant systems within EnergyPlus are fairly simple though there is some flexibility through the use of schedules. The program user is allowed to define a setpoint temperature as well as a throttling range through which the system varies the flow rate of water (or current) to the system from zero to the user defined maximum flow rate. The flow rate is varied linearly with the flow reaching 50% of the maximum when the controlling temperature reaches the setpoint temperature. Setpoint temperatures can be varied on an hourly basis throughout the year if desired. The controlling temperature can be the mean air temperature, the mean radiant temperature, or the operative temperature of the zone, and this choice is also left to the user's discretion. (Operative temperature for radiant system controls is the average of MAT and MRT.) Since flow rate is varied, there is no explicit control on the inlet water temperature or mixing to achieve some inlet water temperature in a hydronic system. However, the user does have the ability to specify on an hourly basis through a schedule the temperature of the water that would be supplied to the radiant system.

Graphical descriptions of the controls for the low temperature radiant system model in EnergyPlus are shown in Figure 207 for a hydronic system. In a system that uses electric resistance heating, the power or heat addition to the system varies in a manner similar to mass flow rate variation shown in Figure 207.

In the constant flow-variable temperature systems, the controls are also considered piecewise linear functions, but in this case the user selects both the control temperatures and the water temperatures via schedules. This offers greater flexibility for defining how the radiant system operates though it may not model every situation. Figure 208 shows how the "desired" inlet water temperature is controlled based on user schedules. The user has the ability to specify the high and low water and control temperature schedules for heating and cooling (separately; a total of eight temperature schedules). Note that this inlet temperature is a "desired" inlet temperature in that there is no guarantee that the system will provide water to the system at that temperature. The model includes a local loop that attempts to meet this demand temperature through mixing and recirculation.

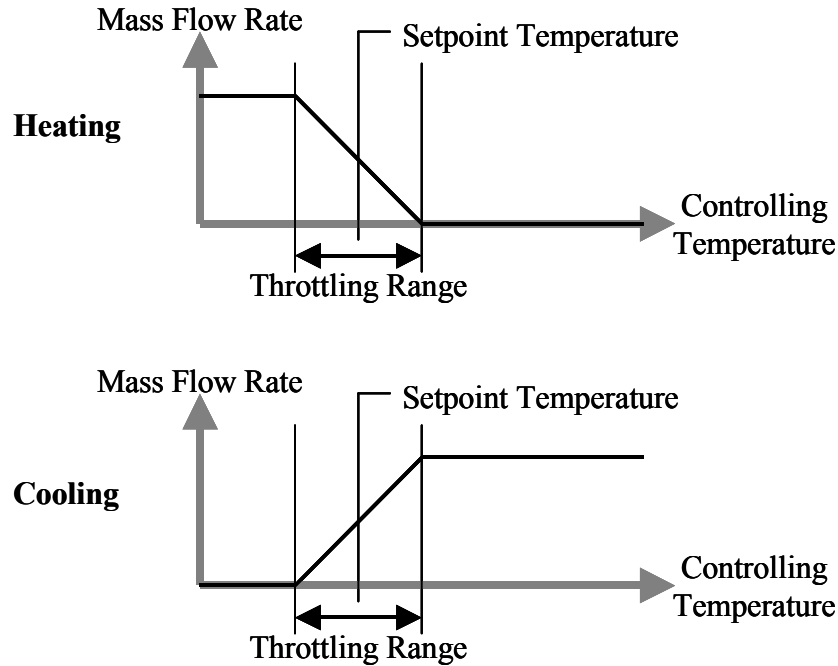


Figure 207. Variable Flow Low Temperature Radiant System Controls

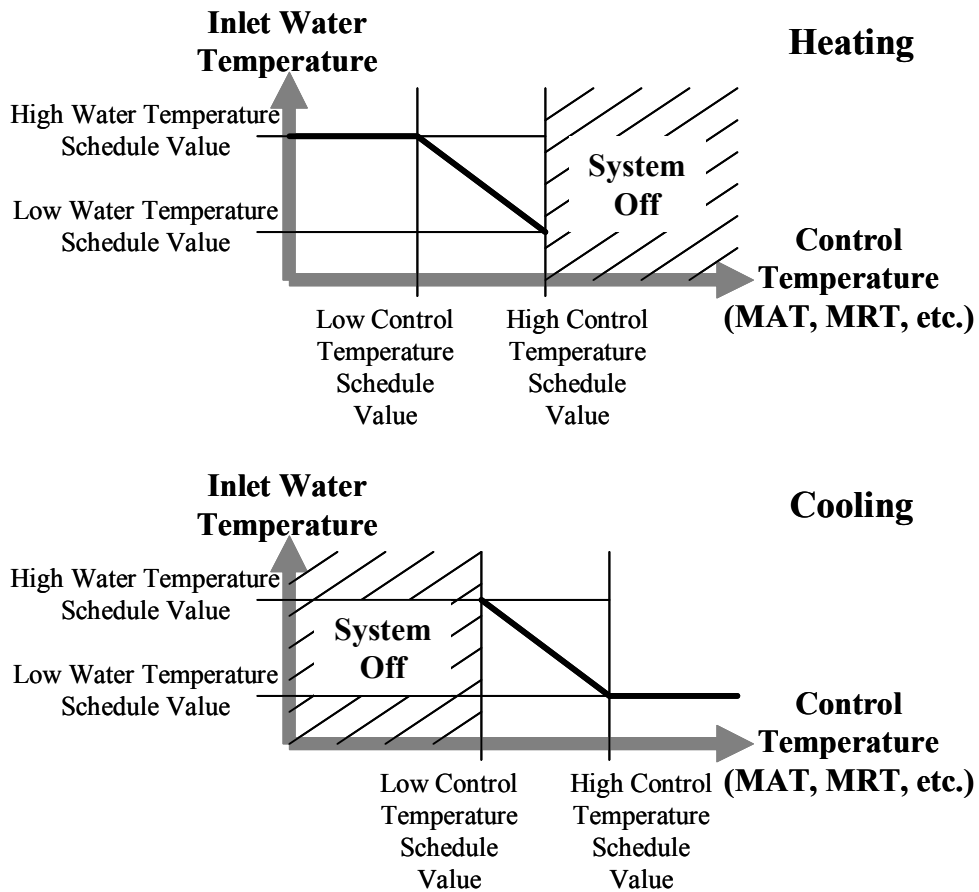


Figure 208. Variable Temperature Low Temperature Radiant System Controls

The constant flow (variable temperature) low temperature radiant system model is actually a combination of mixing valves, a pump (constant speed, but the maximum flow can be modified by a schedule), and the radiant system (surface, panel, or group of surfaces/panels). This is connected to the main loop through the standard inlet connections as shown in Figure 209. The system controls determine the desired inlet temperature and system flow rate while loop controls determine the flow rate and temperature of the loop. Note that pump heat also factors into the model through a simple constant speed pump model and user input.

There are four possible conditions (separate for heating and cooling). First, if the loop has adequate temperature and flow to meet system requests, then the model sets the radiant system inlet temperature and controls to the desired values based on the controls and simulates. This is the best condition and recirculation and bypass amounts are adjusted accordingly based on radiant system outlet temperatures. Second, if the loop temperature is adequate but the loop flow rate is less than the radiant system flow rate, we may or may not be able to meet the desired inlet temperature since recirculation might lower the temperature below the desired temperature. In this second case, the model first simulates the radiant system with the desired conditions and then resimulates it to solve for the actual inlet temperature (see later in this section) if it cannot achieve the desired inlet temperature. Third, if the loop flow is greater than the radiant flow but the temperature of the loop is not adequate, then there is no amount of mixing that will solve this problem. All of the radiant flow comes from the loop and the loop temperature (after pump heat addition) becomes the radiant system inlet regardless of the temperature controls. Finally, if both the temperature and the flow of the loop are inadequate, then the model simply solves for the actual radiant system inlet temperature and does not try to meet the controls (merely tries to get as close as physically possible given the loop conditions).

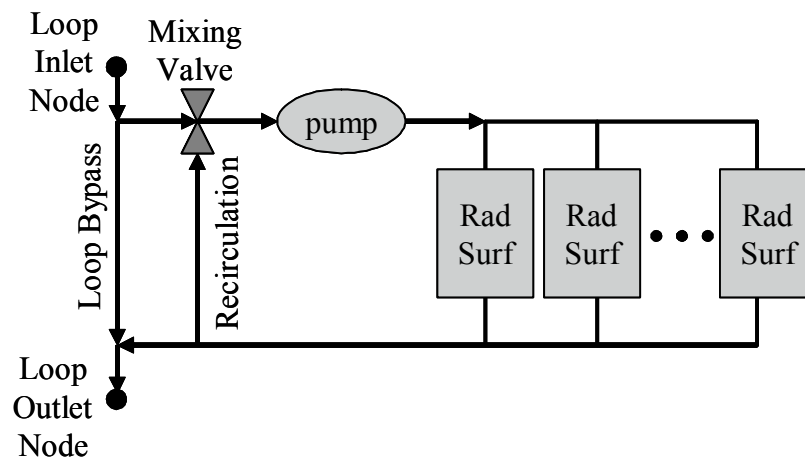


Figure 209. Variable Temperature Low Temperature Radiant System Component Details

One remaining challenge is the merging of the low temperature radiant system model with an integrated building simulation program. In the past, most simulation programs have simulated the building envelope, the space conditioning systems, and the central plant equipment in three separate steps. While this had some advantages and was partly due to a lack of computing capacity, the large drawback for this arrangement is that there is no feedback from the space conditioning system or central plant response to the building conditions. Thus, if the system or plant was undersized, it was reported as an “unmet load” and does not affect the temperatures experienced within the building. IBLAST, a predecessor (Taylor 1991) to EnergyPlus, resolved this issue by integrating all three major components of a building simulation and thus allowing feedback between the equipment and the building envelope.

This integration was not a trivial task and required that the systems be simulated at shorter time steps in some cases to maintain solution stability. In essence, the system simulation will shorten its time step whenever it senses that conditions are changing too rapidly. While this is effective in maintaining solution stability, it can present problems for a radiant system. The radiant system has either a direct or an indirect impact on the surfaces within a building. So, it must be simulated with the building envelope. Yet, it is also a space conditioning system that must act on the space like any other system and thus must also be simulated at the system time step, which can be less than the building time step and can also vary within EnergyPlus.

This issue was handled using a multi-step approach. In EnergyPlus, the heat balance is always simulated first. When this happens, the radiant system is temporarily shut-off to find how the building would respond if there was no heat source/sink. Then, as the system and plant are simulated at multiple shorter time steps, the radiant system is allowed to operate per the controls specified by the user. Flow rate is allowed to vary at each system time step, and the radiant system model is simulated at each time step as if the current flow rate was being used throughout the entire zone time step. This means that each time the heat source/sink in the radiant system is varied during the system simulation the zone heat balance must be recomputed to see what the reaction of the rest of the zone is to this change in the conditions of one (or more) of the surfaces.

In reality, this is not physically correct because each change in the flow rate throughout the system simulation will have an impact on the system time steps remaining before the heat balance is simulated during the next zone time step. Yet, other approaches to solving the mismatch between the system and the zone response of radiant systems are not feasible. One could force the system to run at the same time step as the zone, but this could result in instabilities in other types of systems that might be present in the simulation. On the other hand, one could try to force the zone to run at the shorter time steps of the system, but this could lead to instability within the heat balance due to limits on the precision of the conduction transfer function coefficients.

Despite the fact that the simulation algorithm described above may either over- or under-predict system response dependent on how the system has been controlled in previous system time steps, it is reasonable to expect that the effect of these variations will balance out over time even though it might lead to slightly inaccurate results at any particular system time step. The long-term approach is also in view in the final simulation step at each zone time step. After the system has simulated through enough system time steps to equal a zone time step, the radiant system will rerun the heat balance using the average heat source/sink over all of the system time steps during the past zone time step. This maintains the conservation of energy within the heat balance simulation over the zone time steps and defines more appropriate temperature and flux histories at each surface that are critical to the success of a conduction transfer function based solution. A graphical picture of this somewhat complex multiple step simulation is shown in the figure below.

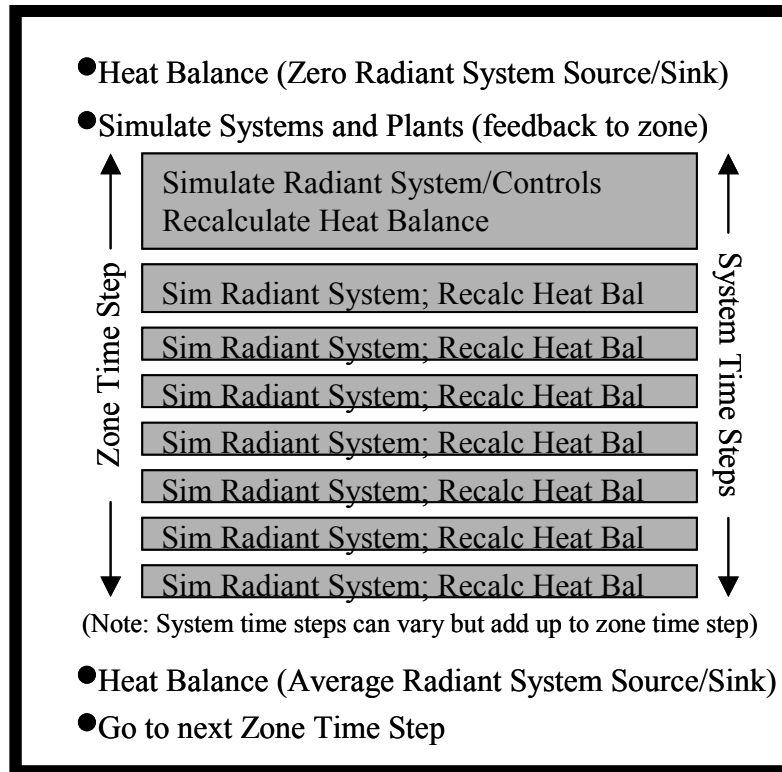


Figure 210. Resolution of Radiant System Response at Varying Time Steps

Heat Exchanger Formulation for Hydronic Systems

As has been mentioned previously, the actual heat transferred between the building element and the hydronic loop is related to the temperature of the building element at the source location as well as the water system inlet and outlet temperatures. In EnergyPlus, it is assumed that the inlet temperature to the slab (defined by a user schedule and the plant simulation) and the mass flow rate (determined by the control algorithm) are known and that the remaining parameters must be calculated. However, the heat balance equations require the heat transferred to the building element from the water loop in order to calculate the heat transferred from the element to the building environment.

Even though systems defined by this model can vary somewhat, the same characteristic link between the system variables exist. For modeling purposes, the overall water/slab system can be thought of as a heat exchanger. While in principle there are two alternative heat exchanger methodologies, it is more convenient to use the effectiveness-NTU method in this case.

Several assumptions will be incorporated into the heat exchanger analysis. It is assumed that the building element that contains the hydronic loop is stationary and that its temperature along the length of the tubing is constant. The latter part of this assumption stems from assumptions made in both the one and two dimensional heat source transfer function derivations. In either case, the source was added at a single node that was characterized by a single temperature. For consistency, this assumption must be made again in the heat exchanger analysis. Another assumption for the current EnergyPlus model is that the fluid in the tubing is water. Additionally, it is assumed that the thermal properties of the water do not vary significantly over the length of the tubing and that the water flows at a constant flow rate. Finally, the temperature at the inside surface of the water tubing is assumed to be equal to the temperature at the source location. In other words, it is assumed that the water tubing itself has no appreciable effect on the heat transfer process being modeled.

Using these assumptions and the effectiveness-NTU heat exchanger algorithm, several equations can be defined which establish the relationship between the heat source and the water temperatures. First, a heat balance on the water loop results in:

$$q = (\dot{m}c_p)_{water} (T_{wi} - T_{wo}) \quad (524)$$

where q is the energy transferred between the water loop and the building element, \dot{m} is the mass flow rate of the water, c_p is the specific heat of the water, T_{wi} is the inlet water temperature, and T_{wo} is the outlet water temperature.

The maximum amount of heat transfer that is possible according to the Second Law of Thermodynamics is:

$$q_{max} = (\dot{m}c_p)_{water} (T_{wi} - T_s) \quad (525)$$

where q_{max} is the maximum amount of energy transfer that is possible and T_s is the temperature at the source location.

The effectiveness of the heat exchanger, ε , is defined as the ratio of the actual energy transfer to the maximum possible, or:

$$\varepsilon \equiv \frac{q}{q_{max}} \quad (526)$$

For a heat exchanger where one fluid is stationary, the effectiveness can be related to NTU, the number of transfer units, by the following equation (Incropera and DeWitt 1985):

$$\varepsilon = 1 - e^{-NTU} \quad (527)$$

where NTU is defined by:

$$NTU \equiv \frac{UA}{(\dot{m}c_p)_{water}} \quad (528)$$

Since the water tubes were assumed to have no effect on the heat transfer process, the only term present in the overall heat transfer coefficient, UA , is a convection term. Thus, the equation for UA is:

$$UA = h(\pi DL) \quad (529)$$

where h is the convection coefficient, D is the interior tube diameter, and L is the total length of the tube.

The convection coefficient can be obtained from internal flow correlations that relate the Nusselt dimensionless number to other flow properties. For laminar flow in a tube of constant surface temperature, the Nusselt number is defined by:

$$Nu_D = \frac{hD}{k} = 3.66 \quad (530)$$

where k is the thermal conductivity of the water.

For turbulent internal flow, the Colburn equation can be used to define the Nusselt number:

$$Nu_D = \frac{hD}{k} = 0.023 Re_D^{4/5} Pr^{1/3} \quad (531)$$

where Pr is the Prandtl number of water and Re_D is the Reynolds number which is defined by:

$$Re_D = \frac{4\dot{m}}{\pi\mu D} \quad (532)$$

The parameter μ is the absolute viscosity of water. For internal pipe flow, the flow is assumed to be turbulent for $Re_D \geq 2300$.

Knowledge of the flow conditions allows Equations (526) through (532) to be calculated. This essentially eliminates ε as an unknown in Equation (525). The controls and the plant define the water mass flow rate and the inlet water temperature, leaving two equations (Equations (524) and (525)) and three unknowns. The third equation that can be used in conjunction with Equations (524) and (525) is Equation (523), which is the CTF/QTF equation for the temperature at the source location.

Knowing the inlet water temperature and water mass flow rate, the calculation procedure is somewhat involved and requires, in addition to Equations (523), (524), and (525), the use of a modified form of Equation (504). Equation (504) is the standard conduction transfer function formula for a building element with an embedded source/sink of heat. In EnergyPlus, the surface flux on the left hand side of the equation is replaced with a surface heat balance:

$$\left[\begin{array}{c} \text{Surface} \\ \text{Heat} \\ \text{Balance} \end{array} \right] = \sum_{m=1}^M X_{k,m} T_{1,t-m+1} - \sum_{m=1}^M Y_{k,m} T_{3,t-m+1} + \sum_{m=1}^k F_m q''_{1,t-m} + \sum_{m=1}^M W_m q_{source,t-m+1} \quad (533)$$

The surface heat balance includes terms for incident solar energy, radiation heat transfer from internal heat sources such as lights and electrical equipment, radiation between surfaces using Hottel's Gray Interchange concept, and convection to the surrounding air. The presence of the surface temperature in the heat balance does not pose any problems since Equation (533) will be rearranged to solve for this temperature. Since the radiation heat balance is dependent on conditions at the other surfaces, an iteration loop is required to provide a more accurate estimate of the radiative exchange within the building. This is not the case with the mean air temperature. An assumption of the heat balance is that the mean temperature of the surrounding air is equal to the final air temperature of the previous time step. Using this estimate in the heat balance avoids a second iterative loop around the radiative iteration loop.

Thus, the terms of the heat balance on the left hand side of the equation have been set with the only unknown quantity being T_i , the inside surface temperature at the current time step. On the right hand side of Equation (533), most of the terms are already defined since they depend on known values from previous time steps (temperature, flux, and source histories). The only terms which are not defined are the inside surface temperature (T_i), outside surface temperature (T_o), and internal heat source/sink (q_{source}) of the current time step.

The outside surface temperature will depend on the type of environment to which it is exposed. For example, if the surface is a slab on grade floor, the outside surface temperature is defined as ground temperature and does not require an outside surface heat

balance. If the element is an interior surface which has both surfaces exposed to the same air space, the outside surface temperature is redefined to be equal to the inside surface temperature. In cases where the outside surface temperature is not simply defined such as a surface exposed to the exterior environment, a heat balance similar to Equation (533) is required to define the outside surface temperature. However, to again avoid iteration, the heat balance equation for the outside surface assumes that conditions at the inside surface were the same as the previous time step. In most cases, since the influence of the current inside surface temperature on the outside surface temperature is very small, this is a valid assumption. In cases where the inside surface temperature has a significant effect, an approximate inside surface heat balance which defines the inside surface temperature is used. This approximate inside balance uses mean air and radiant temperatures from the previous time step.

At this point in the simulation algorithm then, all of the terms in Equation (533) have been defined except the value at the current time step of the inside surface temperature and the heat source/sink. Thus, Equation (533) can be rewritten in a simpler form:

$$T_{i,t} = C_1 + C_2 q_{source,t} \quad (534)$$

where the variable C_1 includes surface heat balance and past history terms as well as the influence of the current outside temperature. The term C_2 will depend on the heat source transfer function term and the coefficients of terms linked directly to $T_{i,t}$.

Equation (523), which was the CTF/QTF equation for the temperature at the source location, can be simplified in a similar manner. Grouping the temperature and source history terms which are known quantities together with the effect of the outside surface temperature which is defined as described above, the original equation

$$T_{s,t} = \sum_{m=1}^M x_{k,m} T_{i,t-m+1} - \sum_{m=1}^M y_{k,m} T_{o,t-m+1} + \sum_{m=1}^k f_m T_{s,t-m} + \sum_{m=1}^M w_m q_{source,t-m+1} \quad (523)$$

can be reduced to:

$$T_s = C_3 + C_4 q_{source,t} + C_5 T_{i,t} \quad (535)$$

where C_3 includes all of the history terms and the effect of the current outside temperature, C_4 is the heat source transfer function for the current time step, and C_5 is the conduction transfer function for the inside surface temperature at the current time step.

Substituting Equation (534) into Equation (535) and noting that $q_{source,t}$ is the same quantity as q in Equations (524) and (525) results in:

$$T_s = C_3 + C_4 q + C_5 (C_1 + C_2 q) \quad (536)$$

When this equation is combined with Equation (525), the heat source, which results from a known water inlet temperature, can be shown to be:

$$q = \frac{T_{wi} - C_3 - C_1 C_5}{\frac{1}{\varepsilon (\dot{m} c_p)_{water}} + C_4 + C_2 C_5} \quad (537)$$

With both q and T_{wi} known, it is a trivial matter to calculate T_{wo} and T_s from Equations (534) and (535), respectively. Even though the coefficients in Equation (537) are fairly complex, the final equation relating the heat source directly to inlet water temperature is compact and does not require any iteration. As with flux control, once the heat source/sink is defined, the inside surface heat balance can be performed to determine the surface temperatures.

It should be noted that Equations (534) through (537) are a slight simplification of the actual implementation in EnergyPlus. The development shown above follows the heat balance conventions that assume previous values of the inside temperature to calculate the outside temperature. This, in reality, is not necessary and since the radiant system can be significantly influenced by the delay that such an assumption might cause, the initial implementation of radiant systems in EnergyPlus used a development (shown below) that does not lag either the inside or the outside surface temperature. In effect, we can establish three basic equations for the temperature at the inside and outside surface as well as at the location of the heat source/sink:

$$T_{inside} = C_a + C_b T_{outside} + C_c q'' \quad (538)$$

$$T_{outside} = C_d + C_e T_{inside} + C_f q'' \quad (539)$$

$$T_{source} = C_g + C_h q'' + C_i T_{inside} + C_j T_{outside} \quad (540)$$

where: T_{inside} is the temperature at the inside surface

$T_{outside}$ is the temperature at the outside surface

T_{source} is the temperature within the radiant system at the location of the source/sink

C_a is all of the other terms in the inside heat balance (solar, LW exchange, conduction history terms, etc.)

C_b is the current cross CTF term

C_c is the QTF inside term for the current heat source/sink

C_d is all of the other terms in the outside heat balance (solar, LW exchange, conduction history terms, etc.)

C_e is the current cross CTF term (should be equal to C_b)

C_f is the QTF outside term for the current heat source/sink

C_g is the summation of all temperature and source history terms at the source/sink location

C_h is the QTF term at the source/sink location for the current heat source/sink

C_i is the CTF inside term for the current inside surface temperature

C_j is the CTF outside term for the current outside surface temperature

Equations (538) and (539) above can be solved to remove the other surface temperature. Substituting the new equations for T_{inside} and $T_{outside}$ as a function of C and q'' into the equation for T_{source} and simplifying results in the following equation:

$$T_{source} = C_k + C_l q'' \quad (541)$$

$$\text{where: } C_k = C_g + \frac{C_i(C_a + C_b C_d) + C_j(C_d + C_e C_a)}{1 - C_e C_b}$$

$$C_l = C_h + \frac{C_i(C_c + C_b C_f) + C_j(C_f + C_e C_c)}{1 - C_e C_b}$$

Combining this with heat exchanger analysis as shown above, we eventually arrive at the following equation to relate the flux to the slab to the water inlet temperature and mass flow rate:

$$q'' = \frac{T_{water,in} - C_k}{\frac{C_l}{A} + \frac{1}{\varepsilon(\dot{m}c_p)_{water}}} \quad (542)$$

which includes all of the inside and outside heat balance terms (“hidden” in the C_k and C_l coefficients). Once the flux to the slab is known, the remaining terms of interest (outlet water temperature, inside and outside surface temperatures, etc.) can be calculated using the relatively simpler equations shown above.

Note that the above development is valid for both the hydronic (variable flow) low temperature radiant system and for constant flow (variable temperature) low temperature radiant systems where the inlet temperature is known (based on controls). However, when due to loop conditions and the presence of recirculation, it is not possible to know the inlet temperature to the radiant system without simulating it, we must either iterate or perform more mathematics to arrive at the inlet temperature. The implementation in EnergyPlus chose to avoid iteration and solved for the inlet temperature as shown in the next paragraphs.

The previous equation combines with the following equation which is valid for an surface in the current radiant system:

$$q_j = \dot{m}_j c_p (T_{water,in} - T_{water,out,j})$$

where q_j is the heat transfer to the j th surface in the radiant system, \dot{m}_j is the mass flow rate only to this surface, and $T_{water,out,j}$ is the outlet temperature for the j th surface. Combining the previous two equations results in:

$$(T_{water,in} - T_{water,out,j}) = \frac{\varepsilon_j (T_{water,in} - C_{k,j})}{1 + \frac{\varepsilon_j \dot{m}_j c_p C_{l,j}}{A}}$$

If for each surface in the radiant system, we let:

$$C_{m,j} = \frac{\varepsilon_j}{1 + \frac{\varepsilon_j \dot{m}_j c_p C_{l,j}}{A}}$$

then the previous equations become the slightly less complex:

$$(T_{water,in} - T_{water,out,j}) = C_{m,j} (T_{water,in} - C_{k,j})$$

Rearranging to obtain the outlet temperature for the j th surface:

$$T_{water,out,j} = (1 - C_{m,j}) T_{water,in} + C_{m,j} C_{k,j}$$

The overall outlet temperature from the radiant system is just a simple mixing of all of the surface outlet temperatures based on flow fraction and results in $T_{water,out}$.

An energy balance on the mixing valve-pump group results in the following equation that relates the radiant system inlet temperature ($T_{\text{water,in}}$) to the loop inlet water temperature ($T_{\text{loop,in}}$), the radiant system outlet temperature ($T_{\text{water,out}}$), and the pump heat addition:

$$T_{\text{water,in}} = \frac{\dot{m}_{\text{loop}}}{\dot{m}_{\text{system}}} T_{\text{loop,in}} + \frac{\dot{m}_{\text{recirc}}}{\dot{m}_{\text{system}}} T_{\text{water,out}} + \frac{\text{PumpHeat}}{\dot{m}_{\text{system}} c_p}$$

Plugging in the definition of $T_{\text{water,out}}$ based on the summation of $T_{\text{water,out},j}$ equations results in:

$$T_{\text{water,in}} = \frac{\text{LoopTerm} + \text{RecircTerm}}{\text{TwicCoeff}}$$

where:

$$\text{LoopTerm} = \frac{\dot{m}_{\text{loop}}}{\dot{m}_{\text{system}}} T_{\text{loop,in}} + \frac{\text{PumpHeat}}{\dot{m}_{\text{system}} c_p}$$

$$\text{RecircTerm} = \frac{\dot{m}_{\text{recirc}}}{\dot{m}_{\text{system}}} \sum_j \text{FlowFraction}_j T_{\text{water,out},j}$$

$$\text{TwicCoeff} = \left(1 - \frac{\dot{m}_{\text{recirc}}}{\dot{m}_{\text{system}}} \right) \sum_j \left[\text{FlowFraction}_j (1 - C_{m,j}) \right]$$

Once the actual water inlet temperature is calculated with this equation, it is then possible to calculate individual outlet temperatures for each surface, the overall outlet temperature, and finally all of the necessary flow and loop quantities. This procedure avoids iteration but is somewhat complex to follow. However, this second mathematical process is only needed for select cases of the constant flow radiant system when the inlet temperature is not known explicitly. With the proper establishment of input data, it can be avoided.

NOTE: In some “low-energy” applications, it is possible that during cooling mode the elevated water temperatures may result in a heat source (or net heating) to the radiant system. To avoid heating the slab when cooling is desired, EnergyPlus performs a temperature limit check. If heating would result during cooling mode or cooling during heating mode, the model will cut-off the flow rate until the inlet water temperature will produce the proper effect.

High Temperature Radiant Heater Model

The input object ZoneHVAC:HighTemperatureRadiant provides a model for a high temperature radiant heater that is intended to encapsulate an entire class of heating devices that seek to heat the occupants within a zone by direct radiation. This encompasses a wide variety of heaters including both gas-fired and electric. In most cases, the heater appears much like a lamp or a tube that is suspended from the ceiling of a space, and the surface temperatures are high enough that the heaters must be a safe distance away from the occupied portion of the space for safety concerns.

In EnergyPlus, the high temperature radiant heater model allows the user a reasonable amount of flexibility. Rather than specifying an exact location for the radiant heater(s), the user is allowed to specify the percentage of heat leaving the heater as radiation and then on which surfaces this radiation is incident. In addition, the user is also allowed the ability to define what fraction of radiation leaving the heater is incident directly on a person within the zone for thermal comfort purposes. This amount of heat is then used in the thermal comfort

models as shown in Equations (586) for Fanger, (606) for Pierce Two-Node and (635) for KSU Two-node. These equations are similar in form to the equation promoted by Fanger (1970). The input parameters for the high temperature radiant heater model are shown in Table 1.

$$T_{\text{radiant}} = \left[\left(T_{\text{MRT}}^4 \right) + \left(\frac{Q_{\text{heater} \rightarrow \text{person}}}{\sigma A_{\text{person}}} \right) \right]^{0.25} \quad (543)$$

```

HIGH TEMP RADIANT SYSTEM, ! Program keyword for high temp. radiant heaters
Zone 1 Radiant Heater, ! Zone name
Radiant Operation, ! Availability schedule
SHOP ZONE, ! Zone name (name of zone system is serving)
10000, ! maximum power input (in Watts)
GAS, ! type of heater (either gas or electric)
0.85, ! combustion efficiency (ignored for elec. heaters)
0.75, ! fraction radiant
0.05, ! fraction latent
0.05, ! fraction lost
OPERATIVE, ! temperature control type (MAT, MRT also possible)
2.0, ! heating throttling range (in C)
Heating Setpoints, ! schedule of heating setpoint temperatures
0.05, ! fraction of radiant energy to people
Zn001:Flr001, 0.75, ! surface/fraction of radiant energy incident on it
Zn001:Wall001, 0.05, ! surface/fraction of radiant energy incident on it
Zn001:Wall002, 0.05, ! surface/fraction of radiant energy incident on it
Zn001:Wall003, 0.05, ! surface/fraction of radiant energy incident on it
Zn001:Wall004, 0.05; ! surface/fraction of radiant energy incident on it

```

Figure 211. Input Description for High Temperature Radiant Heaters

The input for the high temperature radiant heater has two additive relationships that are assumed. First, the fractions of radiant, convective, latent, and lost heat must sum to unity. The user is required to enter the fractions radiant, latent, and lost with the remainder assumed to be convective energy. The fraction latent is added to the latent energy balance and will affect moisture levels within the zone. The fraction lost is assumed to have no impact on the energy balance of the zone and is assumed to be lost or vented to the exterior environment.

The second additive relationship is within the distribution of the radiant fraction. This energy is distributed to people and to the surfaces within the zone. The sum of all of these distribution fractions (the last six lines of input shown in Figure 211) must sum to unity. Note that each high temperature radiant heater is allowed to distribute energy to up to 20 surfaces and that radiant energy placed on a surface using these distribution fractions is assumed to be completely absorbed. Thus, the distribution fractions should also take into account any differences in long wavelength absorptivity among the surfaces.

Several things should be noted about the fraction of heat that is radiated directly to people. This parameter is somewhat sensitive and will have a direct impact on the thermal comfort models. This is exactly the intent of the high temperature radiant heaters; however, one must use caution when determining this fraction since overestimation of this number might lead to predictions of thermal comfort where in fact it does not exist. In addition, this fraction of radiant energy to people does not have a direct impact on any of the surface heat balances. The thermal comfort energy balance is completely separate from and has no bearing on the zone air or the surface heat balances. Thus, in order to not “lose” this amount of energy from the perspective of the zone air or the surface heat balances, the model assumes that any radiation from the high temperature radiant heater that is incident directly on people is accounted for in the thermal comfort model using Equation (543) but is also assumed to be added to the zone air heat balance via convection from people to the surrounding air. This

guarantees that the people within the space feel the direct radiative effect of the heaters and that this quantity of energy is not “lost” within the heat balance routines.

Many of the control and integration aspects of the high temperature radiant system model in EnergyPlus are very similar to the low temperature radiant system model. The controls are the same as shown in “Figure 207. Variable Flow Low Temperature Radiant System Controls” where the amount of heat generated by the radiant heater varies as a function of the difference between the controlling and the setpoint temperatures. As with the low temperature radiant system, the controlling temperature is allowed to be the mean air, the mean radiant, or the operative temperature, and the setpoint temperature is allowed to vary hourly based on a user defined schedule. (Operative temperature for radiant system controls is the average of MAT and MRT.) Also, since the high temperature radiant heater has a direct impact on the surfaces within a zone, the surface heat balances are recalculated to determine an approximate response to the radiation from the heater. A final “average” heat balance calculation is done after all of the system time steps have been simulated to maintain continuity within the surface heat balances. The algorithm shown in “Figure 210. Resolution of Radiant System Response at Varying Time Steps” is also used for high temperature radiant heaters.

References

- Ceylan, H.T. and G.E. Myers. 1980. Long-time solutions to heat conduction transients with time-dependent inputs. ASME Journal of Heat Transfer, Volume 102, Number 1, pp. 115-120.
- Degiovanni, A. 1988. Conduction dans un “mur” multicouche avec sources: extension de la notion de quadripole. International Journal of Heat and Mass Transfer, Volume 31, Number 3, pp. 553-557.
- Fanger, P.O. 1970. Thermal Comfort-Analysis and Applications in Environmental Engineering, Danish Technical Press, Copenhagen.
- Hittle, D.C. 1981. Calculating building heating and cooling loads using the frequency response of multilayered slabs. Ph.D. Thesis, Department of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign and Technical Manuscript E-169, United States Army Construction Engineering Research Laboratory, Champaign, IL.
- Hittle, D.C. and R. Bishop. 1983. An improved root-finding procedure for use in calculating transient heat flow through multilayered slabs. International Journal of Heat and Mass Transfer, Volume 26, Number 11, pp. 1685-1693.
- Hottel, H.C. and A.F. Sarofim. 1967. Radiative Transfer, McGraw-Hill, New York.
- Incropera, F.P. and D.P. DeWitt. 1985. Introduction to Heat Transfer. New York: John Wiley & Sons.
- Lee, J. and R.K. Strand. 2001. “An Analysis of the Effect of the Building Envelope on Thermal Comfort using the EnergyPlus Program”, submitted for publication in the proceedings of the 2001 ACSA (Association of Collegiate Schools of Architecture) Technology Conference, Austin, TX.
- Liesen, R.J. and C.O. Pedersen. 1997. “An Evaluation of Inside Surface Heat Balance Models for Cooling Load Calculations”, ASHRAE Transactions, Volume 103, Part 2.
- Maloney, D. 1987. “Development of a radiant heater model and the incorporation of thermal comfort considerations into the BLAST energy analysis program”, M.S. thesis, University of Illinois at Urbana-Champaign, Department of Mechanical and Industrial Engineering.
- McClellan, T.M. and C.O. Pedersen. 1997. “Investigation of Outside Heat Balance Models for Use in a Heat Balance Cooling Load Calculation Procedure”, ASHRAE Transactions, Volume 103, Part 2.
- Pedersen, C.O., D.E. Fisher, and R.J. Liesen. 1997. “Development of a Heat Balance Procedure for Cooling Loads”, ASHRAE Transactions, Volume 103, Part 2.

Pedersen, C.O., D.E. Fisher, J.D. Spitler, and R.J. Liesen. 1998. Cooling and Heating Load Calculation Principles, ASHRAE.

Seem, J.E. 1987. Modeling of heat transfer in buildings. Ph.D. Thesis, University of Wisconsin-Madison.

Strand, R.K., and C.O. Pedersen. 1994. "Analytical verification of heat source transfer functions", First Joint Conference of International Simulation Societies, Zürich, Switzerland.

Strand, R.K. 1995. "Heat source transfer functions and their application to low temperature radiant heating systems", Ph.D. dissertation, University of Illinois at Urbana-Champaign, Department of Mechanical and Industrial Engineering.

Strand, R.K. and C.O. Pedersen. 1997. "Implementation of a Radiant Heating and Cooling Model into an Integrated Building Energy Analysis Program", ASHRAE Transactions, Volume 103, Part 1.

Strand, R.K. and C.O. Pedersen. 2001. "Modularization and Simulation Techniques for Heat Balance Based Energy and Load Calculation Programs: the Experience of the ASHRAE LOADS Toolkit and EnergyPlus", International Building Performance Simulation Association, Conference Proceedings of Building Simulation 2001, Rio de Janeiro, Brazil.

Taylor, R.D., C.O. Pedersen, D. Fisher, R. Liesen, and L. Lawrie. 1991. "Impact of simultaneous simulation of building and mechanical systems in heat balance based energy analysis programs on system response and control", International Building Performance Simulation Association, Conference Proceedings of Building Simulation 1991, Nice, France.

Refrigeration Equipment

Overview

EnergyPlus can model refrigerated case equipment consisting of a compressor rack, up to twenty refrigerated cases connected to the compressor rack (extensible by changing the Input Data Dictionary [Energy+.idd] appropriately), and optional heat reclaim air and water heating coils. The refrigerated case equipment models perform four major functions:

- calculate the electric consumption of refrigerated cases connected to a compressor rack
- determine the impact of refrigerated cases on zone cooling and dehumidification loads (i.e., case credits), including the effects of HVAC duct configuration
- calculate the electric consumption and COP of the compressor rack, and the electric and water (if applicable) consumption related to cooling the compressor rack's condenser.
- determine the total amount of heat rejected by the compressor rack's condenser and store this information for use by waste heat recovery models (e.g., using Desuperheater heating coil (object: Coil:Heating:Desuperheater) as an air reheat coil for high humidity control in a supermarket)

The models account for nearly all performance aspects of typical refrigerated case equipment. Refrigerated case performance is based on the combined effects of evaporator load, fan operation, lighting, defrost type, and anti-sweat heater operation. Optional air and water heating coils can be modeled which reclaim available waste heat (superheat) from the compressor rack. The user has two options when describing the balance of the system. A compressor rack object combines the compressors and condenser into a single unit with the performance determined by the heat rejection environment and the total case load. A detailed refrigeration system object requires the use of separate compressor and condenser objects and allows the optional inclusion of subcoolers which can be used to transfer refrigeration load from one system to another. Energy used to cool the condenser is simulated in both approaches. An example schematic of a compressor rack system is shown in Figure 212 below. An example schematic of the detailed refrigeration system is shown in Figure 213 below. Subcooler #2 is shown twice on Figure 213 because it represents a liquid suction heat exchanger. This type of subcooler uses the cool suction gas to subcool the warmer condensed liquid. Subcoolers #1 and #3 on Figure 213 represent mechanical subcoolers.

These subcoolers are used to subcool the condensate on a lower-temperature system using the cold liquid refrigerant from a higher temperature system. On this example, only subcoolers #1 and #2 would be defined as a part of the refrigeration system. However, subcooler #3 would place a refrigerating load, similar to the load of a refrigerated case, on the system.

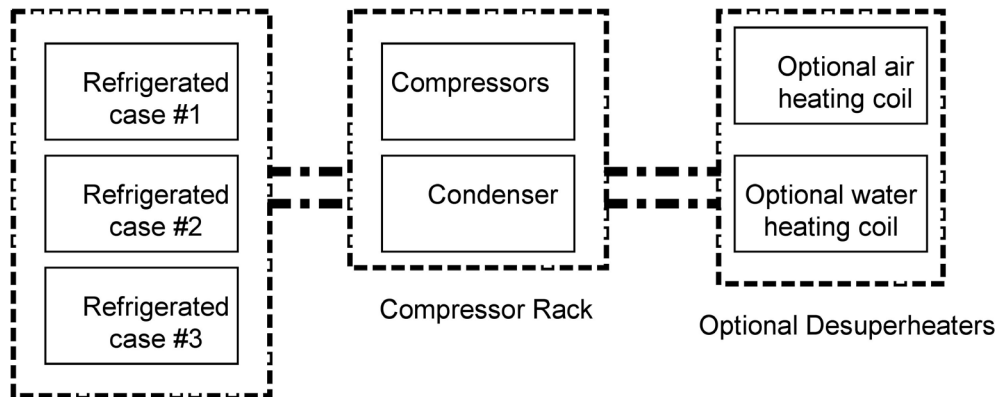


Figure 212. Typical Refrigerated Case and Compressor Rack Equipment Schematic

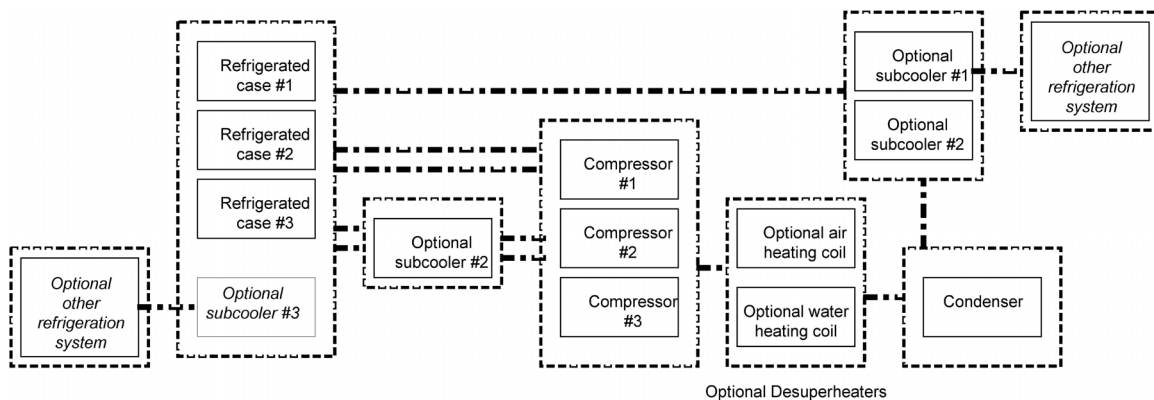


Figure 213. Typical Refrigerated Case and Detailed Refrigeration System Equipment Schematic

The compressor rack, detailed refrigeration system, and refrigerated case models are described below. The optional air and water heating coils are described elsewhere in this document (Ref. objects Coil:Heating:Desuperheater and Coil:WaterHeating:Desuperheater).

Refrigeration Compressor Racks

The refrigerated case compressor rack object works in conjunction with the refrigerated case object (Refrigeration:Case) to simulate the performance of a refrigerated case system. This object (Refrigeration:CompressorRack) models the electric consumption of the rack compressors and the cooling of the compressor rack's condenser. Heat removed from the refrigerated cases and compressor/condenser fan heat can be rejected either outdoors or to a zone. Compressor rack condenser waste heat can also be reclaimed for use by an optional air heating coil (Ref. object Coil:Heating:Desuperheater) or by a user-defined plant water loop (Ref. object Coil:WaterHeating:Desuperheater).

The performance of the compressor rack is simulated using the sum of the evaporator loads for all refrigerated cases connected to the rack. Whether a single refrigerated case is connected to a rack (e.g., stand-alone refrigerated case, meat cooler, or produce cooler) or several cases are connected to a rack, the rack electric consumption is calculated based on the total evaporator load for the connected cases and the coefficient of performance (COP) for the compressor rack. At least one refrigerated case must be connected to the compressor rack. The model assumes the compressor rack has sufficient capacity to meet the connected

refrigerated case load for any simulation time step. Additionally, the model neglects compressor cycling losses at part-load conditions.

For condenser heat rejection to the outdoors, condenser cooling can be modeled as dry air cooling, wet evaporative cooling, or water loop cooling. Using evaporative cooling rather than dry air cooling will allow for more efficient condenser heat rejection based on the entering air approaching the wet-bulb temperature rather than the dry-bulb temperature. Analyses under the International Energy Agency's (IEA) Heat Pumping Programme Annex 26 indicates that this measure can improve refrigeration system efficiency by up to 10% (IEA 2003). The use of an evaporative-cooled condenser requires a water pump and, optionally, a basin sump water heater (to protect against freezing). Makeup water will also be required to replace that lost by evaporation. In colder climates, some evaporative-cooled condensers are drained for the winter months and run as dry air units. This scenario can be modeled by using an optional evaporative condenser availability schedule.

The simulation of the evaporative cooled condenser utilizes an effective air dry-bulb temperature that is assumed to be the result of evaporation of water in the air stream (similar to object `EvaporativeCooler:Direct:CelDekPad`). As discussed below, this effective temperature is used by performance curves that are a function of temperature. While some designs of evaporative coolers use water film cascading across the condenser coil for evaporative cooling, the current model uses the effective temperature method as a surrogate for the more complex water film on coil calculations.

If the condenser heat rejection is specified as water cooled, an appropriate plant water loop must be defined by the user (see documentation on *Plant/Condenser Loops* for additional details about plant loops). This will include defining cooling supply components, such as pumps, water storage tanks, and cooling towers, as well as related branches, nodes, and connectors. The heat rejection from the refrigeration condenser is modeled as a cooling demand, which is satisfied by heat extraction devices (e.g., water tank and cooling tower) on the cooling supply side of a water loop. An example of such an arrangement is shown in Figure 214.

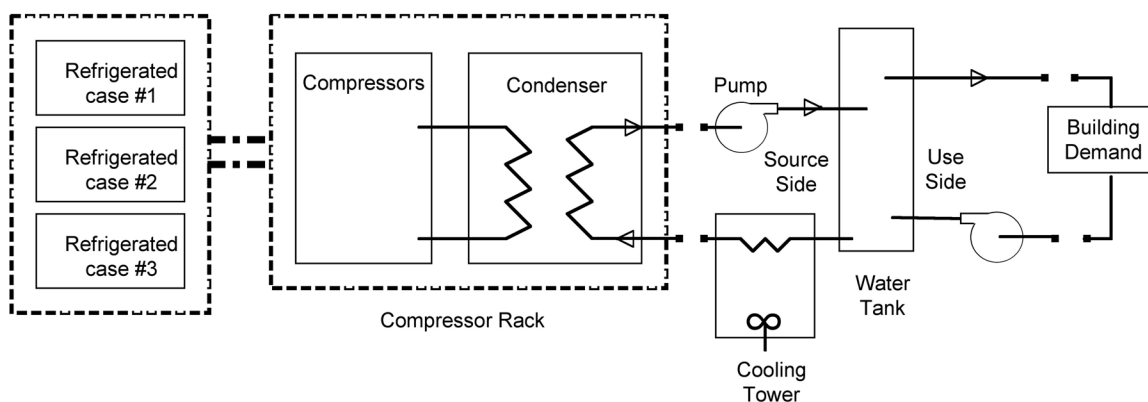


Figure 214. Example Of Condenser Heat Recovery To Water Storage Tank

Compressor Rack Energy Use

Calculation of compressor rack electric power uses a simple model based on the total evaporator load (sum of the evaporator loads for all refrigerated cases connected to a rack) and the compressor rack operating COP which accounts for the air temperature entering the condenser:

$$COP_{operating} = COP_{design} (COP_{fTemp})$$

where:

$COP_{operating}$ = compressor coefficient of performance at actual operating conditions (W/W)

COP_{design} = compressor coefficient of performance at design conditions (W/W)

$COPfTemp$ = output of the normalized “Compressor Rack COP as a Function of Temperature Curve” (dimensionless, range from 0 to 1)

The air temperature used to evaluate the “Compressor Rack COP as a Function of Temperature Curve” depends on where the condenser rack’s condenser is located (Heat Rejection Location). When modeling condenser heat rejected directly to a zone (typical of a stand-alone packaged refrigerated case with integral condenser located in a building zone), the zone air dry-bulb temperature is used to calculate the change in compressor COP from the design value. If more than one refrigerated case is attached to a compressor rack that rejects its condenser heat to a zone, then all cases served by this rack must reside in the same zone. When modeling a compressor rack with condenser heat rejected to outdoors, the refrigerated cases connected to this rack may be located in different zones. If the condenser type is specified as “Air Cooled”, the outdoor air dry-bulb temperature is used to evaluate the “Compressor Rack COP as a Function of Temperature Curve.” If the condenser type is specified as “Evap Cooled”, the air temperature leaving the condenser is related to the effectiveness of the evaporative cooling system. If the evaporative process were 100% effective, the effective temperature of air leaving the evaporative media would equal the air wet-bulb temperature. However, the efficiency of the direct evaporative process is typically less than 100%, and the effective temperature leaving the condenser is determined by:

$$T_{effective} = T_{owb} + (1 - \varepsilon) * [T_{odb} - T_{owb}]$$

where:

$T_{effective}$ = effective dry-bulb temperature of air leaving the condenser cooling coil (°C)

T_{owb} = outdoor air wet-bulb temperature (°C)

T_{odb} = outdoor air dry-bulb temperature (°C)

ε = evaporative condenser effectiveness.

If the user is modeling an evaporative cooled condenser and is using COPfTemp curve data (e.g., manufacturer’s data) based on wet-bulb temperature rather than dry-bulb temperature, the evaporative condenser effectiveness should be set to 1.0 for consistency.

If the condenser is water cooled, the effective temperature experienced by the condenser is based on the return water temperature from the plant loop heat rejection system (e.g., cooling tower) that is defined by the user. This return water temperature is typically related to the outdoor ambient conditions at each time step.

The electric power input to the rack compressor(s) is calculated for each simulation time step as the sum of the connected refrigerated case evaporator loads divided by the operating COP:

$$P_{rack} = \frac{\sum \dot{Q}_{case}}{COP_{operating}}$$

where:

P_{rack} = output variable “Refrigerated Case Compressor Rack Electric Power [W]”, electric power input to the rack compressor(s)

\dot{Q}_{case} = evaporator load for each refrigerated case connected to the rack (W)

Condenser Fan Energy Use

Condenser fan power for any simulation time step is calculated by multiplying the design fan power by the condenser fan power as a function of temperature curve.

$$P_{CondFan} = P_{CondFan,design} (CondFanfTemp)$$

where:

$P_{CondFan}$ = output variable “Refrigerated Case Condenser Fan Electric Power [W]”

$P_{CondFan,design}$ = design condenser fan power (W)

$CondFanfTemp$ = output of the optional “Condenser Fan Power as a Function of Temperature Curve”

Similar to the compressor rack energy use described above, the air temperature used to evaluate the “Condenser Fan Power as a Function of Temperature Curve” depends on where the condenser rack’s condenser is located (i.e., zone air dry-bulb temperature if the condenser is located in a zone, outdoor air dry-bulb temperature if the condenser is located outdoors and is specified as air cooled, or effective temperature if the condenser is outdoors and is specified as evaporative cooled). If the sum of the evaporator loads for the refrigerated cases connected to the rack is equal to zero, the condenser fan power is set equal to zero. If the user does not provide a “Condenser Fan Power as a Function of Temperature Curve”, then the model assumes the condenser fan power is at the design power level when any of the refrigerated cases connected to this rack are operating.

If the user is modeling an evaporative cooled condenser and is using CondFanfTemp curve data based on wet-bulb temperature rather than dry-bulb temperature, the evaporative condenser effectiveness should be set to 1.0 for consistency.

For a water cooled condenser, there is no fan load at the condenser (i.e., the water/refrigerant heat exchanger). Any fan load would be related to and accounted for at the heat rejection object (e.g., cooling tower).

Superheat Reclaim Heating Coil

EnergyPlus can simulate waste heat being reclaimed from a compressor rack for use by a refrigerant-to-air or refrigerant to water heating coil. Heat reclaimed from the compressor rack is assumed to be recovered from the superheated refrigerant gas leaving the compressor(s) and does not directly impact the performance of the compressor rack or refrigerated cases connected to the rack. The total heat rejected by the condenser (in Watts) is calculated each time step as follows:

$$\dot{Q}_{condenser} = \sum \dot{Q}_{case} \left(1 + \frac{1}{COP_{operating}} \right)$$

The heat reclaim heating coil is able to transfer a fixed percentage of this total amount of rejected energy (not to exceed 30%) and use it to heat air and water. Refer to objects Coil:Heating:Desuperheater and Coil:WaterHeating:Desuperheater for a complete description of how these coils are modeled.

NOTE: When modeling a heat reclaim coil, the heat rejection location in the Refrigeration:CompressorRack object must be “Outdoors”. If the compressor rack heat rejection location is “Zone”, the total amount of waste heat available for reclaim (e.g., by a desuperheater heating coil) is set to zero by the compressor rack object and the simulation proceeds.

Heat Rejection to Zone

The compressor rack model can simulate condenser heat being rejected to a zone. As explained previously, if this heat rejection option is selected then all refrigerated cases connected to the rack must be located in the same zone and a superheat reclaim heating coil can not be modeled (Ref. Superheat Reclaim Heating Coil). The refrigerated case object (Refrigeration:Case) already calculates and reports the sensible case credits which impact the zone air heat balance (Ref. Sensible Case Credits). When refrigerated cases are served by a compressor rack that rejects condenser waste heat directly to the zone (e.g., a stand-alone refrigerated case with integral compressor and condenser), this condenser waste heat also impacts the zone air heat balance and offsets some or all of the sensible case credits. The amount of condenser waste heat rejected to the zone and/or the HVAC return air (zone return air path outlet node) is calculated and reported by the refrigerated case compressor rack object as follows:

$$\dot{Q}_{Zone,heating} = \frac{\sum(\dot{Q}_{case}[1-RAF])}{\sum(\dot{Q}_{case})}(\dot{Q}_{condenser} + P_{CondFan})$$

$$\dot{Q}_{HVAC,heating} = (\dot{Q}_{condenser} + P_{CondFan}) - \dot{Q}_{zone,heating}$$

where:

$\dot{Q}_{Zone,heating}$ = output variable “Rack Sensible Heating Rate To Zone [W]”

RAF = return air factor for each case connected to the rack (Ref. Figure 215)

$\dot{Q}_{HVAC,heating}$ = output variable “Rack Sensible Heating Rate To HVAC Return Air [W]”

If the HVAC system is off for a simulation time step (no return air mass flow), the rack condenser heat normally attributed to the HVAC return is set equal to zero and all condenser heat energy is applied to the zone air heat balance.

Water Cooled Condenser

If the refrigeration condenser is water cooled, a water plant loop must be defined in the input file. At a minimum, the loop must contain a pump and one or more heat sinks of sufficient capacity to remove the condenser heat load. In the system shown in Figure 214, the heat sinks are the water heater tank and the cooling tower. The water pump in the loop can be either constant (Ref. Pump:ConstantSpeed) or variable speed (Ref. Pump:VariableSpeed). A variable speed pump permits the loop flow to vary and allows for a setpoint to be established on the condenser outlet water temperature. As the refrigeration condenser heat load varies through time, the speed of the pump can be adjusted to achieve a mass flow consistent with a desired outlet water temperature according to

$$m = \frac{Q_{condenser}}{c_p \cdot (T_{out} - T_{in})}$$

where:

m = mass flow in the water loop

$Q_{condenser}$ = heat rejected by the condenser

c_p = specific heat of water

T_{out} = desired water outlet temperature

T_{in} = return water inlet temperature.

The desired water outlet temperature is specified using a schedule, subject to a maximum water outlet temperature (input specified). The maximum temperature is typically defined by constraints on the refrigerant loop pressures and temperatures. The desired mass flow in the water loop to meet the temperature schedule is also compared to the user-supplied maximum flow rate. If the desired mass flow is greater than the maximum allowed flow, the flow rate is set to the maximum value and the resulting water outlet temperature is determined.

The return water inlet temperature is a function of the cooling system defined by the user. A minimum return water temperature may need to be taken into consideration to prevent lowering the resulting refrigerant condensing pressure to the point that refrigerant expansion valve operation becomes impaired. When ambient conditions produce low temperature warnings based on the minimum return water temperature, an outlet temperature setpoint control may need to be placed on the water heat sink object (e.g., cooling tower) to keep the return water temperature above the minimum.

If the water loop flow is constant (i.e., driven by a constant speed pump), then the outlet water temperature will vary with the amount of heat rejected by the condenser. Using the equation above, the resulting water outlet temperature is calculated as

$$T_{out} = \frac{Q_{condenser}}{c_p \cdot m} + T_{in}$$

Evaporative Condenser Water Pump

If the condenser type is specified as “Evap Cooled”, a water pump is required to circulate water in the evaporative condenser. The pump power can be input directly or be autocalculated using a relationship of 0.004266 W per watt [15 W/ton] of rated total cooling capacity where the total cooling capacity is the sum of the rated total cooling capacities for the refrigerated cases connected to this compressor rack (Ref. Refrigeration:Case). Following manufacturer’s recommendations regarding the avoidance of scaling, the water pump does not cycle when there is no cooling demand (i.e., when the compressors are not running), but rather runs continuously. However, if the evaporative condenser availability schedule is set such that evaporative cooling is not available (e.g., during very cold months to avoid freezing), then the pump power consumption will be zero during that period.

Evaporative Condenser Water Consumption

With evaporative cooling of the condenser’s entering air, makeup water is needed to replenish the water lost due to evaporation. The quantity required is calculated as the product of the air mass flow rate and the difference between the entering and leaving air humidity ratio, divided by the density of water. The air mass flow rate is determined by multiplying the evaporative condenser air volume flow rate times the density of the entering air (i.e., at the condenser air inlet node if provided, or outdoor air conditions [e.g., no adjustment for height above ground] if the condenser air inlet node field is left blank). The volumetric air flow rate is either specified directly in the user input or is autocalculated using the relationship 0.000144 m³/s per watt of rated total cooling capacity [850 cfm/ton] where the total cooling capacity is the sum of the rated total cooling capacities for the refrigerated cases connected to this compressor rack (Ref. Refrigeration:Case). The air mass flow rate is multiplied by the variable *CondFanfTemp*, described above, to simulate the modulation of air flow by the condenser fans (e.g., staging, multi-speed, or variable speed) as a function of temperature. Mathematically,

$$\dot{V}_{evaporation,makeup} = \frac{\dot{m}_{air} (CondFanfTemp) (\omega_{air,outlet} - \omega_{air,inlet})}{\rho_{water}}$$

where:

$\dot{V}_{evaporation,makeup}$ = Refrigeration Compressor Rack Evap Condenser Water Consumption Rate (m^3/s)

\dot{m}_{air} = mass flow rate of air through the evaporative condenser (kg/s)

$\omega_{air,outlet}$ = humidity ratio of air leaving the evaporative media ($kg_{water}/kg_{dry\ air}$) based on the effective dry-bulb temperature $T_{effective}$, as described above, outdoor air wet-bulb temperature, and outdoor barometric pressure

$\omega_{air,inlet}$ = humidity ratio of inlet air ($kg_{water}/kg_{dry\ air}$) based on conditions at the condenser air inlet node if provided, or outdoor air conditions (e.g., no adjustment for height above ground) if the condenser air inlet node field is left blank

ρ_{water} = density of water evaluated at the effective air temperature (kg/m^3)

The source of the makeup water may be specified as a water storage tank. If not specified, the makeup water is assumed to come from the building mains (Ref. Water Mains Temperatures).

Evaporative Condenser Basin Heater

In cold climates, a basin heater may be needed to prevent freezing of the evaporative cooling water. This feature is included in the model whereby an electric basin heater provides heat to the sump water only when the condenser cooling system is idle (i.e., no refrigeration load) and when the outdoor air dry-bulb temperature is below a user-specified setpoint. Since heat balances and basin water temperatures are not explicitly determined, a linear loading relationship, as a function of the difference in outdoor air dry-bulb temperature and the setpoint temperature, is used calculate the power demand at a given time step by the basin heater.

$$P_{basin\ heater} = P_{heater\ capacity} * (T_{setpoint} - T_{OutDb})$$

where:

$P_{basin\ heater}$ = electric power demand for basin heater in current time step (W)

$P_{heater\ capacity}$ = electric heater capacity as a function of differential temperature (W/deg K)

$T_{setpoint}$ = setpoint temperature below which the heater turns on ($^{\circ}C$)

T_{OutDb} = outdoor air dry-bulb temperature ($^{\circ}C$)

A default value for the basin heater capacity of 200 W/deg K has been established based on manufacturer data.

Evaporative Condenser Availability Schedule

Some manufacturer's evaporative cooling systems for refrigeration condensers permit seasonal draining in the colder months and operation as an air-cooled system during that time. This optional feature is available through an availability schedule. This is important in climates subject to freezing weather in order to avoid excessive ice formation on the condenser surfaces and surroundings. (The Availability Schedule is the correct way to model the use of evaporative condensers in cold climates. However, some users may take a single input description and use it to model a building with a refrigeration system in a variety of climates. To avoid modeling the use of evaporative coolers in freezing weather, the code includes a cutout to switch to dry operation whenever the outdoor drybulb temperature drops below 4C.) During periods when evaporative cooling is not available, the outdoor condenser behaves as an air-cooled system with no water consumption or pump and basin heater loads.

The effective temperature of air entering the condenser coil during this period (used to evaluate COPfTemp and CondFanfTemp) is equal to the outdoor air dry-bulb temperature at the condenser air inlet node if provided, or outdoor air conditions (e.g., no adjustment for height above ground) if the condenser air inlet node field is left blank.

Refrigerated Cases

The refrigerated case object (Refrigeration:Case) works in conjunction with the compressor rack object (Refrigeration:CompressorRack) or detailed refrigeration object (Refrigeration:System) to simulate the performance of a refrigerated case system. The refrigerated case model uses performance information at rated conditions along with performance curves for latent case credits and defrost heat load to determine performance at off-rated conditions. Energy use for lights, fans and anti-sweat heaters is modeled based on inputs for nominal power, schedules, and control type. The refrigerated case model accounts for the sensible and latent heat exchange with the surrounding environment (termed "case credits") which impacts the temperature and humidity in the zone where the case is located. The simplified model described here provides the flexibility to simulate a broad range of refrigerated case types.

Model Description

The total load on the refrigerated case evaporator is made up of various components:

$$\dot{Q}_{case} = \dot{Q}_{walls} + \dot{Q}_{rad} + \dot{Q}_{inf,sens} + \dot{Q}_{inf,lat} + \dot{Q}_{lights} + \dot{Q}_{as} + \dot{Q}_{def} + \dot{Q}_{fan} + \dot{Q}_{restock} \quad (544)$$

where:

\dot{Q}_{case} = total load on the refrigerated case evaporator (W)

\dot{Q}_{walls} = heat transfer through case walls due to the difference between the refrigerated case operating dry-bulb temperature and the zone air dry-bulb temperature (W)

\dot{Q}_{rad} = radiant heat transfer to the refrigerated case (W)

$\dot{Q}_{inf,sens}$ = sensible heat transfer by air infiltration to the refrigerated case through the air curtain or via door openings (W)

$\dot{Q}_{inf,lat}$ = latent heat transfer by air infiltration to the refrigerated case through the air curtain or via door openings (W)

\dot{Q}_{lights} = lighting heat load (W)

\dot{Q}_{as} = anti-sweat heater load (W)

\dot{Q}_{def} = defrost heat load (W)

\dot{Q}_{fan} = fan heat load (W)

$\dot{Q}_{restock}$ = sensible load on the refrigerated case due to restocking of products that are at a higher temperature than the case (W)

The model assumes that these load components are known for a refrigerated case at rated ambient air conditions (typically 23.9°C [75°F] and 55% relative humidity) and the specified case operating temperature. A combination of user input curves and fixed correlations (defined within EnergyPlus) adjust for case performance at off-rated conditions. Several of the load components are typically provided by the case manufacturer (e.g., fan, lighting, anti-sweat heater, and defrost loads). The remaining load components are not usually provided by

the manufacturer and must be estimated (heat conduction through case walls, radiation heat transfer, sensible/latent air infiltration, and restocking).

For estimating the latent air infiltration load, the model requires that the user provide the latent heat ratio (LHR) for the refrigerated case at rated conditions. Research results are available to provide guidance in selecting this value (ASHRAE 2002, Howell 1993a, Howell 1993b). The rated LHR for refrigerated cases typically ranges from 0.1 to 0.3 depending on case configuration (e.g., glass door reach-in versus multi-deck open case) and case operating temperature.

The case loads due to wall heat conduction, radiation, and sensible air infiltration are estimated by the model as a single lumped value (sensible case credits). The sensible case credits are calculated by subtracting the known loads at rated conditions (fan, lighting, anti-sweat heater, defrost and latent case credits) from the rated total cooling capacity of the case which is provided by the case manufacturer ($\dot{Q}_{case,rated}$).

Using these assumptions and the schedule inputs provided by the user, the refrigerated case evaporator load components in Equation (544) are determined for each simulation time step. The variation in certain loads with respect to changes in ambient air temperature and/or humidity (e.g., latent and sensible case credits, defrost load, and anti-sweat heater load) are factored into the calculation based on user-provided inputs or by the model itself. The specific calculations for case evaporator load components and electric power for these loads (as applicable) are provided below.

Case Evaporator Fan

The refrigerated case evaporator fan electric power is calculated for each simulation time step as the product of the operating case fan power per unit length of case, the length of the refrigerated case, and the fraction of time that the case is not being defrosted. For cases with hot-gas or electric defrost (with or without temperature termination), the fan is disabled during the entire scheduled defrost drip-down time period. The evaporator fan operates continuously for off-cycle defrost or no defrost.

$$P_{fan} = P'_{fan,oper} (L_{case}) (1 - SCH_{defrost,dripdown})$$

where:

P_{fan} = output variable “Refrigerated Case Evaporator Fan Electric Power [W]”

$P'_{fan,oper}$ = operating case fan power per unit length (W/m)

L_{case} = case length (m)

$SCH_{defrost,dripdown}$ = fraction of time case is being defrosted (0 to 1), including drip-down period (based on the defrost drip-down schedule) for hot-gas or electric defrost. For off-cycle defrost or no defrost, this value is set to zero for this calculation.

The model assumes that the evaporator fan is entirely contained within the thermal envelope of the case, and that all fan power results in a direct heat load on the case evaporator:

$$\dot{Q}_{fan} = P_{fan}$$

Case Lighting

The refrigerated case lighting electric power is calculated for each simulation time step as the product of the standard case lighting power per unit length of case, the lighting schedule value, and the length of the refrigerated case:

$$P_{lights} = P'_{lights, std} (L_{case}) (SCH_{lights})$$

where:

P_{lights} = output variable “Refrigerated Case Lighting Electric Power [W]”

$P'_{lights, std}$ = standard case lighting power per unit length (W/m)

SCH_{lights} = case lighting schedule value (0 to 1)

A maximum schedule value of 1.0 means the lights are fully on at the standard case lighting power level. If the user wants to model high-efficiency lighting, the maximum lighting schedule value(s) should be the ratio of high-efficiency to standard-efficiency lighting power (e.g., 70W/m vs 100 W/m = 0.7).

The user can specify the fraction of lighting energy that directly contributes to the case evaporator heat load:

$$\dot{Q}_{lights} = P_{lights} (F_l)$$

where:

F_l = fraction of lighting energy to case

The remainder of the lighting energy ($1 - F_l$) is a heating load to the zone where the case is located, which is discussed further in section Sensible Case Credits below. This fraction ($1 - F_l$) can be used to represent lighting ballasts and/or bulbs located outside the air curtain of the refrigerated case.

Anti-Sweat Heater Performance

Anti-sweat heaters warm the refrigerated case rails or doors to provide protection from moisture condensation. Different anti-sweat heater control strategies are used depending on the case temperature and the type of anti-sweat heater installed. Several types of anti-sweat heater control strategies can be simulated with this model: constant, linear variation with ambient relative humidity or dewpoint temperature, and a theoretical model that determines the minimum anti-sweat heater power required to maintain the case surface just above the temperature where condensation would occur. Additionally, anti-sweat heater performance can be disregarded if the type of refrigerated case does not warrant its use. For the control strategies described below (except “None” and “Constant Method”), the model does not allow the anti-sweat heater power to be less than the minimum power nor greater than the case anti-sweat heater power specified by the user. Each anti-sweat heater control type is described in detail below.

None

Used for refrigerated cases that do not require an anti-sweat heater.

$$\dot{Q}_{as} = 0$$

where:

\dot{Q}_{as} = anti-sweat heater load on the case evaporator (W)

Constant Method

For refrigerated cases requiring constant anti-sweat heater output, the power use is simply calculated as the case anti-sweat heater power per unit length multiplied by the length of the case. This method is used when the manufacturer recommends that cycling of the heaters not occur.

$$P_{as} = P'_{as} (L_{case})$$

where:

P_{as} = output variable “Refrigerated Case Anti-Sweat Heater Electric Power [W]”

P'_{as} = case anti-sweat heater power per unit length (W)

Relative Humidity Method

Anti-sweat heater power can be reduced at lower ambient relative humidity levels to save energy while still protecting from moisture condensation on cold surfaces. For this control type, anti-sweat heater power use is reduced linearly based on case anti-sweat heater power at the rated ambient relative humidity (typically 55% RH), the relative humidity specified by the user where no anti-sweat heater power is required, and the relative humidity of the ambient (zone) air surrounding the case.

$$P_{as} = P'_{as} (L_{case}) \left(1 - \left[\frac{RH_{rated} - RH_{air}}{RH_{rated} - RH_{min}} \right] \right)$$

where:

RH_{air} = relative humidity of the ambient (zone) air (%)

RH_{rated} = rated ambient relative humidity (%)

RH_{min} = relative humidity at zero anti-sweat heater energy (%)

Dewpoint Method

Anti-sweat heater power can also be reduced as a function of ambient air dewpoint temperature based on a similar correlation to that used by the relative humidity method. This control method varies the anti-sweat heater power linearly based on the ambient air dewpoint temperature, the case operating temperature, and the rated ambient dewpoint temperature (calculated by the model using the rated ambient temperature and rated ambient relative humidity entered by the user).

$$P_{as} = P'_{as} (L_{case}) \left(\frac{T_{dp,air} - T_{case}}{T_{dp,rated} - T_{case}} \right)$$

where:

$T_{dp,air}$ = dewpoint temperature of the ambient (zone) air (°C)

$T_{dp,rated}$ = rated ambient dewpoint temperature (°C)

T_{case} = case operating temperature (°C)

Heat Balance Method

A theoretical model may also be used to simulate the performance of anti-sweat heater operation at various indoor dewpoint temperatures (Henderson and Khattar 1999). The model calculates that amount of heat required to hold the case or door surface at (or slightly above) the dewpoint temperature of the ambient air using the following simple heat balance equation:

$$P_{as} = \left(\frac{(T_{dp,air} - T_{db,air})H_{case}}{R_{air}} + \frac{(T_{dp,air} - T_{case})H_{case}}{R_{case}} \right) L_{case}$$

where:

$T_{dp,air}$ = dewpoint temperature of the ambient (zone) air (°C)

$T_{db,air}$ = dry-bulb temperature of the ambient (zone) air (°C)

H_{case} = height of the case (m)

R_{air} = air film resistance (assumed constant at 0.3169 m²-°C/W)

R_{case} = heat transfer resistance of case (m²-°C/W)

T_{case} = case operating temperature (°C)

L_{case} = case length (m)

The model above provides a linear relationship of anti-sweat heater power with varying ambient air dewpoint temperature at constant ambient air dry-bulb and case temperatures. By assuming that the 'nominal' anti-sweat heater power entered by the user is required to avoid moisture condensation at rated ambient air conditions, the value of R_{case} can be determined by rearranging the equation and solving as follows:

$$R_{case} = \frac{(T_{dp,rated} - T_{case})}{\left(\frac{P'_{as}}{H_{case}} \right) - \left(\frac{T_{dp,rated} - T_{db,rated}}{R_{air}} \right)}$$

where:

$T_{db,rated}$ = rated ambient temperature (°C)

With R_{case} known, P_{as} can be calculated for each simulation time step using the actual ambient (zone) air dry-bulb and dewpoint temperatures.

All Anti-Sweat Heater Control Methods

For all control methods, the user can specify the fraction of anti-sweat heater energy that directly contributes to the case evaporator heat load:

$$\dot{Q}_{as} = P_{as} (F_{as})$$

where:

F_{as} = fraction of anti-sweat heater energy to case

The remainder of the anti-sweat heater energy (1 - F_{as}) is a heating load to the zone where the case is located, which is discussed further in section Sensible Case Credits below.

Case Restocking

The impact of restocking the refrigerated case with product that is not at the case operating temperature is modeled with the case restocking schedule. The schedule is entered as a heat gain rate per unit length of the refrigerated case (W/m). The heat load due to restocking is calculated as the scheduled load multiplied by the length of the refrigerated case. The load

due to product restocking is assumed to be only sensible (temperature) heat; a latent (moisture) component is not modeled.

$$\dot{Q}_{restock} = SCH_{restock} (L_{case})$$

where:

$SCH_{restock}$ = refrigerated case restocking schedule value (W/m)

The restocking heat load is removed by the refrigerated case evaporator any time the case is not being defrosted and excess sensible cooling capacity is available. If the evaporator cooling capacity is insufficient to remove the entire restocking load, the unmet portion is carried over to the next simulation time step.

Case Defrost

Six refrigerated case defrost strategies can be simulated: none, off-cycle, electric, electric with temperature termination, hot-gas, and hot-gas with temperature termination. For each of these strategies, the refrigerated case evaporator is turned off for the required time period to allow accumulated frost to melt. Additional time can be scheduled (drip-down) to allow the water to drip from the evaporator and drain from the case.

Refrigerated cases typically require a specific number of defrost cycles per day for a pre-determined length of time. Refer to manufacturer's recommendations for proper defrost frequency and duration. For example, a refrigerated case may have a single defrost period each day with defrost scheduled from 7:00 – 7:40 am and defrost drip-down scheduled from 7:00 – 7:55 am. Notice the drip-down schedule and the defrost schedule start at the same time, and the drip-down schedule is longer than the defrost schedule. These schedules should normally repeat for each day of the year.

For electric defrost types, electrical energy use by the defrost heater occurs during the scheduled defrost period. For electric defrost with temperature termination, the electrical energy is also multiplied by the defrost ratio simulating a defrost duration shorter than the defined (maximum) period. For all other defrost types, defrost electric power is set equal to zero (and is not available as an output variable).

If (DefrostType = Electric) Then

$$P_{def} = P'_{def} (L_{case}) (SCH_{defrost})$$

ElseIf (DefrostType = ElectricWithTempTermination) Then

$$P_{def} = P'_{def} (L_{case}) (SCH_{defrost}) (DefrostRatio)$$

Else

$$P_{def} = 0.0$$

EndIf

where:

P_{def} = output variable "Refrigerated Case Defrost Electric Power [W]"

P'_{def} = case defrost power per unit length (W)

L_{case} = case length (m)

$SCH_{defrost}$ = case defrost schedule value (0 to 1)

DefrostRatio = fraction of maximum defrost time, used with temperature termination

Frost accumulation on the case evaporator will vary with the humidity level in the ambient air surrounding the case. Therefore, defrost heater operation can be reduced when ambient air humidity levels are low. Several methods are used to reduce unnecessary defrost heater operation, including terminating heater operation when the measured evaporator temperature indicates that the accumulated frost has been completely melted. For modeling refrigerated cases with temperature-terminated defrost, EnergyPlus allows the user to specify a defrost energy correction curve to account for variations in defrost energy as ambient air humidity levels change. The user can select from four correction curve types: None, Case Temperature Method, Relative Humidity Method, or Dewpoint Method.

None (default):

$$DefrostRatio = 1$$

Case Temperature Method:

$$DefrostRatio = 1 - (RH_{rated} - RH_{air}) \left[a + b(T_{case}) + c(T_{case})^2 + d(T_{case})^3 \right]$$

RH method:

$$DefrostRatio = e + f(RH_{air}) + g(RH_{air})^2 + h(RH_{air})^3$$

Dewpoint method:

$$DefrostRatio = i + j(T_{dp,air}) + k(T_{dp,air})^2 + l(T_{dp,air})^3$$

where:

RH_{rated} = rated ambient relative humidity (%)

RH_{air} = relative humidity of the ambient (zone) air (%)

T_{case} = case operating temperature (°C)

$T_{dp,air}$ = dewpoint temperature of the ambient (zone) air (°C)

a...l = user-defined coefficients using a cubic curve object (Curve:Cubic)

The user specifies the defrost energy correction curve type and the name of the cubic curve object (Curve:Cubic) that defines the curve coefficients. Representative curve coefficients for curve type "Case Temperature Method" are provided in Table 47.

Table 47. Representative Defrost Energy Correction Curve Coefficients for Case Temperature Method

Coefficient	Single-shelf horizontal display case	Multi-shelf vertical display case
a	2.3632E-2	2.4598E-2
b	6.2320E-4	7.6439E-4
c	2.8320E-5	-3.8637E-5
d	4.4035E-7	7.45686E-7

Note: Coefficients derived for $RH_{rated} = 55\%$ and a rated ambient temperature of 23.9°C (75°F). Source: Howell 1993b.

As mentioned above, the refrigerated case evaporator is turned off while it is being defrosted. Heat gains during defrost must be removed once the defrost period (drip-down schedule) has

ended. The model assumes that heat gains due to defrost heater operation are at least partially offset by converting accumulated frost to liquid water (condensate) which drains from the case. Frost accumulation during each simulation time step is estimated by the model using the actual latent heat transfer to the refrigerated case and the heat of vaporization plus the heat of fusion for water. The model assumes that frost is not accumulated on the evaporator during the defrost drip-down time period.

$$Frost = Frost + \left(\frac{\dot{Q}_{case, rated} (L_{case}) (RTF_{rated}) (LHR_{rated}) (LatentRatio) (t_{zn})}{(h_f + h_{fg})} \right) (1 - SCH_{defrost, dripdown})$$

where:

$Frost$ = amount of accumulated frost on the case evaporator (kg)

$\dot{Q}_{case, rated}$ = case rated total cooling capacity per unit length (W/m)

L_{case} = case length (m)

RTF_{rated} = runtime fraction of the refrigerated case at rated conditions

LHR_{rated} = latent heat ratio of the refrigerated case at rated conditions

$LatentRatio$ = ratio of actual latent load to rated latent load on the case, based on latent case credit curve (see section Latent Case Credits below)

t_{zn} = duration of zone simulation time step (s)

h_{fg} = heat of vaporization of water (assumed constant at 2,498,000 J/kg)

h_f = heat of fusion of water (335,000 J/kg)

$SCH_{defrost, dripdown}$ = defrost drip-down schedule value (0 to 1)

During defrost ($SCH_{defrost}$), the model assumes that the hot gas or electric heater energy directly contributes to melting the frost (heat of fusion of water). Defrost energy not attributed to melting frost from the evaporator coil results in a heat load on the refrigerated case

evaporator (\dot{Q}_{def}). When the defrost drip-down time period ends, this defrost energy heat load is added to the actual case load (up to the maximum evaporator capacity) until the total defrost energy heat load is removed (which may take several simulation time steps). For off-cycle or temperature-terminated defrost types, the model assumes that there is no defrost energy heat load.

If ($DefrostType = Electric$ or $HotGas$) *Then*

$$\dot{Q}_{def} = MAX \left(0.0, \left[P'_{def} (L_{case}) (SCH_{defrost}) - \frac{Frost (h_f)}{t_{zn}} \right] \right)$$

Else

$$\dot{Q}_{def} = 0.0$$

Endif

where:

\dot{Q}_{def} = defrost heat load (W)

Sensible Case Credits

Refrigerated cases remove sensible energy from the surrounding environment (termed “sensible case credits”). In this model, the sensible case credits are composed of wall heat conduction, radiation heat transfer, and sensible heat transfer by air infiltration ($\dot{Q}_{walls} + \dot{Q}_{rad} + \dot{Q}_{inf,sens}$ in equation (544)). To quantify this energy transfer, the model first calculates the rated sensible case credits by subtracting the known loads at rated conditions (fan, lighting, and anti-sweat heater) from the rated sensible cooling capacity of the case. It should be noted that the lighting and fan heat discussed here are for standard-efficiency equipment. Manufacturers typically provide ratings for both standard and high-efficiency fan and lighting equipment; however, the standard equipment is used to determine rated sensible case credits.

$$\dot{Q}_{cc\ sens, rated} = \left[\dot{Q}_{case, rated} (RTF_{rated}) (1 - LHR_{rated}) - P'_{lights, std} (F_l) - P'_{as} (F_{as}) - P'_{fan, std} \right] L_{case}$$

where:

$$\dot{Q}_{cc\ sens, rated} = \text{sensible case credits at rated conditions (W)}$$

$$\dot{Q}_{case, rated} = \text{case rated total cooling capacity per unit length (W/m)}$$

$$RTF_{rated} = \text{runtime fraction of the refrigerated case at rated conditions}$$

$$LHR_{rated} = \text{latent heat ratio of the refrigerated case at rated conditions}$$

$$P'_{lights, std} = \text{standard case lighting power per unit length (W/m)}$$

$$F_l = \text{fraction of lighting energy to case}$$

$$P'_{as} = \text{case anti-sweat heater power per unit length (W)}$$

$$F_{as} = \text{fraction of anti-sweat heater energy to case}$$

$$P'_{fan, std} = \text{standard case fan power per unit length (W/m)}$$

$$L_{case} = \text{case length (m)}$$

For every simulation time step, the rated sensible case credits are then adjusted to account for variations at off-rated ambient air temperatures. The model also allows the user to define a case credit fraction using a schedule object. This case credit fraction can be useful for modeling cases that operate differently during specific time periods. For example, metal or plastic coverings may be installed on refrigerated display cases during unoccupied hours which would significantly reduce case credits (e.g., air infiltration) compared to occupied hours when the coverings are removed. If the user does not define a case credit fraction schedule, then the fraction is assumed to be 1 for the entire simulation.

$$\dot{Q}_{cc\ sens} = \dot{Q}_{cc\ sens, rated} \left(\frac{T_{db, air} - T_{case}}{T_{db, rated} - T_{case}} \right) (SCH_{cc})$$

where:

$$\dot{Q}_{cc\ sens} = \text{sensible case credits adjusted for ambient temperature and case credit fraction (W)}$$

$$T_{db, air} = \text{dry-bulb temperature of the ambient (zone) air (°C)}$$

- T_{case} = case operating temperature (°C)
 $T_{db,rated}$ = rated ambient (zone) dry-bulb temperature (°C)
 SCH_{cc} = case credit fraction (schedule value, 0 to 1)

The sensible case credits calculated above are considered heat loads on the refrigerated case evaporator. The **net** impact of the case credits on the surrounding zone includes adjustment for the portion of the lighting and anti-sweat heater power that does not directly contribute to the case evaporator load. Sensible case credits are negative values when heat is removed from the zone load.

$$\dot{Q}_{cc_{sens,NET}} = P_{lights}(1 - F_l) + P_{as}(1 - F_{as}) - \dot{Q}_{cc_{sens}}$$

where:

- $\dot{Q}_{cc_{sens,NET}}$ = net impact of the sensible case credits on the surrounding zone, negative for cooling (W)
 P_{lights} = case lighting electric power (W)
 F_l = fraction of lighting energy to case
 P_{as} = anti-sweat heater electric power (W)
 F_{as} = fraction of anti-sweat heater energy to case

When refrigerated cases are served by a compressor rack that rejects condenser waste heat directly to the zone (e.g., a stand-alone refrigerated case with integral compressor and condenser), this condenser waste heat offsets some or all of the sensible case credits. The amount of condenser waste heat rejected to the zone is calculated and reported by the refrigerated case compressor rack object (Ref. Heat Rejection to Zone).

Latent Case Credits

Refrigerated cases also remove latent energy (moisture) from the surrounding environment (termed “latent case credits”). In this model, the latent case credit is composed solely of the latent heat transfer by air infiltration $\dot{Q}_{inf,lat}$ in equation (544). The latent case credits are calculated as the product of the case length and the total cooling capacity per unit length, latent heat ratio, and runtime fraction at rated conditions. As described previously (Ref. Sensible Case Credits), a case credit fraction schedule is used to model cases that operate differently during specific time periods. The same case credit fraction is used to modify both the sensible and latent case credits. If the user does not define a case credit fraction schedule, then the fraction is assumed to be 1 for the entire simulation. The calculation of latent case credits also includes a factor (*LatentRatio*) that accounts for lower ambient humidity levels.

$$\dot{Q}_{inf,lat} = -\dot{Q}_{cc_{lat}} = \dot{Q}_{case,rated} (LHR_{rated})(RTF_{rated})(SCH_{cc})(LatentRatio)L_{case}$$

where:

- $\dot{Q}_{inf,lat}$ = latent load on the refrigerated case evaporator at current ambient conditions (W)
 $\dot{Q}_{cc_{lat}}$ = latent case credit impact on zone load, negative for dehumidification (W)

$\dot{Q}_{case, rated}$ = case rated total cooling capacity per unit length (W/m)

LHR_{rated} = latent heat ratio of the refrigerated case at rated conditions

RTF_{rated} = runtime fraction of the refrigerated case at rated conditions

SCH_{CC} = case credit fraction (schedule value, 0 to 1)

$LatentRatio$ = ratio of actual latent load to rated latent load on the case, based on latent case credit curve

L_{case} = case length (m)

Latent load on the refrigerated case evaporator will vary with ambient humidity levels. Therefore, the refrigerated case model allows the user to specify a latent case credit curve to adjust case credits based on ambient humidity, and the user can select from three curve types: Case Temperature Method, Relative Humidity Method, or Dewpoint Method.

Case Temperature Method:

$$LatentRatio = 1 - (RH_{rated} - RH_{air}) \left[m + n(T_{case}) + o(T_{case})^2 + p(T_{case})^3 \right]$$

RH method:

$$LatentRatio = q + r(RH_{air}) + s(RH_{air})^2 + t(RH_{air})^3$$

Dewpoint method:

$$LatentRatio = u + v(T_{dp,air}) + w(T_{dp,air})^2 + x(T_{dp,air})^3$$

where:

RH_{rated} = rated ambient relative humidity (%)

RH_{air} = relative humidity of the ambient (zone) air (%)

T_{case} = case operating temperature (°C)

$T_{dp,air}$ = dewpoint temperature of the ambient (zone) air (°C)

m...x = user-defined coefficients using a cubic curve object (Curve:Cubic)

The user specifies the latent case credit curve type and the name of the cubic curve object (Curve:Cubic) that defines the curve coefficients. Representative curve coefficients for curve type "Case Temperature Method" are provided in Table 48.

Table 48. Representative Latent Case Credit Curve Coefficients for Case Temperature Method

Coefficient	Single-shelf horizontal	Multi-shelf vertical
m	2.0376E-2	2.6520E-2
n	2.4378E-4	1.0780E-3
o	1.1400E-5	-6.0256E-5
p	1.8110E-7	1.2373E-6

Note: Coefficients derived for $RH_{rated} = 55\%$ and a rated ambient temperature of 23.9°C (75°F). Source: Howell 1993b.

Refrigerated Case Credits With Under Case Return Air

For certain refrigerated case types, the sensible case credits provided to the zone can create an uncomfortably cold environment in the surrounding area. For this reason, return air ducts are frequently placed behind these cases to draw this cold air under the case and direct it back to the HVAC system. This reduces localized over-cooling and improves occupant comfort.

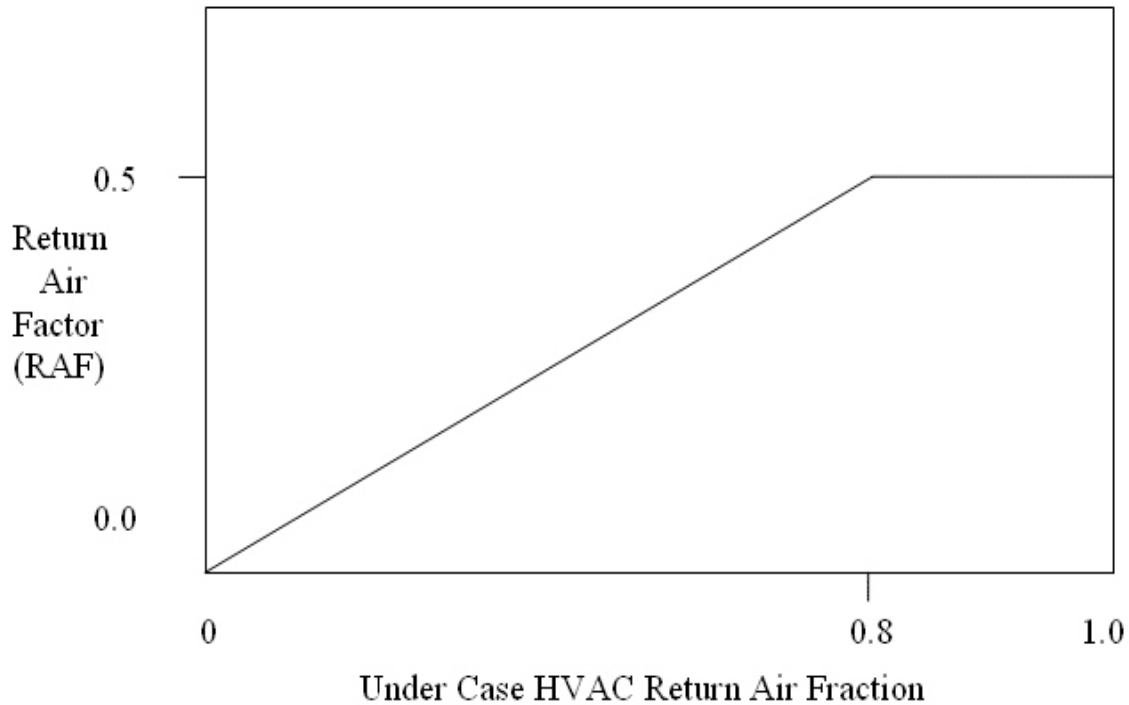


Figure 215. Return Air Factor Versus Under Case HVAC Return Air Fraction

Since under case return ducts reduce the temperature and humidity of the air being recirculated to the HVAC system, this can impact HVAC system performance. Figure 215 shows the relationship that is used by the refrigerated case model to determine the fraction of case credits that directly cool and dehumidify the HVAC system return air. This fraction, referred to as the Return Air Factor (RAF), is a function of the fraction of the HVAC system return air that comes from under the cases. The remaining fraction of the case credits (1-RAF) becomes part of the overall zone air energy balance. If the HVAC system is off for a simulation time step (no return air mass flow), the sensible and latent case credits normally attributed to the HVAC return are set equal to zero (even though they get calculated and reported here as non-zero values) and all case credit energy is applied to the zone air heat balance.

$$\dot{Q}cc_{sens,zone} = \dot{Q}cc_{sens,NET} (1 - RAF)$$

$$\dot{Q}cc_{lat,zone} = \dot{Q}cc_{lat} (1 - RAF)$$

$$\dot{Q}cc_{sens,HVAC} = \dot{Q}cc_{sens,NET} (RAF)$$

$$\dot{Q}cc_{lat,HVAC} = \dot{Q}cc_{lat} (RAF)$$

where:

$\dot{Q}_{cc_{sens,zone}}$ = sensible case credit applied to the zone air heat balance (W)

$\dot{Q}_{cc_{lat,zone}}$ = latent case credit applied to the zone air heat balance (W)

$\dot{Q}_{cc_{sens,HVAC}}$ = sensible case credit applied to the HVAC return air (zone return air path outlet node) (W)

$\dot{Q}_{cc_{lat,HVAC}}$ = latent case credit applied to the HVAC return air (zone return air path outlet node) (W)

RAF = return air factor (see Figure 215 above)

Variable Evaporator Temperature

Control systems are now available that increase the evaporator temperature to improve compressor efficiency whenever the total loads on a system are less than the system capacity. To model these systems, a variable evaporator temperature is an option available with the detailed refrigeration system object (Refrigeration:System). If this option is selected, the model will compare the case load on each case to the case load at rated conditions. If the case load in a particular time step is less than the rated load, an acceptable elevated evaporator temperature is determined for that case. The evaporator temperature for the whole refrigeration system is then set by the minimum evaporator temperature needed for any particular case.

$$LF_{case} = \frac{\dot{Q}_{case,actual}}{\dot{Q}_{case,rated}}; 0.5 \leq LF_{case} \leq 1.0$$

$$T_{Evap,Allowed} = T_{case} - LF_{case} (T_{case} - T_{Evap,Design})$$

where:

LF_{case} = Load factor for a particular case

T_{evap} = Evaporator temperature, C.

Detailed Refrigeration Systems

The detailed refrigeration system object (Refrigeration:System) is an alternative to the refrigerated case compressor rack object (Refrigeration:CompressorRack). Either works in conjunction with the refrigerated case object (Refrigeration:Case) to simulate the performance of a refrigerated case system. However, the detailed system model is never used for a stand-alone case that rejects heat to the zone. The detailed system modeling differs from the simpler modeling, using a pair of refrigerated case and compressor objects, in that it:

- (1) requires performance data for each compressor,
- (2) requires condenser performance curves for air- and evaporative-cooled condensers,
- (3) explicitly calculates the amount of superheat available for reclaim in an optional air or water heating coil,
- (4) allows the suction temperature to rise when the case loads are less than the design loads, thus improving compressor efficiency,
- (5) allows the use of .mechanical subcoolers to transfer a part of the refrigeration load from a lower-temperature system to a more efficient higher-temperature system,
- (6) allows the use of liquid suction heat exchangers which will improve the cycle efficiency for some refrigerants,
- (7) models three condenser fan types,

(8) allows the user to keep track of refrigerant inventory,

(9) does not assume that the compressor and condenser capacity is sufficient to meet the case loads, but carries unmet load over to the next time step.

Refrigeration System Convergence

The performance of a refrigeration system is highly dependent upon the condensing temperature. The calculation starts with an estimated condensing temperature, which is used to calculate the compressor power use and the refrigerant properties for any optional subcoolers. These values are in turn used to determine the total heat rejection load on the condenser, which produces a new estimate for the condensing temperature. A few iterations are usually necessary to converge upon the final condensing temperature for each time step.

Condenser Performance

Only one condenser is allowed per system. The condenser can be modeled as dry air cooling, wet evaporative cooling, or water loop cooling. The condenser performance is modeled to determine: (1) the condensing temperature, which will influence the efficiency of the compressors, (2) the enthalpy of the refrigerant entering the refrigerated cases attached to the suction group, (3) auxiliary power consumption for fans and pumps, and (4) water consumption for evaporative and water-cooled condensers. The total heat rejection load on the condenser is the sum of the case load, any mechanical subcooler loads on the system, and the total compressor power. The condenser load is reduced by any heat reclaimed by desuperheating coils for HVAC or water heating purposes.

$$H_{rej} = \sum \text{Case Loads} + \sum \text{Compressor Power} + \sum \text{Mechanical Subcooler Loads} - \sum \text{Heat Reclaimed}$$

where:

H_{rej} = Heat rejected by the condenser (W)

$\sum \text{Case Loads}$ = Sum of all the refrigeration loads from the connected cases (W)

$\sum \text{Compressor Power}$ = Sum of all the power input to all compressors operating (W)

$\sum \text{Mechanical Subcooler Loads}$ = Sum of all the refrigeration loads for all subcoolers served by this system (W)

$\sum \text{Heat Reclaimed}$ = Sum of all the heat reclaimed by desuperheater coils (W)

The heat rejection environment is either set to the ambient conditions or to those corresponding to a defined outside air node (sometimes used to represent condensers located above ground level).

The enthalpy of the condensed refrigerant leaving the condenser is equal to:

$$h_{\text{condenser, out}} = h_{\text{sat, liquid}}(T_{\text{condense}}) - c_{p, \text{sat, liquid}}(T_{\text{condense}}) \times \Delta T_{\text{RatedSubcooling}}$$

where

$h_{\text{condenser, out}}$ = enthalpy leaving the condenser, J/kg

$h_{\text{sat, liquid}}$ = enthalpy of saturated liquid at the condensing temperature, J/kg

$c_{p, \text{sat, liquid}}$ = specific heat of saturated liquid at the condensing temperature, J/kg-C

$\Delta T_{\text{RatedSubcooling}}$ = Amount of subcooling included in condenser rated heat rejection, C

A minimum condensing temperature is specified for the detailed refrigeration system, and is usually required to maintain acceptable thermal expansion valve performance. When the calculated condensing temperature is less than this minimum, the air flow for air and evaporative-cooled condensers is reduced to reduce the condenser capacity and maintain the required condensing temperature.

Air-Cooled Condensers

The heat rejection capacity of a dry air-cooled condenser object (Refrigeration:Condenser:AirCooled) is directly proportional to the difference between the condensing temperature and the outdoor air drybulb temperature. The manufacturers typically provide the performance data, at one standard atmosphere, in a linear relationship between heat rejection and temperature difference. A correction factor is applied to account for the variation in air density with elevation (Carrier 1999).

$$Hrej_{Rated} = C_1 + C_2 \times (T_{condense} - T_{drybulb})$$

$$Hrej_{Rated,corrected} = Hrej_{Rated} \times (1 - 7.17E - 5 \times Elevation)$$

where:

$Hrej_{Rated}$ = Manufacturer's rated heat rejected by the condenser (W)

C_1 = Intercept taken from manufacturer's condenser performance data (W)

C_2 = Coefficient taken from the manufacturer's condenser performance data (W/C)

$T_{condense}$ = Condensing temperature (C)

$T_{drybulb}$ = Drybulb temperature for the local environment (C)

Elevation = Local elevation (m)

The manufacturer's form of performance data is used internally to define the condensing temperature as a function of the heat rejection load.

$$T_{condense} = T_{drybulb} + \left(\frac{Hrej - C_1}{C_2} \right) \div (1 - 7.17E - 5 \times Elevation)$$

This calculated condensing temperature is then compared to the minimum condensing temperature allowed for that system. If necessary, the air flow to the condenser is reduced to maintain the condensing temperature at or above that minimum value.

Condenser Fan Energy Use

Condenser fan power for air-cooled condensers is determined by the type of fan control, fixed, variable speed, or two-speed. For all three fan control types, the fan power is set equal to the rated fan power whenever the calculated condensing temperature is greater than or equal to the minimum allowed condensing temperature. If the calculated temperature is less than the minimum allowed, the condenser air flow must be reduced. The air flow for reduced condenser capacity is therefore:

$$Hrej \propto (AirVelocity)^N$$

$$Air\ Velocity \propto (Hrej)^{1/N}$$

$$Air\ Volume\ Ratio = \left(\frac{Hrej}{Hrej_{Rated}} \right)^{1/N}$$

where:

N = 0.633 for turbulent air flow over cylinders (ASHRAE 2005)

$T_{environment}$ = environmental sink temperature; $T_{drybulb}$.

For a fixed-speed fan, the air flow is reduced through the use of dampers or by cycling the fan on and off. In either case, the power variation with air flow volume is approximately linear above a minimum power level of 10%. This is approximated by the following equation:

$$P_{CondFan} = (\text{Air Volume Ratio})P_{CondFan,design}$$

where:

$P_{CondFan}$ = output variable “Refrigerated Case Condenser Fan Electric Power [W]”, not allowed to exceed the design condenser fan power

$P_{CondFan,design}$ = design condenser fan power (W)

For a variable speed fan, the power is proportional to the cube of the air flow, above a minimum value of 25%, and as shown in the following equation:

$$P_{CondFan} = (\text{Air Volume Ratio})^3 P_{CondFan,design}$$

For a two-speed fan, the fan power is held constant at the design level for Air Volume Ratios greater than or equal to 0.6. For lower Air Volume Ratios, the Fan power is set equal to 1/8 of the design value, which corresponds to a half-speed fan setting.

For a water cooled condenser, there is no fan load at the condenser (i.e., the water/refrigerant heat exchanger). Any fan load would be related to and accounted for at the heat rejection object (e.g., cooling tower).

Evaporative-Cooled Condensers

The input object Refrigeration:Condenser:EvaporativeCooled allows using evaporative cooling rather than dry air cooling which will allow for more efficient condenser heat rejection based on the entering air approaching the wet-bulb temperature rather than the dry-bulb temperature. Analyses under the International Energy Agency’s (IEA) Heat Pumping Programme Annex 26 indicates that this measure can improve refrigeration system efficiency by up to 10% (IEA 2003). The basin heater energy and water pumping power consumption for evaporative condensers in the detailed refrigeration system is modeled as described for the Refrigeration:CompressorRack. Just as for air-dried condensers, an elevation correction is needed to adjust for the variation in density of the air. This correction factor was derived by combining the barometric pressure correction from ARI 490 and a standard correlation for barometric pressure as a function of elevation (ARI 2008, NASA 1976).

$$Hrej_{Rated,Corrected} = Hrej_{Rated} \times \left[1 + k_1 BP_{std} (1 - e^{(A \times Elev)}) \right]$$

$$A = \frac{g_0 \times M_0}{R^* \times T_b} = -0.00012 m^{-1}$$

where

k_1	=	0.0023 for pressure stated in kPa, (ARI 2008)
BP_{std}	=	Standard atmosphere at rating conditions (101.0 kPa)
g_0	=	Gravitational constant (9.80665 m/s ²)
R^*	=	Universal gas constant (8.31432E3 N-m/kmol-K)
M_0	=	Molar mass of air (28.9644 kg/kmol)
T_b	=	Standard temperature (288.15 K)

Although based upon an exponential relationship, the resulting correction is very nearly linear within the range of elevations found upon dry land, so the following form of correction is used:

$$Hrej_{Rated,Corrected} = Hrej_{Rated} \times (1 - 3.07E - 5 \times Elevation)$$

To calculate the condensing temperature for an evaporative cooled condenser, it is necessary to provide the manufacturer’s performance data. The manufacturers typically

provide this data as a table of condensing temperature as a function of both entering wet-bulb temperature and the ratio of the heat rejected to the rated heat rejected. This data can be well represented, as shown in Figure 216, by a regression of the form:

$$(T_{condense} - T_{wetbulb}) = C_1 + C_2 \times HRCF + \frac{C_3}{HRCF} + C_4 T_{wetbulb}, \text{ or}$$

$$T_{condense} = C_1 + C_2 \times HRCF + \frac{C_3}{HRCF} + (1 + C_4) T_{wetbulb} \quad (545)$$

where: $HRCF = \frac{Hrej_{Rated}}{Hrej}$

$C_1, C_2, C_3,$ and C_4 = Coefficients determined by regression from manufacturer's data. Figure 216 shows a comparison between this equation form, which produced an adjusted R^2 of 0.998 and a maximum residual of 0.7C, for one manufacturer of evaporative condensers. Data from two other manufacturers showed similar agreement with this parameterization.

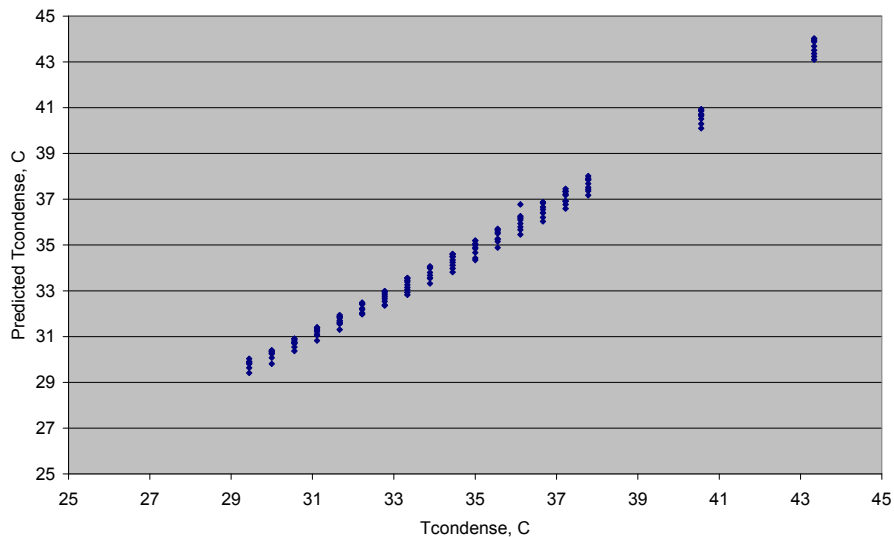


Figure 216. Comparison of the condensing temperature predicted by four-factor equation to manufacturer's data

Again, the condensing temperature is not allowed to fall below the system's minimum allowed condensing temperature. Just as with an air-cooled condenser, the air flow through the condenser is controlled to maintain this minimum condensing temperature and the air velocity reduction is a function of the decreased capacity (Manske, 1999). For an evaporative condenser, relevant capacity is not the amount of heat rejected, but the rated capacity at that reduced air flow. That decreased rated capacity must first be calculated based upon the specified minimum condensing temperature. Using Equation (545), the specified condensing temperature is used to calculate the reduced HRCF, which is used with the current heat rejection to calculate the "reduced Rated Heat Rejection".

$$0 = C_2 \times HRCF^2 + ((C_1 + C_4 T_{wetbulb} - (T_{condense} - T_{wetbulb})) \times HRCF + C_3$$

$$\text{Reduced Rated Heat Rejection} = HRCF \times Hrej$$

$$\text{Air Volume Ratio} = \left(\frac{\text{Reduced Rated Heat Rejection}}{Hrej_{Rated}} \right)^{1/N}$$

where:

N = exponent for evaporative condensers, set to 0.76 (Manske, 1999)

The water consumption for an evaporative condenser is calculated based upon the air flow rate, the total heat rejection, and the heat rejection environment. The amount of water consumption also includes the amount of water that is purged to reduce the concentration of contaminants. The purge water is estimated as proportional to the heat rejection, at a rate of $5.0E-10 \text{ m}^3/\text{s}$ per Watt of heat rejection (B.A.C., 2007). (This value, which corresponds to 3 gal./min. per 100 tons, is slightly more conservative than the value quoted by ASHRAE, 2004.) For the compressor racks, the condenser effectiveness was input as a function of the environmental wetbulb temperature. For the detailed evaporative condenser, the input data instead describes the capacity as a function of environmental conditions and loading. From that data, the water evaporation is calculated using the effectiveness corresponding to a fully loaded condenser. When the condenser is operating outside the bounds of the manufacturer's data, the effectiveness is limited to a maximum value of 0.9.

$$\eta = \frac{Hrej}{\dot{V}_{air, rated} \times \rho_{air} \times (h|_{T_{condense, sat}} - h_{air, in})}$$

$$h_{air, out} = h_{air, in} + \eta \times (h|_{T_{condense, sat}} - h_{air, in})$$

$$T_{air, out} = T_{saturated}(h_{air, out}, P_{barometric})$$

$$\dot{V}_{evaporation} = \frac{\text{AirVolumeRatio} \times \dot{V}_{air, rated} \times \rho_{air, dry} \times (\omega_{air, out} - \omega_{air, in})}{\rho_{water}}$$

$$\dot{V}_{makeup} = \dot{V}_{evaporation} + \dot{V}_{purge}$$

where:

$h|_{T_{condense, sat}}$ = enthalpy of saturated air at the calculated condensing temperature

$h_{air, out}$ = enthalpy of the air leaving the condenser,

$h_{air, in}$ = enthalpy of the inlet air, psychometric function of inlet air drybulb temperature and humidity ratio

$\dot{V}_{air, rated}$ = rated volumetric air flow for the evaporative condenser (input value) m^3/s

ρ_{air} = density of air evaluated at environmental conditions

$\rho_{air, dry}$ = density of dry air evaluated at environmental temperature

$T_{air, out}$ = air temperature leaving the condenser, psychometric function of saturated air at the enthalpy leaving the condenser and the barometric pressure

$\dot{V}_{evaporation}$ = Volumetric rate of water evaporation in the condenser, m^3/s

$\omega_{air, out}$ = humidity ratio ($\text{kg}_{water}/\text{kg}_{dry air}$) of the air leaving the condenser, psychometric function of $T_{air, out}$ and the barometric pressure

$\omega_{air, in}$ = humidity ratio ($\text{kg}_{water}/\text{kg}_{dry air}$) of the air at environmental conditions

ρ_{water} = density of water evaluated at the environmental wetbulb temperature (kg/m³)

\dot{V}_{purge} = Volumetric rate of water purged in the condenser, m³/s

\dot{V}_{makeup} = Volumetric rate of water makeup in the condenser, m³/s

The source of the makeup water may be specified as a water storage tank. If not specified, the makeup water is assumed to come from the building mains (Ref. Water Mains Temperatures).

An evaporative condenser can be scheduled, using the Evaporative Condenser Availability Schedule described previously, so that it operates in a dry mode for a portion of the year. This is important in climates subject to freezing weather in order to avoid excessive ice formation on the condenser surfaces and surroundings. (The Availability Schedule is the correct way to model the use of evaporative condensers in cold climates. However, some users may take a single input description and use it to model a building with a refrigeration system in a variety of climates. To avoid modeling the use of evaporative coolers in freezing weather, the code includes a cutout to switch to dry operation whenever the outdoor drybulb temperature drops below 4C.) Dry operation can also reduce water use when the dry heat rejection capacity of the equipment is sufficient to meet the load during times of the year when the outside drybulb temperature is reduced. In dry operation, the condenser heat rejection capacity is approximately one third of the rated wetted heat rejection capacity (Manske, 2000). In dry operation, the condensing temperature is estimated by using the same four-factor equation, but using the air drybulb temperature instead of the wetbulb temperature and using the reduced heat rejection capacity factor.

$$HRCF_{\text{dry operation}} = HRCF_{\text{wet operation}} / 3.0$$

$$T_{\text{condense,dry operation}} = C_1 + C_2 \times HRCF_{\text{dry operation}} + \frac{C_3}{HRCF_{\text{dry operation}}} + (1 + C_4)T_{\text{drybulb}}$$

Water-Cooled Condensers

If the condenser heat rejection is specified as water cooled (input object Refrigeration:Condenser:WaterCooled), the model uses the same algorithms described above for Refrigeration Compressor Racks. The condensing temperature is set equal to the inlet water temperature plus an approach temperature equal to the difference between the rated values for water inlet temperature and condensing temperature.

Compressor Energy Use

The compressor object (Refrigeration:Compressor) calculations start with the determination of the inlet (suction) and outlet (discharge) conditions. The suction pressure is defined by the saturated suction temperature (equal to the evaporating temperature in the refrigerated cases connected to the suction group) minus the pressure drop in the suction pipes. With proper design, this pressure drop typically corresponds to a saturated suction temperature drop of about 1C. The saturated discharge pressure is defined by the condensing temperature plus the pressure drop in the discharge pipes. With proper design, this discharge pipe pressure drop typically corresponds to a saturated discharge temperature increase of about 0.5C (ASHRAE 2006a). These two temperatures are then used with the manufacturer's performance curves for each compressor. The performance curves are defined in ARI Standard 540 and take the following form (ARI 2004):

$$X = C_1 + C_2(S) + C_3(D) + C_4(S^2) + C_5(SD) + C_6(D^2) + C_7(S^3) + C_8(DS^2) + C_9(SD^2) + C_{10}(D^3)$$

$$S = T_{\text{evap}} - 1.$$

$$D = T_{\text{condense}} + 0.5$$

where:

X = can represent power input (W), or cooling capacity(W)

C = compressor performance coefficient (be sure to see the IO Reference guide because the Energy Plus input order for this equation does not match this ARI form)

S = Saturation temperature corresponding to the suction pressure (C)

D = Saturation temperature corresponding to the discharge pressure (C)

T_{evap} = Evaporating temperature (C)

The rated values for the cooling capacity and power consumption from the manufacturer include a specified amount of subcooling before the thermal expansion valve and a certain amount of superheat in the suction gas. Adjustments must be made to these rated values to reflect the actual subcooling and superheat conditions. Actual subcooling is determined by the condenser's rated subcooling and by the subcooling provided by optional subcoolers. The actual superheat is determined by the refrigerated case superheat (usually set to ensure that there is no liquid in the suction lines leading to the compressors), set here at 4C, and the effect from any optional subcoolers(ASHRAE 2006b). These various state points are shown in Figure 217.

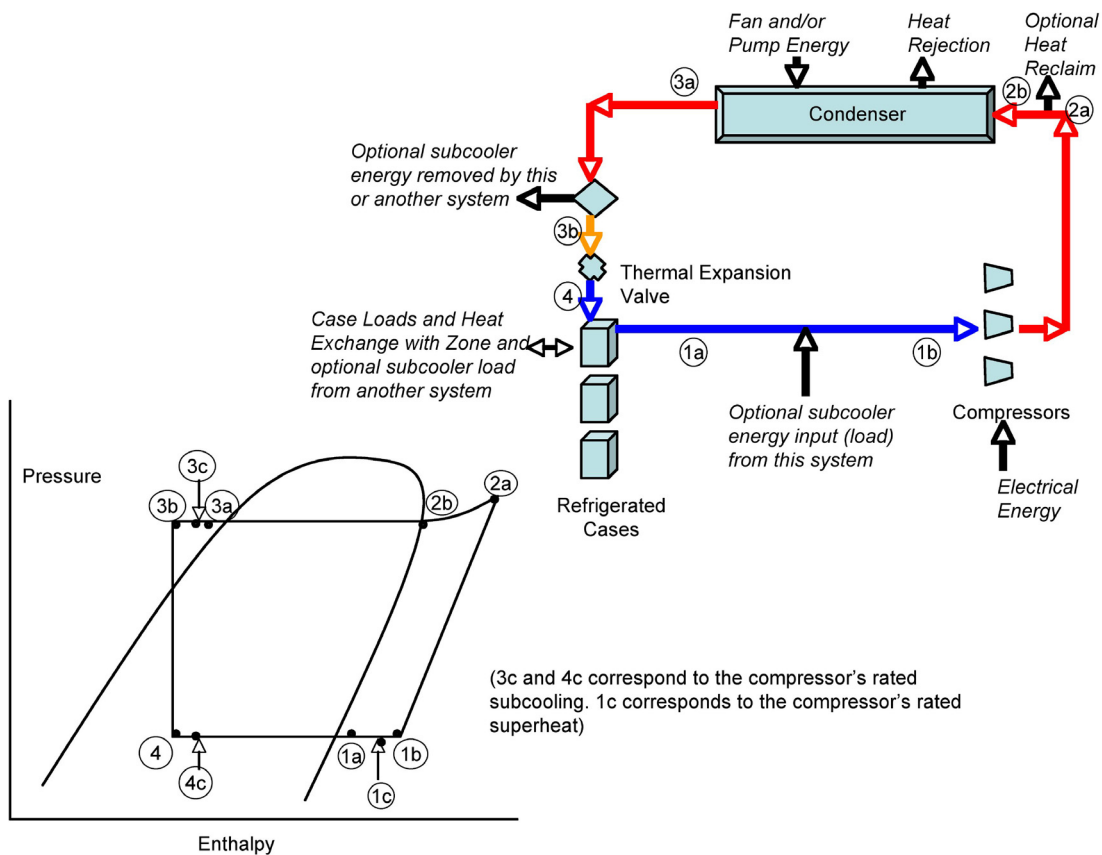


Figure 217. State Points and Energy Flows for Detailed Refrigeration System

Once the corrected capacity is calculated for each compressor, the compressors are dispatched one at a time until the system load is met. The last compressor dispatched is assumed to run at full load for the fraction of the time step necessary to meet the load. That is, the model neglects compressor cycling losses at part-load conditions. Using the state point identification from Figure 217, these corrections are shown in the following equations. If the capacity available from all the compressors is less than the sum of the case loads for that time period, the unmet load is accumulated to be met in succeeding time steps. If this accumulated unmet load becomes too great, a warning message is generated.

$$Cap_{corrected} = \frac{\rho_{1b}}{\rho_{1c}} \times \frac{(h_{1b} - h_4)}{(h_{1c} - h_{4c})} Cap_{rated}$$

$$\dot{m} = \frac{Cap_{corrected}}{(h_{1b} - h_4)}$$

$$\text{System Load} = \sum \text{Case Loads} + \sum \text{Mechanical Subcooler Loads}$$

where:

\dot{m} = mass flow rate of refrigerant, kg/s

ρ = density, kg/m³

h = enthalpy, J/kg

Cap = refrigeration capacity of an individual compressor, W

Load = total system refrigeration load

Compressor performance can also be improved by allowing the suction pressure to rise whenever the sum of the loads on the refrigerated cases served by the compressors is less than the design load. The calculation of the maximum allowable evaporator temperature is described in “Variable Evaporator Temperature” in the discussion of Refrigeration Cases.

Superheat Reclaim Heating Coil

EnergyPlus can simulate waste heat being reclaimed from a detailed refrigeration system for use by refrigerant-to-air and refrigerant-to-water heating coils. Heat reclaimed from the compressor rack is assumed to be recovered from the superheated refrigerant gas leaving the compressor(s) and does not directly impact the performance of the compressors. Heat reclaimed from the detailed refrigeration system is limited to the portion of the rejected heat in the superheat region. Using the state point nomenclature from Figure 217, this value is calculated by the detailed compressor and condenser models each time step as follows:

$$\dot{Q}_{AvailableSuperheat} = \dot{m}(h_{2a} - h_{2b})$$

Refer to objects Coil:Heating:Desuperheater and Coil:WaterHeating:Desuperheater for a complete description of how these coils are modeled.

Subcoolers

Subcooler objects (Refrigeration:Subcooler) reduce the temperature of the liquid refrigerant after it leaves the condenser and before it reaches the thermal expansion valve. The detailed refrigeration system permits the use of two type of subcoolers, mechanical and liquid suction. A mechanical subcooler is used to transfer refrigeration load from a lower-temperature system to a higher-temperature system. For example, the compressors that are used to provide cooling for dairy products could be used to subcool the refrigerant in another system that is serving frozen food cases. For the system providing the cooling, the mechanical subcooler acts like another refrigerated case load. For the system receiving the cooling, the mechanical subcooler reduces the enthalpy of the refrigerant from point 3a to point 3b on Figure 217, and thus reduces the required refrigerant flow rate. Mechanical subcooler performance is defined by the controlled temperature of the subcooled liquid as follows:

$$\dot{Q} = \dot{m} \times c_p (T_{3a} - T_{control})$$

$$h_{3b} = h_{3a} - c_{p,liquid} (T_{3a} - T_{control})$$

where:

\dot{Q} = subcooler load, W

\dot{m} = mass flow rate of refrigerant, kg/s

$c_{p,liquid}$ = specific heat of saturated liquid at the condensing temperature

$T_{control}$ = control temperature specified for the mechanical subcooler

h = enthalpy

A liquid suction heat exchanger (LSHX) subcooler uses the cold gas exiting the refrigerated cases to subcool the condensed liquid refrigerant in the same system. Depending upon the shape of the refrigerant's saturation curve and the operating condensing and evaporating temperature, this can save significant energy by reducing the required refrigerant flow (ASHRAE 2006a). (This model neglects the pressure drop through the suction side of the heat exchanger, although this pressure drop will cause the compressor to operate at a lower suction pressure.) A liquid suction heat exchanger is defined by specifying the design values for: inlet liquid temperature, inlet vapor temperature, and liquid temperature change. A liquid suction heat exchanger subcooler will also increase the superheat of the gas returning to the compressor, as shown by the difference between state points 1a and 1b in Figure 217:

$$\eta_{LSHX} = \frac{\Delta T_{Design}}{(T_{Liquid\ Design} - T_{Vapor\ Design})}$$

$$\dot{Q} = \dot{m} \times c_{p,liquid} \eta_{LSHX} (T_{3a} - T_{1a})$$

$$T_{1b} = T_{1a} + \frac{\dot{Q}}{\dot{m} c_{p,vapor}}$$

$$h_{3b} = h_{3a} - \frac{\dot{Q}}{\dot{m}}$$

where:

η_{LSHX} = subcooler effectiveness, dimensionless

$c_{p,vapor}$ = specific heat of saturated vapor at the evaporating temperature

$T_{control}$ = control temperature specified for the mechanical subcooler

If a system is subcooled by both a mechanical subcooler and a liquid subcooler, the liquid subcooler will follow the mechanical subcooler and those points labeled '3a' in the liquid subcooler equations would correspond to the points labeled '3b' in the mechanical subcooler equations, that is, the inlet of the LSHX would be the outlet of the mechanical subcooler.

Any one system can be cooled (i.e., have energy removed between points '3a' and '3b') by at most one liquid suction heat exchanger and one mechanical subcooler. However, a system can provide cooling to multiple mechanical subcoolers. For example if a building had one high temperature refrigeration system (perhaps cooling fresh produce) and three low temperature systems (perhaps cooling frozen foods and meat), each of the three low temperature systems could include a mechanical subcooler with the refrigeration energy for all three absorbed by the one high-temperature system. For the compressors and condenser on the high-temperature system, these three mechanical subcoolers would represent a load very similar to that of the refrigerated cases served by that system.

References

ARI 2003. Standard for Remote Mechanical-Draft Evaporatively-Cooled Refrigerant Condensers, Standard 490, Air-Conditioning & Refrigeration Institute, Arlington, VA

- ARI 2004. Standard for Performance Rating of Positive Displacement Refrigerant Compressors and Compressor Units, Standard 540, Air-Conditioning & Refrigeration Institute, Arlington, VA
- ARI 2005. Standard for Performance Rating of Remote Mechanical-Draft Air-Cooled Refrigerant Condensers, Standard 460, Air-Conditioning & Refrigeration Institute, Arlington, VA
- ARI 2007. Standard for Performance Rating of Water-Cooled Refrigerant Condensers, Remote Type, Standard 450, Air-Conditioning & Refrigeration Institute, Arlington, VA
- ASHRAE. 2002. *Refrigeration Handbook*, Chapter 47. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- ASHRAE. 2004. *HVAC Systems and Equipment Handbook*, Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- ASHRAE. 2006a. *Refrigeration Handbook*, Chapter 2. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- ASHRAE. 2006b. *Refrigeration Handbook*, Chapter 44. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- B.A.C. 2007, Baltimore AirCoil Company Product and Application Handbook, Volume II, Baltimore, MD
- Henderson, H.I. and Khattar, M. 1999. Measured Impacts of Supermarket Humidity Level on Defrost Performance and Refrigerating System Energy Use. *ASHRAE Transactions* 105(1), 508-520. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- Howell, R.H. 1993a. Effects of Store Relative Humidity on Refrigerated Display Case Performance. *ASHRAE Transactions* 99(1), 667-678. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- Howell, R.H. 1993b. Calculation of Humidity Effects on Energy Requirements of Refrigerated Display Cases. *ASHRAE Transactions* 99(1), 679-693. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.
- IEA Heat Pump Centre. 2003. *Advanced Supermarket Refrigeration/Heat Recovery Systems Vol. 1 - Executive Summary*, Report HPP-AN26-2, April.
- Manske, K.A., Performance Optimization of Industrial Refrigeration Systems, M.S.Thesis Mechanical Engineering, Solar Energy Laboratory, University of Wisconsin-Madison, 2000.
- Mitchell, J.W., et al. 1992. Analysis of Supermarket Dehumidification Alternatives. Final Report to Electric Power Research Institute, Report TR-100352, November.
- NASA 1976, U. S. Standard Atmosphere, NASA-TM-74335, National Oceanic and Atmospheric Administration, National Aeronautics and Space Administration

Setpoint Managers

Overview

Setpoint Managers are one of the high-level control constructs in EnergyPlus. A Setpoint Manager is able to access data from any of the HVAC system nodes and use this data to calculate a setpoint (usually a temperature setpoint) for one or more other HVAC system nodes. Setpoints are then used by Controllers as a goal for their control actions.

Setpoint managers are executed at the start of each HVAC time step, and they reside outside the HVAC system iteration loops. Thus, the Setpoint Managers are executed once per HVAC time step, and they use previous time step information (except for zone load) to calculate their setpoints.

All temperatures in the following descriptions are in degrees C, loads are in Watts, mass flow rates are in kg/s.

Scheduled

The input object SetpointManager:Scheduled provides the simplest setpoint manager that allows the user to specify a seasonal (or other) scheduled reset, for example, of the supply air temperature setpoint.

$$T_{set} = T_{sched}$$

where T_{set} is the supply air temperature setpoint and T_{sched} is the current value (°C) from the user input temperature schedule. In this case, T_{set} will be applied to the node or nodes specified in the input object SetpointManager:Scheduled. There are a number of other types of setpoints that can be set, by choosing different control variables in the object, including:

- Maximum temperature of fluid at node (°C)
- Minimum temperature of fluid at node (°C)
- Humidity ratio of fluid at node (kg water/ kg dry air)
- Maximum humidity ratio of fluid at node (kg water/ kg dry air)
- Minimum humidity ratio of fluid at node (kg water/ kg dry air)
- Mass flow rate of fluid at node (kg/s)
- Maximum mass flow rate of fluid at node (kg/s)
- Minimum mass flow rate of fluid at node (kg/s)

Outdoor Air Reset

The input object SetpointManager:OutdoorAirReset provides a setpoint manager that implements an outdoor air reset strategy for the supply air temperature. The user defines a reset rule for this strategy by specifying two setpoint temperatures at two outside drybulb temperatures. Generally the lower setpoint temperature is matched with the higher drybulb temperature and the higher setpoint temperature with the lower drybulb temperature. The user can specify two reset rules and schedule their usage. A schedule value of 1 indicates use of the 1st rule; 2 indicates use of the 2nd.

IF $SchedVal = 2$

$$T_{out,low} = T_{2,out,low}$$

$$T_{out,high} = T_{2,out,high}$$

$$T_{set,atOALow} = T_{2,set,atOALow}$$

$$T_{set,atOAhigh} = T_{2,set,atOAhigh}$$

otherwise

$$T_{out,low} = T_{1,out,low}$$

$$T_{out,high} = T_{1,out,high}$$

$$T_{set,atOALow} = T_{1,set,atOALow}$$

$$T_{set,atOAhigh} = T_{1,set,atOAhigh} \cdot$$

If $T_{out,low} < T_{out,high}$ and $T_{set,atOAlow} > T_{set,atOAhigh}$ then

If $T_{out} \leq T_{out,low}$ then

$$T_{set} = T_{out,low}$$

if $T_{out} \geq T_{out,high}$ then

$$T_{set} = T_{out,high}$$

otherwise

$$T_{set} = T_{set,atOAlow} - ((T_{out} - T_{out,low}) / (T_{out,high} - T_{out,low})) \cdot (T_{set,atOAlow} - T_{set,atOAhigh})$$

otherwise

$$T_{set} = 0.5(T_{set,atOAlow} + T_{set,atOAhigh}) \cdot$$

T_{set} will be applied to the node or nodes specified in the *SetpointManager:OutdoorAirReset* object input.

Single Zone Reheat Heating and Cooling

The input object *SetpointManager:SingleZone:Reheat* provides a setpoint manager that is used to implement a variable supply air setpoint temperature in which the setpoint is varied each timestep to meet the heating or cooling load in a control zone. The manager operates in the following manner.

$$T_{set} = T_z + \dot{Q}_z / (C_{p,air} \cdot \dot{m}_z)$$

where T_z is the control zone temperature, \dot{Q}_z is the zone load (greater than zero for heating, less than zero for cooling), \dot{m}_z is the zone supply air mass flow rate, and $C_{p,air}$ is the specific heat of air. If \dot{m}_z is very small ($\leq .001$ kg/s) T_{set} is set equal to $T_{set,min}$ if the control zone has a cooling load and to $T_{set,max}$ if the control zone has a heating load. $T_{set,min}$ is the user specified minimum setpoint and $T_{set,max}$ is the user specified maximum setpoint. T_{set} will be applied to the node or nodes specified in the *SetpointManager:SingleZone:Reheat* object input.

Single Zone Heating Only

The input object *SetpointManager:SingleZone:Heating* provides a model that detects the control zone load to meet the current heating setpoint (Report Variable "Zone/Sys Sensible Load to Heating Setpoint Predicted[W]", zone inlet node flow rate, and zone node temperature, and calculates a setpoint temperature for the supply air that will satisfy the zone heating load for the control zone. "Zone/Sys Sensible Load to Heating Setpoint Predicted[W]" > 0 indicates a heating load. The following rules are applied:

- If there is no zone heating load, then the setpoint is set at the specified minimum supply air temperature.
- If there is a zone heating load and the zone supply mass flow rate is less than 0.001 kg/s, then the setpoint is set at the specified maximum supply air temperature.
- If there is a zone heating load and the zone supply mass flow rate is greater than 0.001 kg/s, then the setpoint is set calculated as follows:

$$\text{SetPoint} = \text{ZoneTemp} + \text{ZoneLoadtoHeatSP}/(\text{CpAir} * \text{ZoneMassFlow})$$

where:

SetPoint = Setpoint temperature applied to the specified setpoint node(s)

ZoneTemp = Current zone temperature

ZoneLoadtoHeatSP = Zone heating load (Report Variable "Zone/Sys Sensible Load to Heating Setpoint Predicted[W]")

CpAir = Specific heat of zone supply air

ZoneMassFlow = Zone supply air mass flow rate

Single Zone Cooling Only

The input object SetpointManager:SingleZone:Cooling provides a model that detects the control zone load to meet the current cooling setpoint (Report Variable "Zone/Sys Sensible Load to Cooling Setpoint Predicted[W]", zone inlet node flow rate, and zone node temperature, and calculates a setpoint temperature for the supply air that will satisfy the zone cooling load for the control zone. "Zone/Sys Sensible Load to Cooling Setpoint Predicted[W]" < 0 indicates a cooling load. The following rules are applied:

- If there is no zone cooling load, then the setpoint is set at the specified maximum supply air temperature.
- If there is a zone cooling load and the zone supply mass flow rate is less than 0.001 kg/s, then the setpoint is set at the specified minimum supply air temperature.
- If there is a zone cooling load and the zone supply mass flow rate is greater than 0.001 kg/s, then the setpoint is set calculated as follows:

$$\text{SetPoint} = \text{ZoneTemp} + \text{ZoneLoadtoCoolSP}/(\text{CpAir} * \text{ZoneMassFlow})$$

where:

SetPoint = Setpoint temperature applied to the specified setpoint node(s)

ZoneTemp = Current zone temperature

ZoneLoadtoCoolSP = Zone cooling load (Report Variable "Zone/Sys Sensible Load to Cooling Setpoint Predicted[W]")

CpAir = Specific heat of zone supply air

ZoneMassFlow = Zone supply air mass flow rate

Single Zone Minimum Humidity

The input object SetpointManager:SingleZone:Humidity:Minimum provides a model where the zone moisture load from ZoneTempPredictorCorrector (via DataZoneEnergyDemands) is used to calculate the minimum/maximum supply air humidity ratio needed to meet minimum/maximum zone relative humidity requirement. For the Max setpoint manager is currently used in Controller:WaterCoil, when the control variable "TemperatureAndHumidityRatio". A negative MoistureLoad means a dehumidification load for the coil controller. This could be used for any system.

Single Zone Maximum Humidity

The input object SetpointManager:SingleZone:Humidity:Maximum provides a setpoint manager that allows the control of high air humidity levels in a single zone. This setpoint manager, used in conjunction with object ZoneControl:Humidistat, detects the air humidity level in a single control zone and uses air/moisture mass balances to calculate the supply air humidity ratio needed to maintain the zone relative humidity at or below a given setpoint. The calculated supply air humidity ratio is then entered as a setpoint on a designated supply air stream node. A dehumidification component placed upstream of this node can then use the

humidity ratio setpoint to control its moisture removal rate (e.g. desiccant dehumidifiers). In the case of a chilled water coil which is used for both temperature and high humidity control, this setpoint manager works in conjunction with a Controller:WaterCoil object to determine the minimum supply air temperature required to meet both the temperature (sensible) and humidity (latent) load in the control zone. (Ref: Controller:WaterCoil).

Model Description

The user must input the required information according to the IO Reference Manual (ref: SetpointManager:SingleZone:Humidity:Maximum). Specific inputs include an object name, control variable (HumidityRatio), name of the schedule defining the maximum relative humidity for the control zone, setpoint node name or list, and the zone air node name associated with the control zone (ref: ZoneHVAC:EquipmentConnections). The schedule name must refer to a valid schedule type (range 0-1) and contain values of fractional relative humidity.

This setpoint manager first converts the desired relative humidity setpoint for the control zone to humidity ratio based on the control zone dry-bulb temperature, the scheduled maximum relative humidity setpoint and outdoor barometric pressure.

$$\omega_{sp} = \text{PsyWFnTdbRhPb}(T_{db}, RH_{sp}, P)$$

where:

ω_{sp} = humidity ratio setpoint, kg H₂O/kg air

PsyWFnTdbRHPb = EnergyPlus psychrometric function, returns humidity ratio as a function of dry-bulb temperature, relative humidity, and barometric pressure

T_{db} = dry-bulb temperature in the control zone, °C

RH_{sp} = maximum relative humidity setpoint, fraction

P = outdoor barometric pressure, Pa

The model then calculates the supply air humidity ratio required to reduce the control zone relative humidity to the desired level. Using the humidity ratio setpoint (ω_{sp}) calculated above,

$$\omega_{sa} = \omega_{sp} + \frac{\dot{Q}_l}{\dot{m}}$$

where:

ω_{sa} = maximum supply air humidity ratio setpoint, kg H₂O/kg air

\dot{Q}_l = control zone latent load, kg H₂O/s (calculated by Zone Control:Humidistat)

\dot{m} = control zone mass flow rate, kg/s

All setpoint managers are executed at the beginning of the simulation time step. Therefore, the calculated setpoint is based on the resulting control zone air temperature and air mass flow rate for the previous simulation time step.

The maximum supply air humidity ratio setpoint is placed on the node(s) specified in the input for this object (using node property Humidity Ratio Maximum).

Mixed Air

The input object SetpointManager:MixedAir provides a setpoint manager that takes an already established setpoint (usually the supply air outlet node setpoint temperature), subtracts the supply fan heat gain, and applies the result as the setpoint temperature at the mixed air node (or any other node the user specifies).

$$T_{set} = T_{set,ref} - (T_{fan,outlet} - T_{fan,inlet})$$

Outdoor Air Pretreat

The input object SetpointManager:OutdoorAirPretreat provides a setpoint manager that is meant to be used in conjunction with an OutdoorAir:Mixer. The outdoor air pretreat setpoint manager is used to establish a temperature or humidity ratio setpoint in the outdoor air stream flowing into the outdoor air stream node of an outdoor air mixer. This setpoint manager determines the required setpoint in the outdoor air stream to produce the reference setpoint in the mixed air stream after mixing with return air. For example, if the temperature setpoint at the mixed air node is 15C, the return air temperature is 20C, and the outdoor air flow fraction is 0.5, the outdoor air pretreat setpoint would be set to 10C. This setpoint manager references four user-specified nodes to obtain the following values:

- Reference setpoint node – desired mixed flow condition = RefNodeSetPoint
- Mixed air stream node – mixed air flow rate = MixedOutNodeMassFlowRate
- Outdoor air stream node – outdoor air flow rate = OAInNodeMassFlowRate
- Return air stream node – return air condition = ReturnInValue

The following calculations are used to determine the new setpoint value (SetPoint):

$$OAFraction = OAInNodeMassFlowRate / MixedOutNodeMassFlowRate$$

$$SetPoint = ReturnInValue + (RefNodeSetPoint - ReturnInValue)/OAFraction$$

Depending on the specified control variable type, the setpoints and conditions may be humidity ratio (max or min) or temperature. Note that zero is not allowed as the computed setpoint humidity ratio, because zero is used as a special value to indicate that no humidification or dehumidification is needed. If the reference humidity ratio setpoint value is zero, the zero value will be passed directly to the setpoint Node(s).

Warmest Zone Supply Air Reset

The input object SetpointManager:Warmest provides a setpoint manager that attempts to establish a supply air setpoint that will meet the cooling load of the zone needing the coldest air at the maximum zone supply air flowrate. The algorithm loops over all the zones that the system can cool and calculates

$$T_{set,z} = T_z + \dot{Q}_z / (C_{p,air} \cdot \dot{m}_{z,max})$$

Note that for cooling $\dot{Q}_z < 0$. The lowest $T_{set,z}$ is chosen as T_{set} . T_{set} is constrained to be less than or equal to the maximum setpoint temperature (user input) and greater than or equal to the minimum setpoint temperature (user input). If the sum of the zone cooling loads is very small, the setpoint temperature is set to the maximum. T_{set} will be applied to the node or nodes specified in the SetpointManager:Warmest object input.

Coldest Zone Supply Air Reset

The input object SetpointManager:Coldest provides a setpoint manager that attempts to establish a supply air setpoint that will meet the heating load of the zone needing the warmest air at the maximum zone supply air flowrate. The algorithm loops over all the zones that the system can heat and calculates

$$T_{set,z} = T_z + \dot{Q}_z / (C_{p,air} \cdot \dot{m}_{z,max})$$

Note that for heating $\dot{Q}_z > 0$. The highest $T_{set,z}$ is chosen as T_{set} . T_{set} is constrained to be less than or equal to the maximum setpoint temperature (user input) and greater than or equal to the minimum setpoint temperature (user input). If the sum of the zone heating loads is very small, the setpoint temperature is set to the minimum. T_{set} will be applied to the node or nodes specified in the SetpointManager:Coldest object input.

Return Air Bypass Flow

The input object SetpointManager:ReturnAirBypassFlow provides a setpoint manager that sets the air flow rate in a bypass duct such that when the bypassed and non-bypassed air are mixed the resultant air stream will be at the user-specified setpoint temperature.

The user specifies the desired setpoint temperature T_{set} through a input temperature schedule.

This temperature is modified to account for any potential fan heat gain:

$$T_{set,mod} = T_{set} - (T_{loop,out} - T_{mixer,out})$$

Here $T_{loop,out}$ is the temperature at the air loop outlet node and $T_{mixer,out}$ is the temperature at the outlet node of the bypass – nonbypass air mixer. Depending on the system configuration these may be the same node. Then

$$\dot{m}_{bypass} = (\dot{m}_{tot} \cdot T_{set,mod} - \dot{m}_{nonbypass} \cdot T_{nonbypass}) / T_{bypass}$$

where \dot{m}_{tot} is the total supply air flowrate in kg/s sent to the zones, $\dot{m}_{nonbypass}$ is the nonbypassed air flowrate (the conditioned air), $T_{nonbypass}$ is the corresponding temperature just before mixing with the bypassed air, and T_{bypass} is the temperature of the bypassed (unconditioned) air. The resulting \dot{m}_{bypass} is the mass flow rate setpoint for the bypass air branch.

Warmest Temp Flow

The input object SetpointManager:WarmestTemperatureFlow provides a setpoint manager that attempts to establish a supply air setpoint that will meet the cooling load of the zone needing the coldest air at the maximum zone supply air flowrate. The supply air temperature setpoint is established by SetpointManager:WarmestTemperatureFlow using one of two strategies: 'Temp First' sets the supply air temperature to the highest temperature that will meet the cooling load of the coolest zone at minimum cfm. 'Flow First' sets the supply air temperature to the highest temperature that will meet the cooling load of the warmest zone at maximum cfm. 'Temp First' gives higher priority to reducing fan energy whereas 'Flow First' gives higher priority to reducing chiller energy.

The algorithm loops over all the zones that the system can cool and calculates according to strategy:

Temp First:

$$T_{set,z} = T_z + \dot{Q}_z / (C_{p,air} \cdot \dot{m}_{z,min})$$

Flow First:

$$T_{set,z} = T_z + \dot{Q}_z / (C_{p,air} \cdot \dot{m}_{z,max})$$

where $\dot{m}_{z,min}$ is the minimum air flow rate to the zone produced by reducing the capacity of the supply fan or by reducing the opening of the damper in the terminal box, if present, and $\dot{m}_{z,max}$ is the design air flow rate to the zone.

Note that for cooling $\dot{Q}_z < 0$. The lowest $T_{set,z}$ is chosen as T_{set} . T_{set} is constrained to be less than or equal to the maximum setpoint temperature (user input) and greater than or equal to the minimum setpoint temperature (user input). If the unconstrained value of T_{set} is less than the minimum setpoint temperature and there are no VAV terminal boxes, the fan capacity is increased so as to meet the zone cooling loads with the constrained value of T_{set} :

$$\dot{m}_{fan} = \sum_z \dot{m}_{z,min} = \sum_z \dot{Q}_z / [C_{p,air} \cdot (T_{set} - T_z)]$$

This is more likely to occur in the ‘Temp First’ case. If the sum of the zone cooling loads is very small, the setpoint temperature is set to the maximum. T_{set} will be applied to the node or nodes specified in the SetpointManager:WarmestTemperatureFlow object input.

Solar Collectors

Solar collectors are devices that convert solar energy into thermal energy by raising the temperature of a circulating heat transfer fluid. The fluid can then be used to heat water for domestic hot water usage or space heating. Flat-plate solar collectors using water as the heat transfer fluid and unglazed transpired solar collectors using air are currently the only types of collector available in EnergyPlus.

Flat-Plate Solar Collectors

The input object SolarCollector:FlatPlate:Water provides a model for flat-plate solar collectors that are the most common type of collector. Standards have been established by ASHRAE for the performance testing of these collectors (ASHRAE 1989; 1991) and the Solar Rating and Certification Corporation (SRCC) publishes a directory of commercially available collectors in North America (SRCC 2003).

The EnergyPlus model is based on the equations found in the ASHRAE standards and Duffie and Beckman (1991). This model applies to glazed and unglazed flat-plate collectors, as well as banks of tubular, i.e. evacuated tube, collectors.

Solar and Shading Calculations

The solar collector object uses a standard EnergyPlus surface in order to take advantage of the detailed solar and shading calculations. Solar radiation incident on the surface includes beam and diffuse radiation, as well as radiation reflected from the ground and adjacent surfaces. Shading of the collector by other surfaces, such as nearby buildings or trees, is also taken into account. Likewise, the collector surface can shade other surfaces, for example, reducing the incident radiation on the roof beneath it.

Thermal Performance

The thermal efficiency of a collector is defined as the ratio of the useful heat gain of the collector fluid versus the total incident solar radiation on the gross surface area of the collector.

$$\eta = \frac{(q / A)}{I_{solar}} \quad (546)$$

where

q = useful heat gain

A = gross area of the collector

I_{solar} = total incident solar radiation

Notice that the efficiency η is only defined for $I_{solar} > 0$.

An energy balance on a solar collector with double glazing shows relationships between the glazing properties, absorber plate properties, and environmental conditions.

$$\frac{q}{A} = I_{solar} \tau_{g1} \tau_{g2} \alpha_{abs} - \frac{T_{abs}^4 - T_{g2}^4}{R_{rad}} - \frac{T_{abs} - T_{g2}}{R_{conv}} - \frac{T_{abs} - T_{air}}{R_{cond}} \quad (547)$$

where

τ_{g1} = transmittance of the first glazing layer

τ_{g2} = transmittance of the second glazing layer

α_{abs} = absorptance of the absorber plate

R_{rad} = radiative resistance from absorber to inside glazing

R_{conv} = convective resistance from absorber to inside glazing

R_{cond} = conductive resistance from absorber to outdoor air through the insulation

T_{abs} = temperature of the absorber plate

T_{g2} = temperature of the inside glazing

T_{air} = temperature of the outdoor air

The equation above can be approximated with a simpler formulation as:

$$\frac{q}{A} = F_R [I_{solar} (\tau\alpha) - U_L (T_{in} - T_{air})] \quad (548)$$

where

F_R = an empirically determined correction factor

$(\tau\alpha)$ = the product of all transmittance and absorptance terms

U_L = overall heat loss coefficient combining radiation, convection, and conduction terms

T_{in} = inlet temperature of the working fluid

Substituting this into Equation (546),

$$\eta = F_R(\tau\alpha) - F_R U_L \frac{(T_{in} - T_{air})}{I_{solar}} \quad (549)$$

A linear correlation can be constructed by treating $F_R(\tau\alpha)$ and $-F_R U_L$ as characteristic constants of the solar collector:

$$\eta = c_0 + c_1 \frac{(T_{in} - T_{air})}{I_{solar}} \quad (550)$$

Similarly, a quadratic correlation can be constructed using the form:

$$\eta = c_0 + c_1 \frac{(T_{in} - T_{air})}{I_{solar}} + c_2 \frac{(T_{in} - T_{air})^2}{I_{solar}} \quad (551)$$

Both first- and second-order efficiency equation coefficients are listed in the *Directory of SRCC Certified Solar Collector Ratings*.

Incident Angle Modifiers

As with regular windows the transmittance of the collector glazing varies with the incidence angle of radiation. Usually the transmittance is highest when the incident radiation is normal to the glazing surface. Test conditions determine the efficiency coefficients for normal incidence. For off-normal angles, the transmittance of the glazing is modified by an *incident angle modifier* coefficient.

$$K_{\tau\alpha} = \frac{(\tau\alpha)}{(\tau\alpha)_n} \quad (552)$$

Additional testing determines the incident angle modifier as a function of incident angle θ . This relationship can be fit to a first-order, linear correlation:

$$K_{\tau\alpha} = 1 + b_0 \left(\frac{1}{\cos \theta} - 1 \right) \quad (553)$$

or a second-order, quadratic correlation:

$$K_{\tau\alpha} = 1 + b_0 \left(\frac{1}{\cos \theta} - 1 \right) + b_1 \left(\frac{1}{\cos \theta} - 1 \right)^2 \quad (554)$$

The incident angle modifier coefficients b_0 and b_1 are usually negative, although some collectors have a positive value for b_0 . Both first- and second-order incident angle modifier equation coefficients are listed in the *Directory of SRCC Certified Solar Collector Ratings*.

The SRCC incident angle modifier equation coefficients are only valid for incident angles of 60 degrees or less. Because these curves can be valid yet behave poorly for angles greater than 60 degree, the EnergyPlus model cuts off collector gains for incident angles greater than 60 degrees.

For flat-plate collectors, the incident angle modifier is generally symmetrical. However, for tubular collectors the incident angle modifier is different depending on whether the incident angle is parallel or perpendicular to the tubes. These are called bi-axial modifiers. Some special flat-plate collectors may also exhibit this asymmetry. The current model cannot yet handle two sets of incident angle modifiers. In the meantime it is recommended that tubular collectors be approximated with caution using either the parallel or perpendicular correlation. Incident angle modifiers are calculated separately for sun, sky, and ground radiation. The net incident angle modifier for all incident radiation is calculated by weighting each component by the corresponding modifier.

$$K_{\tau\alpha,net} = \frac{I_{beam} K_{\tau\alpha,beam} + I_{sky} K_{\tau\alpha,sky} + I_{gnd} K_{\tau\alpha,gnd}}{I_{beam} + I_{sky} + I_{gnd}} \quad (555)$$

For sky and ground radiation the incident angle is approximated using Brandemuehl and Beckman's equations:

$$\theta_{sky} = 59.68 - 0.1388\phi + 0.001497\phi^2 \quad (556)$$

$$\theta_{ground} = 90.0 - 0.5788\phi + 0.002693\phi^2 \quad (557)$$

where ϕ is the surface tilt in degrees.

The net incident angle modifier is then inserted into the useful heat gain equation (548):

$$\frac{q}{A} = F_R \left[I_{solar} K_{\tau\alpha,net} (\tau\alpha)_n - U_L (T_{in} - T_{air}) \right] \quad (558)$$

Equation (549) is also modified accordingly.

$$\eta = F_R K_{\tau\alpha,net} (\tau\alpha)_n - F_R U_L \frac{(T_{in} - T_{air})}{I_{solar}} \quad (559)$$

Outlet Temperature

Outlet temperature is calculated using the useful heat gain q as determined by Equation (558), the inlet fluid temperature T_{in} , and the mass flow rate available from the plant simulation:

$$\frac{q}{A} = \dot{m} c_p (T_{out} - T_{in}) \quad (560)$$

where

\dot{m} = fluid mass flow rate through the collector

c_p = specific heat of the working fluid

Solving for T_{out} ,

$$T_{out} = T_{in} + \frac{q}{\dot{m} c_p A} \quad (561)$$

If there is no flow through the collector, T_{out} is the stagnation temperature of the fluid. This is calculated by setting the left side of Equation (558) to zero and solving for T_{in} (which also equals T_{out} for the no flow case).

References

ASHRAE. 1989. ASHRAE Standard 96-1980 (RA 89): Methods of Testing to Determine the Thermal Performance of Unglazed Flat-Plate Liquid-Type Solar Collectors. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

ASHRAE. 1991. ASHRAE Standard 93-1986 (RA 91): Methods of Testing to Determine the Thermal Performance of Solar Collectors. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

Duffie, J. A., and Beckman, W. A. 1991. Solar Engineering of Thermal Processes, Second Edition. New York: Wiley-Interscience.

Solar Rating and Certification Corporation. 2004. Directory of SRCC Certified Solar Collector Ratings, OG 100. Cocoa, Florida: Solar Rating and Certification Corporation.

Photovoltaic Thermal Flat-Plate Solar Collectors

Photovoltaic-Thermal solar collectors (PVT) combine solar electric cells and thermal working fluid to collect both electricity and heat. Although there are currently comparatively few commercial products, PVT research has been conducted for the past 30 years and many different types of collectors have been studied. Zondag (2008) and Charalambous et. al (2007) provide reviews of the PVT literature. Because PVT is much less commercially-mature, there are no standards or rating systems such as for thermal-only, hot-water collectors. EnergyPlus currently has one simple model based on user-defined efficiencies but a more detailed model based on first-principles and a detailed, layer-by-layer description is under development.

The PVT models reuse the PV models for electrical production. These are described elsewhere in this document in the section on Photovoltaic Arrays-Simple Model

Simple PVT Thermal Model

The input object SolarCollector:FlatPlate:PhotovoltaicThermal provides a simple PVT model that is provided for quick use during design or policy studies. The user simply provides values for a thermal efficiency and the incident solar heats the working fluid. The model also includes a cooling mode for air-based systems where a user-provided surface emittance is used to model cooling of the working fluid to the night sky (water-based cooling will be made available once a chilled water storage tank is available). No other details of the PVT collector's construction are required as input data.

The simple model can heat either air or liquid. If it heats air, then the PVT is part of HVAC air system loop with air nodes connected to an air system. If it heats liquid, then the PVT is part of plant loop with nodes connected to a plant loop and the plant operating scheme determines flows.

Air-system-based PVT modeling include a modulating bypass damper arrangement. Control logic decides if the air should bypass the collector to better meet setpoint. The model requires a drybulb temperature setpoint be placed on the outlet node. The model assume the collector is intended and available for heating when the incident solar is greater than 0.3 W/m^2 and otherwise it is intended for cooling. The inlet temperature is compare to the setpoint on the outlet node to determine if cooling or heating are beneficial. If they are, then the PVT thermal models are applied to condition the air stream. If they are not beneficial, then the PVT is completely bypassed and the inlet node is passed directly to the outlet node to model a completely bypassed damper arrangement. A report variable is available for bypass damper status.

Plant-based PVT do not include a bypass (although one could be used in the plant loop). The collector requests its design flow rate but it otherwise relies on the larger plant loop for control.

When the PVT thermal collector is controlled to be “on,” in heating mode, and working fluid is flowing, the model calculates the outlet temperature based on the inlet temperature and the collected heat using the following equations.

$$Q_{therm} = A_{surf} \cdot f_{activ} \cdot G_T \cdot \eta_{thermal}$$

where,

Q_{therm} is the thermal energy collected [W]

A_{surf} is the net area of the surface [m²]

f_{activ} is the fraction of surface area with active PV/T collector, and

$\eta_{thermal}$ is the thermal conversion efficiency.

$$T_{out} = T_{in} + \frac{Q_{therm}}{\dot{m} c_p}$$

where,

T_{out} is the temperature of the working fluid leaving the PV/T

T_{in} is the temperature of the working fluid entering the PV/T

\dot{m} is the entire mass flow of the working fluid through the PV/T

c_p is the specific heat of the working fluid.

For air-based systems, the value of T_{out} is then compared to the temperature setpoint on the outlet node. If T_{out} exceeds the desired outlet temperature, $T_{set,out}$, then a bypass fraction is calculated to model a modulating bypass damper using:

$$f_{bypass} = \frac{(T_{set,out} - T_{out})}{(T_{in} - T_{out})}$$

When the PVT thermal collector is controlled to be “on,” in cooling mode, and working fluid is flowing, the model calculates the outlet temperature based on the inlet temperature and the heat radiated and convected to the ambient using a heat balance on the outside face of the collector:

$$\dot{m} c_p (T_{in} - T_{out}) = \dot{Q}_{LWR} + \dot{Q}_{conv}$$

Where,

\dot{Q}_{LWR} is the net rate of long wavelength (thermal) radiation exchange with the air, night sky, and ground. See the section “External Longwave Radiation” in the Outside Surface Heat Balance, for full discussion of how this modeled in EnergyPlus using linearized radiation coefficients.

\dot{Q}_{conv} is the net rate of convective flux exchange with outdoor air. See the section “Exterior/External Convection” in the Outside Surface Heat Balance, for full discussion of how this modeled in EnergyPlus. The surface roughness is assumed to be “very smooth.”

The simple model assumes that the effective collector temperature, T_{col} , is the average of the working fluid inlet and outlet temperatures so that we can make the following substitution:

$$T_{out} = 2T_{col} - T_{in}$$

Substituting and solving for T_{col} we obtain the following model for collector temperatures during a (possible) cooling process :

$$T_{col} = \frac{2\dot{m}c_p T_{in} + A_{surf} f_{activ} (h_{r,gnd} T_{gnd} + h_{r,sky} T_{sky} + h_{r,air} T_{air} + h_{c,ext} T_{air})}{2\dot{m}c_p + A_{surf} f_{activ} (h_{r,gnd} + h_{r,sky} + h_{r,air} + h_{c,ext})}$$

Then the outlet temperature can be calculated and heat losses determined. However, the model allows only sensible cooling of the air stream and limits the outlet temperature to not go below the dewpoint temperature of the inlet.

PVT collectors have a design volume flow rate for the working fluid that is autosizable. For air-based systems used as pre-conditioners, the volume flow rate is sized to meet the maximum outdoor air flow rate. For water-based systems on the supply side of a plant loop, each of the PVT collectors are sized to the overall loop flow rate. For water-based systems on the demand side of a plant loop, the collectors are sized using a rule-of-thumb for typical flow rates per unit of collector area. This rule-of-thumb is based on a constant factor of $1.905 \times 10^{-5} \text{ m}^3/\text{s-m}^2$ that was developed by analyzing SRCC data set for conventional solar collectors (see data set SolarCollectors.idf) and averaging the ratio for all 171 different collectors.

References

- Charalambous, P.G., Maidment, G.G., Kalagirou, S.A., and Yiakoumetti, K., Photovoltaic thermal (PV/T) collectors: A review. *Applied Thermal Engineering* 27 (2007) 275-286.
- Zondag, H.A. 2008. Flat-plate PV-Thermal collectors and systems: A review. *Renewable and Sustainable Energy Reviews* 12 (2008) 891-959.

Unglazed Transpired Solar Collectors

The input object SolarCollector:UnglazedTranspired provides a model for transpired collectors that are perhaps one of the most efficient ways to collect solar energy with demonstrated instantaneous efficiencies of over 90% and average efficiencies of over 70%. They are used for preheating outdoor air needed for ventilation and processes such as crop drying.

In EnergyPlus, an unglazed transpired solar collector (UTSC) is modeled as a special component attached to the outside face of a heat transfer surface that is also connected to the outdoor air path. A UTSC affects both the thermal envelope and the HVAC air system. From the air system's point of view, a UTSC is heat exchanger and the modeling needs to determine how much the device raises the temperature of the outdoor air. From the thermal envelope's point of view, the presence of the collector on the outside of the surface modifies the conditions experienced by the underlying heat transfer surfaces. EnergyPlus models building performance throughout the year and the UTSC will often be "off" in terms of forced airflow, but the collector is still present. When the UTSC is "on" there is suction airflow that is assumed to be uniform across the face. When the UTSC is "off" the collector acts as a radiation and convection baffle situated between the exterior environment and the outside face of the underlying heat transfer surface. We distinguish these two modes of operation as *active* or *passive* and model the UTSC component differently depending on which of these modes it is in.

Heat Exchanger Effectiveness

The perforated absorber plate is treated as a heat exchanger and modeled using a traditional effectiveness formulation. The heat exchanger effectiveness, ε_{HX} , is determined from correlations derived from small-scale experiments. Two correlations available in the literature are implemented in EnergyPlus. The first is based on research by Kutscher at the National Renewable Energy Laboratory. The second is based on the research by Van Decker, Hollands, and Brunger at the University of Waterloo. Because both correlations are considered valid, the choice of which correlation to use is left to the user.

Kutscher Correlation

Kutscher's (1994) correlation encompasses surface convection between the collector and the incoming outdoor air stream that occurs on the front face, in the holes, and along the back face of the collector. The correlation uses a Reynolds number based on the hole diameter as a length scale and the mean velocity of air as it passes through the holes as the velocity scale:

$$\text{Re}_D = \frac{V_h D}{\nu} \quad (562)$$

where,

V_h is the velocity through the holes [m/s]

D is the hole diameter [m]

ν is the kinematic viscosity of air [m²/s]

The correlation is a function of Reynolds number, hole geometry, the free stream air velocity, and velocity through the holes:

$$Nu_D = 2.75 \left[\left(\frac{P}{D} \right)^{-1.2} \text{Re}_D^{0.43} + 0.011 \sigma \text{Re}_D \left(\frac{U_\infty}{V_h} \right)^{0.48} \right] \quad (563)$$

where,

P is the pitch, or distance between holes, [m],

D is the diameter of the holes, [m],

σ is the porosity, or area fraction of the holes, [dimensionless],

V_h is the mean velocity of air passing through the holes, [m/s],

U_∞ is the free stream velocity (local wind speed) [m/s].

The Nusselt number is formulated as:

$$Nu_D = \frac{U D}{k} \quad (564)$$

where,

U is the overall heat transfer coefficient based on log mean temperature difference, [W/m²·K], and

k is the thermal conductivity of air [W/m·K].

The heat exchanger effectiveness is:

$$\varepsilon_{HX} = 1 - e^{-\left[\frac{UA}{\dot{m}c_p}\right]} \quad (565)$$

Kutscher's relation was formulated for triangular hole layout, but based on Van Decker et al. (2001) we allow using the correlation for square hole layout and scale P by a factor of 1.6.

Van Decker, Hollands, and Brunger Correlation

Van Decker et. al. extended Kutscher's measurements to include a wider range of collector parameters including plate thickness, pitch, suction velocities, and square hole patterns. Their model formulation differs from Kutscher's in that the model was built up from separate effectiveness models for the front, back, and holes of the collector. Their published correlation is:

$$\varepsilon_{HX} = \left[1 - \left(1 + \text{Re}_s \text{Max} \left(1.733 \text{Re}_w^{-1/2}, 0.02136 \right)^{-1} \right) \right] \times \left[1 - \left(1 + 0.2273 \text{Re}_b^{1/2} \right)^{-1} \right] \times e^{\left(-0.01895 \frac{P}{D} - \frac{20.62 t}{\text{Re}_D D} \right)} \quad (566)$$

where,

$$\text{Re}_s = \frac{V_s P}{\nu}$$

$$\text{Re}_w = \frac{U_\infty P}{\nu}$$

$$\text{Re}_b = \frac{V_h P}{\nu}$$

V_s is the average suction velocity across the front face of the collector [m/s]

t is the collector plate thickness

Heat Exchanger Leaving Temperature

Using either of the correlations above allows determining the heat exchanger effectiveness from known values. By definition the heat exchanger effectiveness is also:

$$\varepsilon_{HX} = \frac{T_{a,HX} - T_{amb}}{T_{s,coll} - T_{amb}} \quad (567)$$

where,

$T_{a,HX}$ is the temperature of the air leaving the collector and entering the plenum [°C]

$T_{s,coll}$ is the temperature of the collector's absorber plate, [°C], and

T_{amb} is the temperature of the ambient outdoor air [°C].

By rewriting equation (567) to solve for $T_{a,HX}$ we see that the temperature of the heated outdoor air entering the plenum can be determined once the collector surface temperature is known,

$$T_{a,HX} = \varepsilon_{HX} T_{s, coll} + (1 - \varepsilon_{HX}) T_{amb} \quad (568)$$

Collector Heat Balance

The collector is assumed to be sufficiently thin and high-conductivity so that it can be modeled using a single temperature (for both sides and along its area). This temperature $T_{s, coll}$ is determined by formulating a heat balance on a control volume that just encapsulates the collector surface. The heat balances are formulated separately for active and passive modes and are diagrammed in the following figure.

Observe that for the passive case, we do not use the heat exchanger relations to directly model the interaction of ventilating air with the collector. This is because these relations are considered to not apply when the UTSC is in passive mode. They were developed for unidirectional flow (rather than the balanced-in-and-out flow expected from natural forces) and for specific ranges of suction face velocity. Therefore, this heat transfer mechanism is handled using classical surface convection models (as if the collector was not perforated). (Air exchanges are modeled as ventilation in the plenum air heat balance but do not interact with the hole edges in the collector surface.)

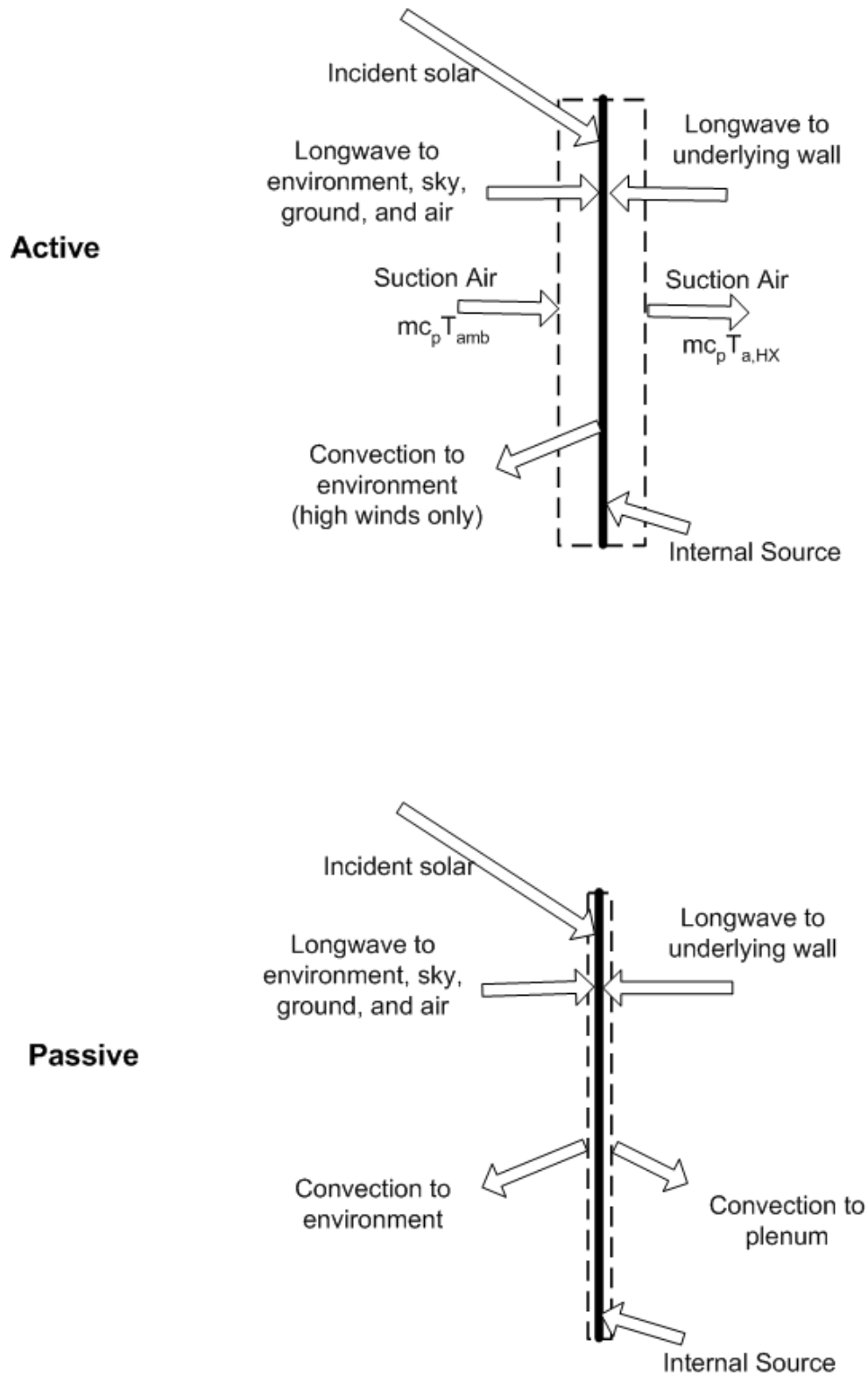


Figure 218. Transpired Collector Heat Balance

When the UTSC is active, the heat balance on the collector surface control volume is:

$$q''_{\alpha sol} + q''_{LWR,Env} + q''_{conv,wind} - q''_{HX} + q''_{LWR,plen} + q''_{source} = 0 \quad (569)$$

where:

$q''_{\alpha sol}$ is absorbed direct and diffuse solar (short wavelength) radiation heat flux.

$q''_{LWR,Env}$ is net long wavelength (thermal) radiation flux exchange with the air and surroundings.

$q''_{conv,wind}$ is surface convection flux exchange with outdoor air under high wind and rain conditions. Note that this term is usually assumed to be zero in UTSC model development but we add the term to allow for deteriorated performance of the UTSC under poor conditions.

q''_{HX} is heat exchanger flux from collector to incoming outdoor air.

$q''_{LWR,plen}$ is net long wavelength (thermal) radiation flux exchange with the outside face of the underlying surface(s).

q''_{source} is a source/sink term that accounts for energy exported out of the control volume when the collector's absorber plate is a hybrid device such as a photovoltaic panel.

While the heat balance on the passive collector surface control volume is:

$$q''_{\alpha sol} + q''_{LWR,Env} + q''_{conv,Env} + q''_{LWR,plen} + q''_{conv,plen} + q''_{source} = 0 \quad (570)$$

where:

$q''_{conv,Env}$ = surface convection flux exchange with outdoor air.

$q''_{conv,plen}$ = surface convection flux exchange with plenum air.

All terms are positive for net flux to the collector except the heat exchanger term, which is taken to be positive in the direction from collector to incoming air stream. Each of these heat balance components is introduced briefly below.

External SW Radiation

$q''_{\alpha sol}$ is calculated using procedures presented elsewhere in this manual and includes both direct and diffuse incident solar radiation absorbed by the surface face. This is influenced by location, surface facing angle and tilt, shading surfaces, surface face material properties, weather conditions, etc.

External LW Radiation

$q''_{LWR,Env}$ is a standard radiation exchange formulation between the surface, the sky, the ground, and the atmosphere. The radiation heat flux is calculated from the surface absorptivity, surface temperature, sky, air, and ground temperatures, and sky and ground view factors. Radiation is modeled using linearized coefficients.

External Convection

$q''_{conv,Env}$ is modeled using the classical formulation: $q''_{conv} = h_{co}(T_{air} - T_o)$ where h_{co} , is the convection coefficient. This coefficient will differ depending on whether or not the UTSC is active or passive. When the UTSC is passive, h_{co} is treated in the same way as an outside face with ExteriorEnvironment conditions. When the UTSC is active, the special suction airflow situation of a transpired collector during operation means that h_{co} is often zero because the suction situation can eliminate mass transport away from the collector. However when the winds are high, the strong turbulence and highly varying pressures can cause the suction flow situation to breakdown. Therefore, we include the $q''_{conv,wind}$ term in the heat balance and use a special coefficient $h_{c,wind}$ to model this lost heat transfer. In addition, when it is raining outside, we assume the collector gets wet and model the enhanced surface heat transfer using a large value for $h_{c,wind}$.

Heat Exchanger

q''_{HX} is modeled using the classical formulation $q''_{HX} = \frac{\dot{m}c_p(T_{a,HX} - T_{amb})}{A}$ where $T_{a,HX}$ is determined using correlations described above. When the UTSC is active, the air mass flow is determined from the operation of the outdoor air mixer component. When the UTSC is off, this term is zero.

Plenum LW Radation

$q''_{LWR,plen}$ is a standard radiation exchange formulation between the collector surface and the underlying heat transfer surface located across the plenum. Radiation is modeled using linearized coefficients.

Plenum Convection

$q''_{conv,plen}$ is modeled using the classical formulation: $q''_{conv} = h_{cp}(T_{air} - T_o)$ where h_{cp} , is the convection coefficient. This coefficient is taken as zero when the UTSC is operating because of the suction airflow situation. When the UTSC is off, the value for h_{cp} is obtained from correlations used for window gaps from ISO (2003) standard 15099.

Substituting models into (569) and solving for $T_{s,coll}$ yields the following equation when the UTSC is active ("on"):

$$T_{s,coll} = \frac{\left(I_s \alpha + h_{r,atm} T_{amb} + h_{r,sky} T_{sky} + h_{r,gnd} T_{amb} + h_{r,plen} T_{so} + h_{c,wind} T_{amb} + \frac{\dot{m}c_p}{A} T_{amb} - \frac{\dot{m}c_p}{A} (1 - \varepsilon_{HX}) T_{amb} + q''_{source} \right)}{\left(h_{r,atm} + h_{r,sky} + h_{r,gnd} + h_{r,plen} + h_{c,wind} + \frac{\dot{m}c_p}{A} \varepsilon_{HX} \right)} \quad (571)$$

and substituting into (126) yields the following equation when the UTSC is passive ("off"):

$$T_{s,coll} = \frac{\left(I_s \alpha + h_{co} T_{amb} + h_{r,atm} T_{amb} + h_{r,sky} T_{sky} + h_{r,gnd} T_{amb} + h_{r,plen} T_{so} + h_{c,plen} T_{a,plen} + q''_{source} \right)}{\left(h_{co} + h_{r,air} + h_{r,sky} + h_{r,gnd} + h_{r,plen} + h_{c,plen} \right)} \quad (572)$$

where,

- I_s is the incident solar radiation of all types [W/m^2],
- α is the solar absorptivity of the collector [dimensionless],
- $h_{r,atm}$ is the linearized radiation coefficient for the surrounding atmosphere [$W/m^2 \cdot K$],
- T_{amb} is the outdoor drybulb from the weather data, also assumed for ground surface [$^{\circ}C$],
- $h_{r,sky}$ is the linearized radiation coefficient for the sky [$W/m^2 \cdot K$],
- T_{sky} is the effective sky temperature [$^{\circ}C$],
- $h_{r,grnd}$ is the linearized radiation coefficient for the ground [$W/m^2 \cdot K$],
- $h_{r,plen}$ is the linearized radiation coefficient for the underlying surface [$W/m^2 \cdot K$],
- $h_{c,wind}$ is the convection coefficient for the outdoor environment when the UTSC is active and winds are high or it is raining [$W/m^2 \cdot K$],
- T_{so} is the temperature of the outside face of the underlying heat transfer surface [$^{\circ}C$],
- \dot{m} is the air mass flow rate when in active mode [kg/s],
- c_p is the specific heat of air at constant pressure [J/kg·K],
- A is the area of the collector [m^2],
- h_{co} is the convection coefficient for the outdoor environment [$W/m^2 \cdot K$],
- $h_{c,plen}$ is the convection coefficient for the surfaces facing the plenum [$W/m^2 \cdot K$], and
- $T_{a,plen}$ is the air drybulb for air in the plenum and entering the outdoor air system [$^{\circ}C$].

Plenum Heat Balance

The *plenum* is the volume of air located between the collector and the underlying heat transfer surface. The plenum air is modeled as well-mixed. The uniform temperature of the plenum air, $T_{a,plen}$, is determined by formulating a heat balance on a control volume of air as diagrammed below.

Note that we have formulated the control volumes with slight differences for the active and passive cases. For the active case, the suction air situation and heat exchanger effectiveness formulations dictate that the collector surface control volume encompass part of the air adjacent to both the front and back surfaces of the collector. However for the passive case, the collector surface control volume has no air in it and the plenum air control volume extends all the way to the surface of the collector.

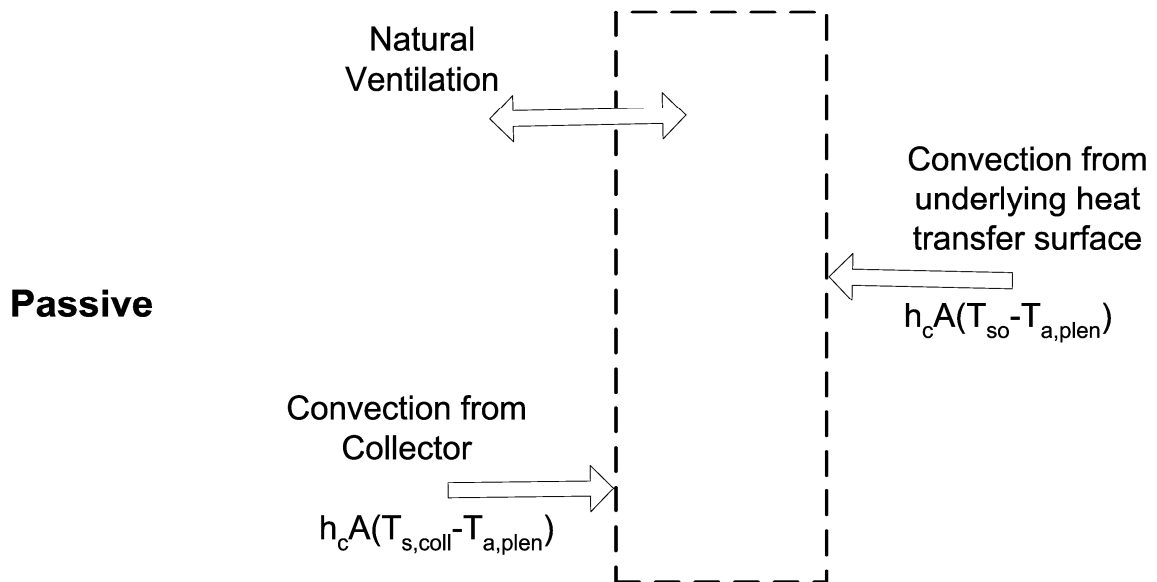
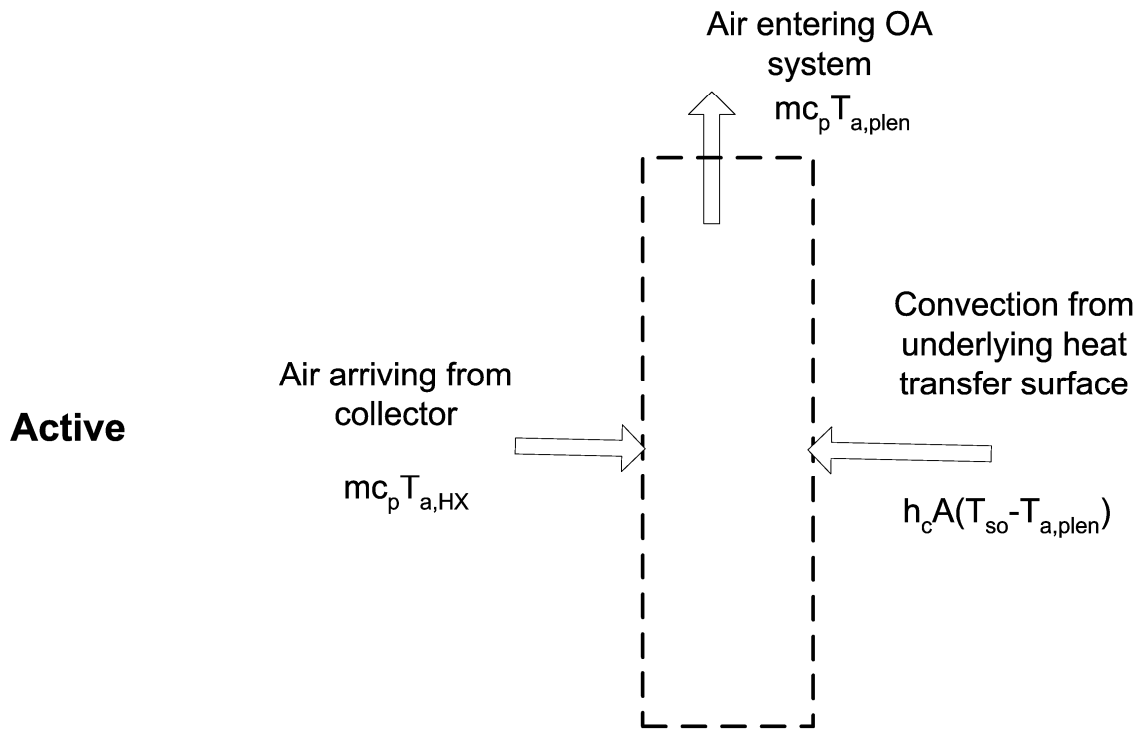


Figure 219. Transpired Collector Plenum Air Heat Balance

When the UTSC is active, the heat balance on the plenum air control volume is:

$$\dot{Q}_{air} + \dot{Q}_{co} = 0 \quad (573)$$

where,

\dot{Q}_{air} is the net rate of energy added by suction air convecting through the control volume.

\dot{Q}_{co} is the net rate of energy added by surface convection heat transfer with the underlying surface.

When the UTSC is passive, the heat balance on the plenum air control volume is:

$$\dot{Q}_{vent} + \dot{Q}_{co} + \dot{Q}_{c, coll} = 0 \quad (574)$$

where,

\dot{Q}_{vent} is the net rate of energy added from infiltration – where outdoor ambient air exchanges with the plenum air.

$\dot{Q}_{c, coll}$ is the net rate of energy added by surface convection heat transfer with the collector.

Substituting into (573) and solving for $T_{a, plen}$ yields the following equation for when the UTSC is active:

$$T_{a, plen} = \frac{(\dot{m}c_p T_{a, HX} + h_{c, plen} A T_{so})}{(\dot{m}c_p + h_{c, plen} A)} \quad (575)$$

And substituting into (128) yields the following equation when the UTSC is passive:

$$T_{a, plen} = \frac{(h_{c, plen} A T_{so} + \dot{m}_{vent} c_p T_{amb} + h_{c, plen} A T_{s, coll})}{(h_{c, plen} A + \dot{m}_{vent} c_p + h_{c, plen} A)}$$

where,

\dot{m}_{vent} is the air mass flow from natural forces [kg/s]

The literature on UTSC does not appear to address the passive mode of operation and no models for \dot{m}_{vent} have been identified. Nevertheless, natural buoyancy and wind forces are expected to drive air exchange between the plenum and ambient and some method of modeling \dot{m}_{vent} is needed. Reasoning that the configuration is similar to single-side natural ventilation, we elect to use correlations for natural ventilation presented in Chapter 26. of ASHRAE HOF (2001).

$$\dot{m}_{vent} = \rho \dot{V}_{tot}$$

where,

ρ is the density of air [kg/m^3], and

$\dot{V}_{\text{tot}} = \dot{V}_{\text{wind}} + \dot{V}_{\text{thermal}}$ is the total volumetric flow rate of air ventilating in and out of the plenum.

$$\dot{V}_{\text{wind}} = C_v A_{in} U_{\infty}$$

$$\dot{V}_{\text{thermal}} = C_D A_{in} \sqrt{2g\Delta H_{NPL} (T_{a,plen} - T_{amb}) / T_{a,plen}} \quad (\text{if } T_{a,plen} > T_{amb})$$

$$\dot{V}_{\text{thermal}} = C_D A_{in} \sqrt{2g\Delta H_{NPL} (T_{amb} - T_{a,plen}) / T_{amb}} \quad (\text{if } T_{amb} > T_{a,plen} \text{ and UTSC vertical})$$

C_v is the effectiveness of the openings that depends on opening geometry and the orientation with respect to the wind. ASHRAE HoF (2001) indicates values ranging from 0.25 to 0.6. In the UTSC model, this value is available for user input and defaulted to 0.25.

C_D is the discharge coefficient for the opening and depends on opening geometry. In the UTSC model, this value is available for user input and defaulted to 0.65.

Mass continuity arguments lead to modeling the area of the openings as one half of the total area of the holes, so we have:

$$A_{in} = \frac{A\sigma}{2}$$

g is the gravitational constant taken as $9.81 \text{ [m/s}^2\text{]}$.

ΔH_{NPL} is the height from midpoint of lower opening to the Neutral Pressure Level. This is taken as one-fourth the overall height of the UTSC if it is mounted vertically. For tilted collectors, the nominal height is modified by the sine of the tilt. If the UTSC is mounted horizontally (e.g. on the roof) then the ΔH_{NPL} is taken as the gap thickness of the plenum.

If the UTSC is horizontal and $T_{amb} > T_{a,plen}$ then $\dot{V}_{\text{thermal}} = 0$ because this is a stable situation.

Underlying Heat Transfer Surface

The UTSC is applied to the outside of a heat transfer surface. This surface is modeled using the usual EnergyPlus methods for handling heat capacity and transients – typically the CTF method. These native EnergyPlus Heat Balance routines are used to calculate T_{so} . The UTSC model is coupled to the underlying surface using the OtherSideConditionsModel mechanism. The UTSC model provides values for $h_{r,plen}$, $T_{s,coll}$, $h_{c,plen}$, and $T_{a,plen}$ for use with the Heat Balance Model calculations for the outside face of the underlying surface (described elsewhere in this manual).

Solar and Shading Calculations

The transpired collector object uses a standard EnergyPlus surface in order to take advantage of the detailed solar and shading calculations. Solar radiation incident on the

surface includes beam and diffuse radiation, as well as radiation reflected from the ground and adjacent surfaces. Shading of the collector by other surfaces, such as nearby buildings or trees, is also taken into account.

Local Wind Speed Calculations

The outdoor wind speed affects terms used in modeling UTSC components. The wind speed in the weather file is assumed to be measured at a meteorological station located in an open field at a height of 10 m. To adjust for different terrain at the building site and differences in the height of building surfaces, the local wind speed is calculated for each surface.

The wind speed is modified from the measured meteorological wind speed by the equation (ASHRAE 2001):

$$U_{\infty} = V_{met} \left(\frac{\delta_{met}}{z_{met}} \right)^{a_{met}} \left(\frac{z}{\delta} \right)^a \quad (576)$$

where z is the height of the centroid of the UTSC, z_{met} is the height of the standard meteorological wind speed measurement, and a and δ are terrain-dependent coefficients. δ is the boundary layer thickness for the given terrain type. The values of a and δ are shown in the following tables:

Table 49. Terrain-Dependent Coefficients (ASHRAE 2001).

Terrain	Description	Exponent, a	Layer Thickness, δ (m)
1	Flat, open country	0.14	270
2	Rough, wooded country	0.22	370
3	Towns and cities	0.33	460
4	Ocean	0.10	210
5	Urban, industrial, forest	0.22	370

The UTSC can be defined such that it has multiple underlying heat transfer surfaces. The centroid heights for each surface are area-weighted to determine the average height for use in the local wind calculation.

Convection Coefficients

UTSC modeling requires calculating up to three different coefficients for surface convection heat transfer. These coefficients are defined in the classic way by:

$$h_c = \frac{T_{air} - T_{surf}}{q_{conv}''}$$

First, h_{c_o} is the convection coefficient for the collector surface facing the outdoors when the UTSC is passive. It is modeled in exactly the same way as elsewhere in EnergyPlus and will depend on the user setting for Outside Convection Algorithm – Outside Surface Heat Balance entry elsewhere in this document.

Second, $h_{c,plen}$ is the convection coefficient for surfaces facing the plenum. This coefficient is applied to just the underlying surface's convection when the UTSC is active and to both the collector and the underlying surface when the UTSC is passive. When the UTSC is active,

we use the convection correlation for forced air developed by McAdams (1954) as published by ASHRAE HoF (2001):

$$h_{c,plen} = 5.62 + 3.9V_p \quad (577)$$

where,

V_p is the mean velocity in the plenum determined from $V_p = \frac{\dot{m}}{2\rho A_p}$ where A_p is the

effective cross section area of the plenum perpendicular to the primary flow direction. When the UTSC is passive, we model the convection in the same way used in EnergyPlus to model air gaps in windows. These correlations vary by Rayleigh number and surface tilt and are based on the work of various research including Hollands et. al., Elsherbiny et. al., Wright, and Arnold. The formulations are documented in ISO (2003) standard 15099. For the UTSC implementation, the routines were adapted from Subroutine NusseltNumber in WindowManager.f90 (by F. Winkelmann), which itself was derived from Window5 subroutine "nusselt".

Third, $h_{c,wind}$ is the convection coefficient used to degrade the UTSC performance under environmental conditions with high wind or rain. If the weather file indicates it is raining, then we set $h_{c,wind} = 1000.0$ which has the effect of making the collector the same temperature as ambient air. The heat exchanger effectiveness correlations described above account for a moderate amount of wind, but the correlations appear confined to the range 0 to 5.0 m/s. Therefore we set $h_{c,wind}$ equal to zero if U_∞ is ≤ 5.0 m/s. If U_∞ is > 5.0 m/s then we use the McAdams correlation (577) but with a reduced velocity magnitude:

$$h_{c,wind} = 5.62 + 3.9(U_\infty - 5.0)$$

Radiation Coefficients

UTSC modeling requires calculating up to four different linearized coefficients for radiation heat transfer. Whereas radiation calculations usually use temperature raised to the fourth power, this greatly complicates solving heat balance equations for a single temperature. Linearized radiation coefficients have the same units and are used in the same manner as surface convection coefficients and introduce very little error for the temperature levels involved.

The radiation coefficient, $h_{r,plen}$, is used to model thermal radiation between the collector surface and the outside face of the underlying heat transfer surface. We assume a view factor of unity. It is calculated using:

$$h_{r,plen} = \sigma_{SB} e_{coll} e_{so} \frac{(T_{s,coll}^4 - T_{so}^4)}{(T_{s,coll} - T_{so})} \quad (578)$$

where,

all temperatures are converted to Kelvin,

σ_{SB} is the Stefan-Boltzmann constant,

e_{coll} is the longwave thermal emittance of the collector, and

e_{so} is the longwave thermal emittance of the underlying heat transfer surface.

The three other coefficients, $h_{r,atm}$, $h_{r,sky}$, and $h_{r,gnd}$ are used elsewhere in EnergyPlus for the outside face surface heat balance and are calculated in the same manner as equation (130) for UTSC collectors. [This is accomplished by calling subroutine InitExteriorConvectionCoeffs in the file HeatBalanceConvectionCoeffs.f90.]

Bypass Control

The UTSC is assumed to be arranged so that a bypass damper controls whether or not air is drawn directly from the outdoors or through the UTSC. The control decision is based on whether or not it will be beneficial to heat the outdoor air. There are multiple levels of control including an availability schedule, whether or not the outdoor air is cooler than the mixed air setpoint, or whether or not the zone air temperature is lower than a so-called free heating setpoint.

Sizing Warnings

Although the design of the transpired collector is left to the user, the program issues warnings when the suction airflow velocity falls outside the range 0.003 to 0.08 m/s.

Overall Efficiency

The overall thermal efficiency of the UTSC is a useful output report and is defined as the ratio of the useful heat gain of the entire system versus the total incident solar radiation on the gross surface area of the collector.

$$\eta = \frac{(\dot{Q}/A)}{I_{sc}} = \frac{\dot{m} c_p (T_{a,plen} - T_{amb})}{I_{sc} A} \quad (579)$$

where

\dot{Q} is useful heat gain

I_{sc} is total incident solar radiation

Note that the efficiency η is only defined for $I_{solar} > 0$. This efficiency includes heat recovered from the underlying wall and can exceed 1.0.

Collector Efficiency

The thermal efficiency of the collector is a useful output report and is defined as the ratio of the useful heat gain of the collector fluid versus the total incident solar radiation on the gross surface area of the collector.

$$\eta = \frac{\dot{m} c_p (T_{a,HX} - T_{amb})}{I_{sc} A} \quad (580)$$

Note that the efficiency η is only defined for $I_{solar} > 0$

References

Kutscher, C.F. 1994. Heat exchange effectiveness and pressure drop for air flow through perforated plates with and without crosswind. *Journal of Heat Transfer*. May 1994, Vol. 116, p. 391. American Society of Mechanical Engineers.

Van Decker, G.W.E., K.G.T. Hollands, and A.P. Brunger. 2001. Heat-exchange relations for unglazed transpired solar collectors with circular holes on a square or triangular pitch. *Solar Energy*. Vol. 71, No. 1. pp 33-45, 2001.

ISO. 2003. ISO 15099:2003. Thermal performance of windows, doors, and shading devices – Detailed calculations. International Organization for Standardization.

System Availability Managers

Overview

System availability managers, or *AvailabilityManagers*, are one of the high-level control constructs in EnergyPlus. An *AvailabilityManager* is able to access data from any of the HVAC system nodes and use this data to set flags telling a central air system or plant loop to turn on or off. Each type of availability manager encapsulates a specific high-level on-off control strategy. For instance, the input object *AvailabilityManager:NightVentilation* allows the user to specify a control strategy for pre-cooling the building with night-time ventilation.

Availability managers are contained in the program module *SystemAvailabilityManager*. The Availability managers are executed at the start of each HVAC time step by a call to *ManageSystemAvailability*, and they reside outside the HVAC system iteration loops. Thus, the availability managers are executed once per HVAC time step, and they use previous time step information to calculate their control actions.

A given instance of an *AvailabilityManager* is specific to an *AirLoopHVAC* or to a *PlantLoop*. In the input each loop references its availability managers through a *AvailabilityManagerAssignmentList*. Once the data has been read in and processed, the *AvailabilityManager* data for each loop is stored in array *PriAirSysAvailMgr* in *DataAirLoop* (for *AirLoopHVAC*) and in array *PlantAvailMgr* in *DataPlant* for plant loops.

The availability status for each *AirLoopHVAC* (*PriAirSysAvailMgr*) is also passed to a zone equipment availability status array (*ZoneEquipAvail*) for possible use by the zones served by that air loop. This sets the availability for certain zone equipment which are assumed to operate in conjunction with the air loop. Specifically, the availability of zone exhaust fans (Ref. Fan:ZoneExhaust) and terminal unit fans (Ref. AirTerminal:SingleDuct:SeriesPIU:Reheat, AirTerminal:SingleDuct:ParallelPIU:Reheat, and AirTerminal:SingleDuct:VAV:Reheat:VariableSpeedFan), if specified in the input, will be the same as the corresponding *AirLoopHVAC* availability if an *AvailabilityManager* is specified. Other zone equipment are not affected by the *AirLoopHVAC* system availability managers (e.g., Window AC, PTAC and PTHP).

The actual action of turning on or off a loop is taken by the loop prime movers: fans for *AirLoopHVACs* and pumps for plant loops. For instance when a fan is deciding whether it is on or off, it checks its on/off schedule and whether the loop availability status flag is *CycleOn* or *ForceOff*. If the schedule is on and the status flag is *ForceOff*, the fan will be off. If the fan schedule says off and the status flag is *CycleOn*, the fan will be on. Thus the availability managers overrule the fan on/off schedule.

Scheduled

The input object *AvailabilityManager:Scheduled* provides the simplest availability manager. The sole input for this manager is a schedule name. If the current schedule value is > 0.0, the availability status flag is set to *CycleOn*; otherwise it is set to *ForceOff*.

Scheduled On

An alternative to the *AvailabilityManager:Scheduled* object is the *AvailabilityManager:ScheduledOn* that is used specifically to turn equipment on while other availability managers may be used to turn the equipment off. The sole input for this manager is a schedule name. If the current schedule value is > 0.0, the availability status flag is set to *CycleOn*; otherwise it is set to *NoAction*.

Scheduled Off

An alternative to the `AvailabilityManager:Scheduled` object is the `AvailabilityManager:ScheduledOff` that is used specifically to turn equipment off while other availability managers may be used to turn the equipment on. The sole input for this manager is a schedule name. If the current schedule value is = 0.0, the availability status flag is set to *ForceOff*; otherwise it is set to *NoAction*.

Night Cycle

The input object `AvailabilityManager:NightCycle` is to specify when a system that has been scheduled off during unoccupied hours should cycle on to prevent building temperatures from becoming too hot or too cold. This manager can not be used to force a system off. The inputs are:

- A manager applicability schedule;
- The name of the fan schedule this manager will override;
- The control type: *Stay Off*, *Cycle On Any*, *Cycle On Control Zone*, or *Cycle On Any - Zone Fans Only*;
- Thermostat on/off tolerance T_{tol} (degrees C);
- Cycling run time in seconds; used to calculate a stop time (in time steps since the start of the current run period) once the status has become *Cycle On*.

If the fan schedule current value is > 0 or the applicability schedule value is ≤ 0 or if the program is in warmup, *AvailStatus* = *NoAction*.

Otherwise,

- if current time (in time steps since the start of the run period) is greater than the start time and less than the stop time, *AvailStatus* = *CycleOn* (or *CycleOnZoneFansOnly* if the control type is *Cycle On Any – Zone Fans only*).
- If the current time equals the stop time, *AvailStatus* = *NoAction* and the fan schedule will determine if the system is on.
- If the current time is greater than the stop time, the manager can potentially cycle the system on.
 - For control types *Cycle On Any* and *Cycle On Any – Zone Fans Only* the manager looks at each zone served by the air loop and detects whether the zone temperature at the thermostat is greater than the cooling setpoint plus T_{tol} or less than the heating setpoint minus T_{tol} . If it is, *AvailStatus* is set to *CycleOn* (or *CycleOnZoneFansOnly*). If not, *AvailStatus* is set to *NoAction*.
 - For control type *Cycle On Control Zone*, the manager does the same check just for the control zone.

Lastly if the new status is *CycleOn* the start time is reset to the current time and the stop time is reset.

Night Ventilation

The input object `AvailabilityManager:NightVentilation` is a manager that looks at indoor and outdoor conditions to see if night ventilation might be beneficial. In addition to being able to cycle the air system on, this manager can specify what the ventilation air flow rate should be. The inputs are:

- A manager applicability schedule;
- The name of the fan schedule this manager will override;
- A ventilation temperature schedule;
- A ventilation delta T;

- A ventilation low limit temperature;
- The night ventilation flow fraction;
- The name of the control zone.

If the fan schedule current value is > 0 or the applicability schedule value is ≤ 0 or if the program is in warmup, *AvailStatus* = *NoAction*.

Otherwise, the manager performs 3 limit checks; if all are true, the status will be set to *CycleOn*. Otherwise there is *NoAction*.

1. If for all the zones connected to the air loop the room temperature at the thermostat is greater than the ventilation temperature schedule value, this check is true.
2. If for all the zones connected to the air loop the room temperature at the thermostat is less than the ventilation low limit temperature, this check is true.
3. If the difference between the control zone room temperature at the thermostat and the outside temperature is greater than the specified night venting delta T, this check is true.

If the status is *CycleOn* this manager sets some additional values in the *AirLoopControlInfo* data structure (a flag is set to indicate night ventilation is occurring and a flag is set to indicate that the air system flow rate has been specified by a manager) and in the *AirLoopFlow* data structure (the system flow fraction is specified). The night ventilation flag indicates to the fan that it should use alternate performance characteristics if they have been specified in a Fan:Night Vent Performance object.

Differential Thermostat

The input object AvailabilityManager:DifferentialThermostat is a manager that overrides fan or pump schedules depending on the temperature difference between two nodes. A typical use would be for one node to be an outdoor air node and the other a zone node. The inputs are:

- name of the hot node;
- name of the cold node;
- temperature difference for turning the system on (ΔT_{on});
- temperature difference for turning the system off (ΔT_{off}).

Note that there is no applicability schedule for this manager. Also, this manager always returns a status of *CycleOn* or *ForceOff*, never *NoAction*.

$$\Delta T = T_{hot\ node} - T_{cold\ node}$$

If $\Delta T \geq \Delta T_{on}$ then

$$AvailStatus = CycleOn$$

Else if $\Delta T \leq \Delta T_{off}$ then

$$AvailStatus = ForceOff$$

Else

AvailStatus remains in its previous state.

High Temperature Turn Off

The input object AvailabilityManager:HighTemperatureTurnOff is used to turn off a central air system or a plant loop if a sensed node temperature exceeds a temperature limit. The inputs are:

- name of the sensed node;
- limit temperature (T_u).

If $T_{sensed\ node} \geq T_u$ then

$$AvailStatus = ForceOff$$

Else

AvailStatus = NoAction

High Temperature Turn On

The input object AvailabilityManager:HighTemperatureTurnOn is used to turn on a central air system or a plant loop if a sensed node temperature exceeds a temperature limit. The inputs are:

- name of the sensed node;
- limit temperature (T_u).

If $T_{sensed\ node} \geq T_u$ then

AvailStatus = CycleOn

Else

AvailStatus = NoAction

Low Temperature Turn Off

The input object AvailabilityManager:LowTemperatureTurnOff is used to turn off a central air system or a plant loop if a sensed node temperature is lower than a temperature limit. The inputs are:

- name of the sensed node;
- limit temperature (T_l);
- applicability schedule name.

If the applicability schedule exists and has a current value of ≤ 0 , the manager returns an availability status of *NoAction*.

Otherwise,

If $T_{sensed\ node} \leq T_l$ then

AvailStatus = ForceOff

Else

AvailStatus = NoAction

Low Temperature Turn On

The input object AvailabilityManager:LowTemperatureTurnOn is used to turn on a central air system or a plant loop if a sensed node temperature is less than a temperature limit. The inputs are:

- name of the sensed node;
- limit temperature (T_l).

If $T_{sensed\ node} \leq T_l$ then

AvailStatus = CycleOn

Else

AvailStatus = NoAction

Hybrid Ventilation Control

The input object AvailabilityManager:HybridVentilation serves two purposes: 1) it prevents simultaneous natural ventilation and HVAC system operation, and 2) it allows users to examine various strategies to maximize natural ventilation in order to reduce heating/cooling loads. This availability manager works with either the AirflowNetwork model or the simple airflow objects to provide controlled natural ventilation. The controlled natural ventilation objects referred to here are either AirflowNetwork:Multizone:ComponentDetailedOpening and AirflowNetwork:Multizone:ComponentSimpleOpening objects, or ZoneVentilation and ZoneMixing objects. The simple airflow objects are not allowed to work with the

AirflowNetwork objects simultaneously. This availability manager performs somewhat differently from the other availability managers:

- This availability manager is called before the program estimates (predicts) the cooling or heating load that needs to be met by the cooling/heating systems for the simulation time step. On the other hand, the other availability managers are called after the system cooling/heating load prediction. The main reason for calling AvailabilityManager:HybridVentilation early is that this manager determines whether natural ventilation is allowed or not, and the loads from natural ventilation are used to predict system loads.
- This availability manager has its own control schedule and does not work in tandem with AvailabilityManager:Scheduled.
- AvailabilityManager:HybridVentilation works completely independent of the other system availability managers, so this manager is not a valid system availability manager type in the AvailabilityManagerAssignmentList object.

Control logic

The control logic for each ventilation control mode is described below.

Temperature control

This control mode checks whether the outdoor air dry-bulb temperature is between the Minimum Outdoor Temperature and Maximum Outdoor Temperature specified. If the outdoor temperature is between the two values then natural ventilation is allowed, else natural ventilation is not allowed.

When natural ventilation is allowed, the control then checks the temperature difference between the zone temperature and the temperature setpoint(s) in the controlled zone based on the specified temperature control type (four available temperature control types) to make a final decision:

Single Heating Setpoint:

If the zone temperature is below the setpoint, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid overcooling a space, which could result in additional heating load.

Single Cooling Setpoint:

If the zone temperature is above the setpoint, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid overheating a space, which could result in additional cooling load.

Single Heating Cooling Setpoint:

Since this temperature control type requires only a single setpoint, natural ventilation is not allowed. A recurring warning message is issued.

Dual Setpoint with DeadBand:

If the zone temperature is beyond the deadband, the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid either overcooling a space, which could result in additional heating load when the zone temperature is below the heating setpoint, or overheating a space, which could result in additional cooling load when the zone temperature is above the cooling setpoint.

Enthalpy control

This control mode checks whether the outdoor air enthalpy is between the Minimum Outdoor Enthalpy and Maximum Outdoor Enthalpy specified. If the outdoor enthalpy is between the two values then natural ventilation is allowed, else natural ventilation is not allowed.

When natural ventilation is allowed, the control then checks the temperature difference between the zone temperature and the temperature setpoint(s) in the controlled zone based

on the specific temperature control type to make a final decision. This procedure is the same as defined above for the temperature control mode.

Single Heating Setpoint:

If the zone temperature is below the setpoint, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid overcooling a space, which could result in additional heating load.

Single Cooling Setpoint:

If the zone temperature is above the setpoint, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid overheating a space, which could result in additional cooling load.

Single Heating Cooling Setpoint:

Since this temperature control type requires only a single setpoint, natural ventilation is not allowed. A recurring warning message is issued.

Dual Setpoint with DeadBand:

If the zone temperature is beyond the deadband, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid either overcooling a space, which could result in additional heating load when the zone temperature is below the heating setpoint, or overheating a space, which could result in additional cooling load when the zone temperature is above the cooling setpoint.

Dew-Point control

This control mode checks whether the outdoor air dewpoint temperature is between the Minimum Outdoor Dew-Point and Maximum Outdoor Dew-Point specified. If the outdoor dewpoint temperature is between the two values then natural ventilation is allowed, else natural ventilation is not allowed.

When natural ventilation is allowed and a humidistat is available in the controlled zone, the control then checks the humidity ratio difference between the zone air humidity ratio and humidistat setpoint in the controlled zone to make a final decision. It should be pointed out that the humidistat object provides the setpoint of relative humidity, the value of relative humidity has to be converted into the setpoint of humidity ratio using the actual zone air temperature for comparison. Since the humidistat only provides a single value for relative humidity, there are two possible scenarios:

If the actual zone air humidity ratio is below the humidity ratio setpoint and the controlled zone needs humidification as the first scenario, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid dehumidifying a space, which could result in additional humidification load.

If the actual zone air humidity ratio is above the humidity ratio setpoint and the controlled zone needs dehumidification as the second scenario, then the initial decision is overridden and natural ventilation is not allowed. This is intended to avoid humidifying a space, which could result in additional dehumidification load.

Outdoor ventilation air control

This control mode does not require any additional checks. The control only works with Airflow Network opening objects only, and is not allowed to work with ZoneVentilation and ZoneMixing objects.

Availability Status

After the hybrid ventilation control determines whether natural ventilation is allowed or not, the availability status flag is set as follows:

If *Natural Ventilation* then

AvailStatus = CycleOn

Else

AvailStatus = ForceOn

Occupant Thermal Comfort

The integration of a sophisticated building thermal analysis tool with thermal comfort models allows one to perform an energy analysis on a zone and simultaneously determine if the environmental control strategy will be sufficient for the occupants to be thermally comfortable. This chapter is intended to provide background on thermal comfort, present an overview of state of the art thermal comfort models and present the mathematical models that have been incorporated into Energy Plus. Thermal comfort modeling is controlled primarily by the People input object.

Background on Thermal Comfort Models

Table 50. General Nomenclature list for Thermal Comfort Models

Mathematical variable	Description	Units	Range	FORTTRAN variable
A_{Du}	Dubois body surface area	m^2	-	-
H	Internal heat production rate of an occupant per unit area = M – W	W/m^2	-	IntHeatProd
I_{cl}	Thermal resistance of the clothing	clo	-	CloUnit
M	Metabolic rate per unit area	W/m^2	-	ActLevel
P_a	Water vapor pressure in ambient air	Torr	-	VapPress
T_a	Air temperature	$^{\circ}C$	-	AirTemp
T_{cr}	Core or internal temperature	$^{\circ}C$	-	CoreTemp
T_r	Mean radiant temperature	$^{\circ}C$	-	RadTemp
T_{sk}	Skin temperature	$^{\circ}C$	-	-
v	Relative air velocity	m/s	-	AirVel
W	The rate of heat loss due to the performance of work	W/m^2	-	WorkEff
w	Skin wettedness	-	-	-

Throughout the last few decades, researchers have been exploring the thermal, physiological and psychological response of people in their environment in order to develop mathematical models to predict these responses. Researchers have empirically debated building occupants' thermal responses to the combined thermal effect of the personal, environmental and physiological variables that influence the condition of thermal comfort.

There are two personal variables that influence the condition of thermal comfort: the thermal resistance of the clothing (I_{cl}), and the metabolic rate (H/A_{Du}). The thermal resistance of the clothing (I_{cl}) is measured in units of "clo." The 1985 ASHRAE Handbook of Fundamentals

(ASHRAE 1985) suggests multiplying the summation of the individual clothing items clo value by a factor of 0.82 for clothing ensembles.

The metabolic rate (H/A_{Du}), is a measure of the internal heat production rate of an occupant (H) w/hr. in per unit of "Dubois" body surface area (A_{Du}) in units of m^2 . The DuBois body surface area is given by :

$$A_{Du} = 0.202(\text{weight})^{0.425}(\text{height})^{0.725} \quad (581)$$

Using this equation, an area of $1.8 m^2$ represents the surface area of an average person of weight 70 kg. and height 1.73 m (Fanger 1967). The metabolic rate is measured in mets, where 1 met = $58.2 W/m^2$.

The environmental variables that influence the conditions of thermal comfort include:

- (1) Air Temperature (T_a),
- (2) Mean Radiant Temperature (T_r),
- (3) Relative air velocity (v),
- (4) Water vapor pressure in ambient air (P_a)

The Air Temperature (T_a), a direct environmental index, is the dry-bulb temperature of the environment. The Mean Radiant Temperature (T_r) is a rationally derived environmental index defined as the uniform black-body temperature that would result in the same radiant energy exchange as in the actual environment. The Relative air velocity (v) a direct environmental index is a measure of the air motion obtainable via a hot wire or vane anemometers. The Water vapor pressure in ambient air (P_a) is a direct environmental index.

The physiological variables that influence the conditions of thermal comfort include:

- (1) Skin Temperature (T_{sk}),
- (2) Core or Internal Temperature (T_{cr}),
- (3) Sweat Rate,
- (4) Skin Wettedness (w),
- (5) Thermal Conductance (K) between the core and skin.

Where the Skin Temperature (T_{sk}), the Core Temperature (T_{cr}) and the Sweat Rate are physiological indices. The Skin Wettedness (w) is a rationally derived physiological index defined as the ratio of the actual sweating rate to the maximum rate of sweating that would occur if the skin were completely wet.

One more consideration is important in dealing with thermal comfort - the effect of asymmetrical heating or cooling. This could occur when there is a draft or when there is a radiant flux incident on a person (which is what is of primary interest to us here). Fanger (1967) noted that the human regulatory system is quite tolerant of asymmetrical radiant flux. A reasonable upper limit on the difference in mean radiant temperature (T_r) from one direction to the opposing direction is 15° (ASHRAE 1984). This limit is lower if there is a high air velocity in the zone.

Mathematical Models for Predicting Thermal Comfort

Many researchers have been exploring ways to predict the thermal sensation of people in their environment based on the personal, environmental and physiological variables that influence thermal comfort. From the research done, some mathematical models that simulate occupants' thermal response to their environment have been developed. Most thermal comfort prediction models use a seven or nine point thermal sensation scale, as in the following tables.

Table 51. Seven point Thermal Sensation Scale

Sensation	Description
3	Hot
2	Warm
1	slightly warm
0	neutral
-1	slightly cool
-2	cool
-3	cold

Table 52. Nine point Thermal Sensation Scale

Sensation Value	Description
4	very hot
3	hot
2	warm
1	slightly warm
0	neutral
-1	slightly cool
-2	cool
-3	cold
-4	very cold

The most notable models have been developed by P.O. Fanger (the Fanger Comfort Model), the J. B. Pierce Foundation (the Pierce Two-Node Model), and researchers at Kansas State University (the KSU Two-Node Model). Berglund (1978) presents a detailed description of the theory behind these three models.

Note for all Thermal Comfort reporting: Though the published values for thermal comfort “vote” have a discrete scale (e.g. –3 to +3 or –4 to +4), the calculations in EnergyPlus are carried out on a continuous scale and, thus, reporting may be “off the scale” with specific conditions encountered in the space. This is not necessarily an error in EnergyPlus – rather a different approach that does not take the “limits” of the discrete scale values into account.

The main similarity of the three models is that all three apply an energy balance to a person and use the energy exchange mechanisms along with experimentally derived physiological parameters to predict the thermal sensation and the physiological response of a person due to their environment. The models differ somewhat in the physiological models that represent the human passive system (heat transfer through and from the body) and the human control system (the neural control of shivering, sweating and skin blood flow). The models also differ in the criteria used to predict thermal sensation.

Fanger Comfort Model

Fanger's Comfort model was the first one developed. It was published first in 1967 (Fanger 1967) and then in 1970 (Fanger 1970), and helped set the stage for the other two models.

The mathematical model developed by P.O. Fanger is probably the most well known of the three models and is the easiest to use because it has been put in both chart and graph form.

Fanger Model Nomenclature List

Table 53. Nomenclature list for Fanger model

Mathematical variable	Description	Units	Range	FORTTRAN variable
A_{Du}	Dubois body surface area	m^2	-	BodySurfaceArea
C_{res}	The rate of dry respiratory heat loss	W/m^2	-	DryRespHeatLoss
E_{dif}	The rate of heat loss from the diffusion of water vapor through the skin	W/m^2	-	EvapHeatLossDiff
E_{res}	The rate of latent respiratory heat loss	W/m^2	-	LatRespHeatLoss
$E_{rsw,req}$	The rate of heat loss from the evaporation of regulatory sweating at the state of comfort	W/m^2	-	EvapHeatLossRegComf
E_{sk}	Total evaporative heat loss from skin	W/m^2		EvapHeatLoss
f_{cl}	The ratio of clothed body	-		CloBodyRat
f_{eff}	The fraction of surface effective for radiation (= 0.72)	-	-	RadSurfEff
H	Internal heat production rate of an occupant per unit area (= M – W)	W/m^2	-	IntHeatProd
h_c	Convective heat transfer coefficient	$W/m^2\text{°C}$	-	Hc
L	All the modes of energy loss from body	W/m^2	-	-
M	Metabolic rate per unit area	W/m^2	-	ActLevel
P_a	Water vapor pressure in ambient air	Torr	-	VapPress
PMV	Predicted Mean Vote	-	-4~4	PMV

P_{sk}	Saturated water vapor pressure at required skin temperature	Torr	-	SatSkinVapPress
Q_c	The rate of convective heat loss	W/m ²	-	ConvHeatLoss
Q_{dry}	Sensible heat flow from skin	W/m ²		DryHeatLoss
Q_r	The rate of radiative heat loss	W/m ²	-	RadHeatLoss
Q_{res}	The rate of respiratory heat loss	W/m ²	-	RespHeatLoss
T_a	Air temperature	°C	-	AirTemp
T_{cl}	Clothing surface temperature	°C	-	CloSurfTemp
T_{cla}	Clothing surface temperature (Absolute)	°K	-	AbsCloSurfTemp
T_{ra}	Mean radiant temperature	°K	-	AbsRadTemp
T_{skr}	Skin temperature required to achieve thermal comfort	°C		SkinComfTemp
W	The rate of heat loss due to the performance of work	W/m ²	-	WorkEff
ε	The emissivity of clothing-skin surface	-	-	SkinEmiss
σ	The Stefan-Boltzman constant (= 5.67×10^{-8})	W/m ² K ⁴	-	StefanBoltz

Description of the model and algorithm

Fanger developed the model based on the research he performed at Kansas State University and the Technical University of Denmark. Fanger used the seven-point form of a thermal sensation scale along with numerous experiments involving human subjects in various environments. He related the subjects in response to the variables, which influence the condition of thermal comfort. Fanger's model is based upon an energy analysis that takes into account all the modes of energy loss (L) from the body, including: the convection and radiant heat loss from the outer surface of the clothing, the heat loss by water vapor diffusion through the skin, the heat loss by evaporation of sweat from the skin surface, the latent and dry respiration heat loss and the heat transfer from the skin to the outer surface of the clothing. The model assumes that the person is thermally at steady state with his environment.

$$M = L \quad \text{W/m}^2 \quad (582)$$

$$L = Q_{res} + Q_{dry} + E_{sk} + W \quad \text{W/m}^2 \quad (583)$$

$$Q_{res} = E_{res} + C_{res} = 0.0023M(44 - P_a) + 0.0014M(34 - T_a) \quad \text{W/m}^2 \quad (584)$$

$$\text{LatRespHeatLoss} = 0.0023 * \text{ActLevel} * (44. - \text{VapPress})$$

$$\text{DryRespHeatLoss} = 0.0014 \cdot \text{ActLevel} \cdot (34 - \text{AirTemp})$$

$$\text{RespHeatLoss} = \text{LatRespHeatLoss} + \text{DryRespHeatLoss}$$

$$Q_c = h_c \times f_{cl} (T_{cl} - T_a) \quad \text{W/m}^2 \quad (585)$$

$$Q_r = f_{eff} f_{cl} \varepsilon \sigma (T_{cla}^4 - T_{ra}^4) \quad \text{W/m}^2 \quad (586)$$

$$Q_{dry} = Q_c + Q_r \quad \text{W/m}^2 \quad (587)$$

$$\text{ConvHeatLoss} = \text{CloBodyRat} \cdot H_c \cdot (\text{CloSurfTemp} - \text{AirTemp})$$

$$\text{RadHeatLoss} = \text{RadSurfEff} \cdot \text{CloBodyRat} \cdot \text{SkinEmiss} \cdot \text{StefanBoltz} \cdot (\text{AbsCloSurfTemp}^4 - \text{AbsRadTemp}^4)$$

$$\text{DryHeatLoss} = \text{ConvHeatLoss} + \text{RadHeatLoss}$$

$$\text{For } H > 58.2, E_{rsw} = 0.42(H - 58.2) \quad \text{W/m}^2 \quad (588)$$

$$\text{For } H \leq 58.2, E_{rsw} = 0 \quad \text{W/m}^2 \quad (589)$$

$$E_{diff} = 0.68 \times 0.61 (P_{sk} - P_a) = 0.4148 (P_{sk} - P_a) \quad \text{W/m}^2 \quad (590)$$

$$E_{sk} = E_{rsw} + E_{diff} \quad \text{W/m}^2 \quad (591)$$

$$\text{EvapHeatLossRegComf} = 0.42 \cdot (\text{IntHeatProd} - \text{ActLevelConv})$$

$$\text{EvapHeatLossRegComf} = 0.0$$

$$\text{EvapHeatLossDiff} = 0.4148 \cdot (\text{SkinComfVpress} - \text{VapPress})$$

$$\text{EvapHeatLoss} = \text{EvapHeatLossRegComf} + \text{EvapHeatLossDiff}$$

Where,

0.68 is the passive water vapor diffusion rate, (g/h·m²·Torr)

0.61 is the latent heat of water, (W·h/g)

P_{sk} is the saturated water vapor pressure at the skin temperature required to achieve the thermal comfort

$$P_{sk} = 1.92 T_{skr} - 25.3 \quad \text{Torr} \quad (592)$$

$$\text{SatSkinVapPress} = 1.92 \cdot \text{SkinTempComf} - 25.3$$

$$T_{skr} = 35.7 - 0.028 H \quad \text{°C} \quad (593)$$

$$\text{SkinTempComf} = 35.7 - 0.028 \cdot \text{IntHeatProd}$$

By determining the skin temperature and evaporative sweat rate that a thermally comfortable person would have in a given set of conditions, the model calculates the energy loss (L). Then, using the thermal sensation votes from subjects at KSU and Denmark, a Predicted Mean Vote (PMV) thermal sensation scale is based on how the energy loss (L) deviates from the metabolic rate (M) in the following form:

$$\text{PMV} = (0.303 e^{-0.036M} + 0.028)(H - L) \quad (594)$$

$$\text{ThermSensTransCoef} = 0.303 \cdot \text{EXP}(-0.036 \cdot \text{ActLevel}) + 0.028$$

$$\text{PMV} = \text{ThermSensTransCoef} \cdot (\text{IntHeatProd} - \text{EvapHeatLoss} - \text{RespHeatLoss} - \text{DryHeatLoss})$$

Pierce Two-Node Model

The Pierce Two-Node model was developed at the John B. Pierce Foundation at Yale University. The model has been continually expanding since its first publication in 1970 (Gagge et.al. 1970). The most recent version on the model appears in the 1986 ASHRAE Transactions (Gagge et.al. 1986).

Pierce Two-Node Model Nomenclature List

Table 54. Nomenclature list for Pierce Two-Node model

Mathematical variable	Description	Units	Range	FORTTRAN variable
C_{dil}	Constant for skin blood flow			SkinBloodFlowConst
C_{res}	The rate of dry respiratory heat loss	W/m ²	-	DryRespHeatLoss
C_{sw}	Proportionality constant for sweat control	g/m ² hr		SweatContConst
DISC	Predicted discomfort vote	-	-5~5	DISC
E_{dif}	The rate of heat loss from the diffusion of water vapor through the skin	W/m ²	-	EvapHeatLossDiff
E_{max}	Maximum evaporative heat loss	W/m ²		EvapHeatLossMax
E_{sk}	Total evaporative heat loss from skin	W/m ²		EvapHeatLoss
E_{res}	The rate of latent respiratory heat loss	W/m ²	-	LatRespHeatLoss
E_{rsw}	The rate of heat loss from the evaporation of regulatory sweating	W/m ²	-	EvapHeatLossRegSweat
$E_{rsw,req}$	The rate of heat loss from the evaporation of regulatory sweating at the state of comfort	W/m ²		EvapHeatLossRegComfort
ET*	Effective Temperature	°C	-	ET
f_{cl}	The ratio of clothed body	-		CloBodyRat
f_{eff}	The fraction of surface effective for radiation (= 0.72)	-	-	RadSurfEff
H	Internal heat production rate of an occupant per unit area (= M – W)	W/m ²	-	IntHeatProd
h	Combined heat transfer coefficient	W/m ² °C		H
h_c	Convective heat transfer coefficient	W/m ² °C	-	Hc

h_e'	Combined evaporative heat transfer coefficient	$W/(m^2kP_a)$		-
h_r	Radiant heat transfer coefficient	$W/m^2°C$	-	Hr
I_{cl}	Clothing insulation	$m^2°C/W$		-
L	All the modes of energy loss from body	W/m^2	-	-
L_{ET^*}	All the modes of energy loss from body at ET^*	W/m^2		-
L_{SET^*}	All the modes of energy loss from body at SET^*	W/m^2		-
M	Metabolic rate per unit area	W/m^2	-	ActLevel
M_{act}	Metabolic heat production due to activity	W/m^2		-
M_{shiv}	Metabolic heat production due to shivering	W/m^2		ShivResponse
P_a	Water vapor pressure in ambient air	Torr	-	VapPress
PMV^*	Predicted Mean Vote modified by ET^* or SET^*	-	-4~4	PMVET PMVSET
P_{SET^*}	Water vapor pressure at SET^*	$°C$		StdVapPressSET
P_{sk}	Saturated water vapor pressure at required skin temperature	Torr	-	SatSkinVapPress
Q_c	The rate of convective heat loss	W/m^2	-	ConvHeatLoss
Q_{crsk}	Heat flow from core to skin	W/m^2		HeatFlow
Q_{dry}	Sensible heat flow from skin	W/m^2		DryHeatLoss
Q_r	The rate of radiative heat loss	W/m^2	-	RadHeatLoss
Q_{res}	The rate of respiratory heat loss	W/m^2	-	RespHeatLoss
S_{cr}	Heat storage in core compartment	W/m^2		CoreheatStorage
SET^*	Standard Effective Temperature	$°C$	-	SET
SIG_b	Thermal signal of body	$°C$		BodyThermSigCold BodyThermSigWarm

SIG_{cr}	Thermal signal of core	$^{\circ}C$		CoreThermSigCold CoreThermSigWarm
SIG_{sk}	Thermal signal of skin	$^{\circ}C$		SkinThermSigCold SkinThermSigWarm
SKBF	Skin blood flow	L/m^2hr		SkinBloodFlow
S_{sk}	Heat storage in skin compartment	W/m^2		SkinHeatStorage
S_{tr}	Constriction constant of skin blood flow for average person			Str
SW_{reg}	The rate of regulatory sweating	g/m^2hr		RegSweat
T_a	Air temperature	$^{\circ}C$	-	AirTemp
T_b	Mean body temperature			AvgBodyTemp
T_{b-c}	Mean body temperature when DISC is zero (lower limit)	$^{\circ}C$		AvgBodyTempLow
T_{b-h}	Mean body temperature when HSI is 100 (upper limit)	$^{\circ}C$		AvgBodyTempHigh
T_{cl}	Clothing surface temperature	$^{\circ}C$	-	CloSurfTemp
T_{cr}	Core or internal temperature	$^{\circ}C$	-	CoreTemp
T_r	Mean radiant temperature	$^{\circ}C$	-	RadTemp
TSENS	Thermal sensation vote	-	-5~5	TSENS
T_{sk}	Skin temperature	$^{\circ}C$		SkinTemp
W	The rate of heat loss due to the performance of work	W/m^2	-	WorkEff
W_{dif}	Skin wettedness due to diffusion through the skin			SkinWetDiff
W_{rsw}	Skin wettedness due to regulatory sweating			SkinWetSweat
ϵ	The emissivity of clothing-skin surface	-	-	SkinEmiss
σ	The Stefan-Boltzman constant ($= 5.67 \times 10^{-8}$)	W/m^2K^4	-	StefanBoltz

Description of the model and algorithm

The Pierce model thermally lumps the human body as two isothermal, concentric compartments, one representing the internal section or core (where all the metabolic heat is assumed to be generated and the skin comprising the other compartment). This allows the passive heat conduction from the core compartment to the skin to be accounted for. The boundary line between two compartments changes with respect to skin blood flow rate per

unit skin surface area (SKBF in L/h•m²) and is described by alpha – the fraction of total body mass attributed to the skin compartment (Doherty and Arens 1988).

$$\alpha = 0.0417737 + 0.7451832 / (SKBF + 0.585417) \quad (595)$$

$$\text{SkinMassRat} = 0.0417737 + 0.7451832 / (\text{SkinBloodFlow} + 0.585417)$$

Furthermore, the model takes into account the deviations of the core, skin, and mean body temperature weighted by alpha from their respective setpoints. Thermoregulatory effector mechanisms (Regulatory sweating, skin blood flow, and shivering) are defined in terms of thermal signals from the core, skin and body (Doherty and Arens 1988).

$$SIG_{cr} = T_{cr} - 36.8 \quad ^\circ\text{C} \quad (596)$$

$$SIG_{sk} = T_{sk} - 33.7 \quad ^\circ\text{C} \quad (597)$$

$$SIG_b = T_b - 36.49 \quad ^\circ\text{C} \quad (598)$$

$$\text{SkinThermSigWarm} = \text{SkinTemp} - \text{SkinTempSet}$$

$$\text{SkinThermSigCold} = \text{SkinTempSet} - \text{SkinTemp}$$

$$\text{CoreThermSigWarm} = \text{CoreTemp} - \text{CoreTempSet}$$

$$\text{CoreThermSigCold} = \text{CoreTempSet} - \text{CoreTemp}$$

$$\text{BodyThermSigWarm} = \text{AvgBodyTemp} - \text{AvgBodyTempSet}$$

$$\text{BodyThermSigCold} = \text{AvgBodyTempSet} - \text{AvgBodyTemp}$$

$$SKBF = (6.3 + C_{dil} \times SIG_{cr}) / (1 + S_{tr} \times (-SIG_{sk})) \quad \text{L/hr}\cdot\text{m}^2 \quad (599)$$

$$\text{VasodilationFac} = \text{SkinBloodFlowConst} \times \text{CoreWarmDelTemp}$$

$$\text{VasoconstrictFac} = \text{Str} \times \text{SkinColdDelTemp}$$

$$\text{SkinBloodFlow} = (6.3 + \text{VasodilationFac}) / (1 + \text{VasoconstrictFac})$$

$$SW_{reg} = C_{sw} \times SIG_b e^{(SIG_{sk} / 10.7)} \quad \text{g/hr}\cdot\text{m}^2 \quad (600)$$

$$\text{RegSweat} = \text{SweatContConst} \times \text{BodyWarmDelTemp} \times \text{EXP}(\text{SkinWarmDelTemp} / 10.7)$$

$$M_{shiv} = 19.4 (-SIG_{cr}) (-SIG_{sk}) \quad \text{W/m}^2 \quad (601)$$

$$\text{ShivResponse} = 19.4 \times \text{SkinThermSigCold} \times \text{CoreThermSigCold}$$

The latest version of the Pierce model (Fountain and Huizenga 1997) discusses the concepts of SET* and ET*. The Pierce model converts the actual environment into a "standard environment" at a Standard Effective Temperature, SET*. SET* is the dry-bulb temperature of a hypothetical environment at 50% relative humidity for subjects wearing clothing that would be standard for the given activity in the real environment. Furthermore, in this standard environment, the same physiological strain, i.e. the same skin temperature and skin wettedness and heat loss to the environment, would exist as in the real environment. The Pierce model also converts the actual environment into a environment at an Effective Temperature, ET*, that is the dry-bulb temperature of a hypothetical environment at 50% relative humidity and uniform temperature (Ta = MRT) where the subjects would experience the same physiological strain as in the real environment.

In the latest version of the model it is suggested that the classical Fanged PMV be modified by using ET* or SET* instead of the operative temperature. This gives a new index PMV* which is proposed for dry or humid environments. It is also suggested that PMV* is very responsive to the changes in vapor permeation efficiency of the occupants clothing.

$$M = M_{act} + M_{shiv} \quad \text{W/m}^2 \quad (602)$$

$$\text{ActLevel} = \text{ActLevel} + \text{ActShiv}$$

$$L = Q_{res} + Q_{dry} + E_{sk} + W \quad \text{W/m}^2 \quad (603)$$

$$\begin{aligned} Q_{res} &= E_{res} + C_{res} = 0.0023M(44 - P_{a(torr)}) + 0.0014M(34 - T_a) \\ &= 0.017251M(5.8662 - P_{a(kPa)}) + 0.0014M(34 - T_a) \end{aligned} \quad \text{W/m}^2 \quad (604)$$

$$\text{LatRespHeatLoss} = 0.017251 * \text{ActLevel} * (5.8662 - \text{VapPress})$$

$$\text{DryRespHeatLoss} = 0.0014 * \text{ActLevel} * (34 - \text{AirTemp})$$

$$\text{RespHeatLoss} = \text{LatRespHeatLoss} + \text{DryRespHeatLoss}$$

$$Q_c = h_c \times f_{cl}(T_{cl} - T_a) \quad \text{W/m}^2 \quad (605)$$

$$Q_r = h_r \times f_{cl}(T_{cl} - T_r) \quad \text{W/m}^2 \quad (606)$$

$$Q_{dry} = Q_c + Q_r \quad \text{W/m}^2 \quad (607)$$

$$\text{DryHeatLoss} = \text{CloBodyRat} * (\text{Hc} * (\text{CloSurfTemp} - \text{AirTemp}) + \text{Hr} * (\text{CloSurfTemp} - \text{RadTemp}))$$

In Pierce model, the convective heat transfer coefficient, h_c , varies with the air velocity around body and metabolic rate. The model uses the maximum value of following equations.

$$h_c = 8.6 \times v^{0.53} \quad \text{W/m}^2\text{°C} \quad (608)$$

$$h_c = 5.66(M / 58.2 - 0.85)^{0.39} \quad \text{W/m}^2\text{°C} \quad (609)$$

$$\text{Hc} = 8.6 * \text{AirVel}^{0.53}$$

$$\text{HcAct} = 5.66 * (\text{ActMet} - 0.85)^{0.39}$$

Also, in the model, the radiant heat transfer coefficient, h_r , is defined by following equation (Doherty and Arens 1988):

$$h_r = 4. \times f_{eff} \varepsilon \sigma ((T_{cl} + T_r) / 2 + 273.15)^3 \quad \text{W/m}^2\text{°C} \quad (610)$$

$$\text{Hr} = 4. * \text{RadSurfEff} * \text{StefanBoltz} * ((\text{CloSurfTemp} + \text{RadTemp}) / 2. + \text{TAbsConv})^{**3}$$

In the Pierce model, T_{cl} is estimated by each iteration using following equation:

$$T_{cl} = (T_{sk} / I_{cl} + f_{cl}(h_c T_a - h_r T_r)) / (1 / I_{cl} + f_{cl}(h_c + h_r)) \quad \text{°C} \quad (611)$$

$$\begin{aligned} \text{CloSurfTemp} &= (\text{CloCond} * \text{SkinTemp} + \text{CloBodyRat} * (\text{Hc} * \text{AirTemp} \& \\ &+ \text{Hr} * \text{RadTemp})) / (\text{CloCond} + \text{CloBodyRat} * (\text{Hc} + \text{Hr})) \end{aligned}$$

Total evaporative heat loss from the skin, E_{sk} , includes evaporation of water produced by regulatory sweating, E_{rsw} , and evaporation of water vapor that diffuses through the skin surface, E_{diff} .

$$E_{sk} = E_{rsw} + E_{diff} \quad \text{W/m}^2 \quad (612)$$

$$\text{EvapHeatLoss} = \text{EvapHeatLossRegSweat} + \text{EvapHeatLossRegDiff}$$

$$E_{rsw} = 0.68 \times SW_{reg} \quad \text{W/m}^2 \quad (613)$$

$$E_{diff} = w_{diff} \times E_{max} \quad \text{W/m}^2 \quad (614)$$

RegHeatLoss = 0.68*RegSweat

DiffHeatLoss = SkinWetDiff*MaxEvapHeatLoss

Where,

0.68 is the passive water vapor diffusion rate in g/h·m²·Torr

and,

$$w_{diff} = 0.06(1 - w_{rsw}) \quad (615)$$

$$E_{max} = h_e (P_{sk} - P_a) \quad \text{W/m}^2 \quad (616)$$

$$w_{rsw} = E_{rsw} / E_{max} \quad (617)$$

SkinWetDiff = (1.-SkinWetSweat)*.06

MaxEvapHeatLoss = (1./TotEvapHeatResist)*(SatSkinVapPress - VapPress)

SkinWetSweat = EvapHeatLossRegSweat/MaxEvapHeatLoss

The Pierce model has one additional heat flow term describing the heat transfer between the internal core compartment and the outer skin shell (Doherty and Arens 1988).

$$Q_{crsk} = (5.28 + 1.163SKBF)(T_{cr} - T_{sk}) \quad \text{W/m}^2 \quad (618)$$

HeatFlow = (CoreTemp-SkinTemp)*(5.28 + 1.163*SkinBloodFlow)

Where

5.28 is the average body tissue conductance in W/m²·°C

1.163 is the thermal capacity of blood in W·h/L·°C

Thus, individual heat balance equations for core and skin compartments are expressed using this term, Q_{c-s}. New temperatures of core, skin and body are calculated by each iteration from rates of heat storage in the core and skin.

$$S_{sk} = Q_{c-s} - Q_c - Q_r - E_{sk} \quad \text{W/m}^2\text{°C} \quad (619)$$

SkinHeatStorage = HeatFlow - DryHeatLoss - EvapHeatLoss

$$S_{cr} = M - W - Q_{res} - Q_{c-s} \quad \text{W/m}^2\text{°C} \quad (620)$$

CoreHeatStorage = IntHeatProd - RespHeatLoss - HeatFlow

Thus,

$$PMVET = (0.303e^{-0.036M} + 0.028)(H - L_{ET*}) \quad (621)$$

$$PMVSET = (0.303e^{-0.036M} + 0.028)(H - L_{SET*}) \quad (622)$$

ThermSensTransCoef = 0.303*EXP(-0.036*ActLevel) + 0.028

PMVET = ThermSensTransCoef*(IntHeatProd - EvapHeatLossDiff &
- EvapHeatLossRegComf - RespHeatLoss - DryHeatLossET)

PMVSET = ThermSensTransCoef*(IntHeatProd - EvapHeatLossDiff &
- EvapRegHeatLossReg Comf - RespHeatLoss - DryHeatLossSET)

Besides PMV*, the Pierce Two Node Model uses the indices TSENS and DISC as predictors of thermal comfort. Where TSENS is the classical index used by the Pierce foundation, and is a function of the mean body temperature. DISC is defined as the relative thermoregulatory strain that is needed to bring about a state of comfort and thermal equilibrium. DISC is a function of the heat stress and heat strain in hot environments and equal to TSENS in cold environments. In summary, the Pierce Model, for our purposes, uses four thermal comfort indices; PMVET-a function of ET*, PMVSET- a function of SET*, TSENS and DISC.

$$T_{b-c} = (0.185 / 58.2)(M - W) + 36.313 \quad ^\circ\text{C} \quad (623)$$

$$T_{b-h} = (0.359 / 58.2)(M - W) + 36.664 \quad ^\circ\text{C} \quad (624)$$

$$TSENS_c = 0.68175(T_b - T_{b-c}) \quad T_b \leq T_{b-c} \quad (625)$$

$$TSENS_h = 4.7(T_b - T_{b-c}) / (T_{b-h} - T_{b-c}) \quad T_b > T_{b-c} \quad (626)$$

$$DISC = 5.(E_{rsw} - E_{rsw-comf}) / (E_{max} - E_{rsw-comf} - E_{diff}) \quad (627)$$

AvgBodyTempLow = (0.185/ActLevelConv)*IntHeatProd + 36.313

AvgBodyTempHigh = (0.359/ActLevelConv)*IntHeatProd + 36.664

TSENS = .68175*(AvgBodyTemp-AvgBodyTempLow)

TSENS = 4.7*(AvgBodyTemp - AvgBodyTempLow) / &
(AvgBodyTempHigh - AvgBodyTempLow)

DISC = 5.*(EvapHeatLossRegSweat - EvapHeatLossRegComf) / &
(MaxEvapHeatLoss - EvapHeatLossRegComf - DiffHeatLoss)

KSU Two-Node Model

The KSU two-node model, developed at Kansas State University, was published in 1977 (Azer and Hsu 1977). The KSU model is quite similar to that of the Pierce Foundation. The main difference between the two models is that the KSU model predicts thermal sensation (TSV) differently for warm and cold environment.

KSU Two Node Model Nomenclature List

Table 55. Nomenclature list for KSU Two-Node model

Mathematical variable	Description	Units	Range	FORTTRAN variable
C_{cr}	Specific heat of body core	Whr/kg $^\circ$ C		
C_{sk}	Specific heat of skin	Whr/kg $^\circ$ C		
C_{res}	The rate of dry respiratory heat loss	W/m 2	-	DryRespHeatLoss
E_{dif}	The rate of heat loss from the diffusion of water vapor through the skin	W/m 2	-	EvapHeatLossDiff

E_{\max}	Maximum evaporative heat loss	W/m^2		EvapHeatLossMax
E_{sk}	Total evaporative heat loss from skin	W/m^2		EvapHeatLoss
E_{sw}	Equivalent evaporation heat loss from the sweat secreted	W/m^2		EvapHeatLossSweat
$E_{sw,d}$	Sweat function for warm and dry skin	W/m^2		DrySweatRate
E_{res}	The rate of latent respiratory heat loss	W/m^2	-	LatRespHeatLoss
F_{cl}	The Burton thermal efficiency factor for clothing		-	CloThermEff
F_{pcl}	Permeation efficiency factor for clothing		-	CloPermeatEff
H	Internal heat production rate of an occupant per unit area $= M - W$	W/m^2	-	IntHeatProd
H	Combined heat transfer coefficient	$W/m^2\text{°C}$		H
h_c	Convective heat transfer coefficient	$W/m^2\text{°C}$	-	Hc
h_r	Radiant heat transfer coefficient	$W/m^2\text{°C}$	-	Hr
KS	Overall skin thermal conductance	$W/m^2\text{°C}$		ThermCndct
KS_o	Skin conductance at thermal neutrality	$W/m^2\text{°C}$		ThermCndctNeut
$KS_{(-4)}$	Skin conductance at thermal sensation very cold	$W/m^2\text{°C}$		ThermCndctMin
M	Metabolic rate per unit area	W/m^2	-	ActLevel
M_{shiv}	Metabolic heat production due to shivering	W/m^2		ShivResponse
P_a	Water vapor pressure in ambient air	Torr	-	VapPress
P_{sk}	Saturated water vapor pressure at required skin temperature	Torr	-	SatSkinVapPress

PT_{accl}	The pattern of acclimation			AcclPattern
Q_c	The rate of convective heat loss	W/m^2	-	ConvHeatLoss
Q_{dry}	Sensible heat flow from skin	W/m^2		DryHeatLoss
Q_r	The rate of radiative heat loss	W/m^2	-	RadHeatLoss
Q_{res}	The rate of respiratory heat loss	W/m^2	-	RespHeatLoss
RH	Relative humidity			RelHum
T_a	Air temperature	$^{\circ}C$	-	AirTemp
T_{cr}	Core or internal temperature	$^{\circ}C$	-	CoreTemp
T_o	Operative temperature	$^{\circ}C$	-	OpTemp
T_r	Mean radiant temperature	$^{\circ}C$	-	RadTemp
T_{sk}	Skin temperature	$^{\circ}C$		SkinTemp
TSV	Thermal sensation vote		-4~4	TSV
V	Relative air velocity	m/s	-	AirVel
W	The rate of heat loss due to the performance of work	W/m^2	-	WorkEff
W	Skin wettedness	-	-	SkinWet
W_{cr}	Mass of body core per unit body surface	kg/m^2		-
W_{rsw}	Skin wettedness due to regulatory sweating			SkinWetSweat
W_{rsw-o}	Skin wettedness at thermal neutrality			SkinWetSweatNeut
W_{sk}	Mass of skin per unit body surface	kg/m^2		-

Description of the model and algorithm

The KSU two-node model is based on the changes that occur in the thermal conductance between the core and the skin temperature in cold environments, and in warm environments it is based on changes in the skin wettedness.

In this model metabolic heat production is generated in the core which exchanges energy with the environment by respiration and the skin exchanges energy by convection and radiation. In addition, body heat is dissipated through evaporation of sweat and/or water vapor diffusion through the skin. These principles are used in following passive system equations.

$$W_{cr} C_{cr} dT_{cr} / dt = M - W - Q_{res} - KS(T_{cr} - T_{sk}) \quad \text{W/m}^2 \quad (628)$$

$$W_{sk} C_{sk} dT_{sk} / dt = KS(T_{cr} - T_{sk}) - Q_{dry} - E_{sk} \quad \text{W/m}^2 \quad (629)$$

Where

$$Q_{res} = E_{res} + C_{res} = 0.0023M(44 - P_{a(torr)}) + 0.0014M(34 - T_a) \quad \text{W/m}^2 \quad (630)$$

LatRespHeatLoss = 0.0023*ActLevelTot*(44. - VapPress)

DryRespHeatLoss = 0.0014*ActLevelTot*(34. - AirTemp)

RespHeatLoss = LatRespHeatLoss + DryRespHeatLoss

$$Q_{dry} = Q_c + Q_r = h f_{cl} F_{cl} (T_{sk} - T_o) \quad \text{W/m}^2 \quad (631)$$

DryHeatLoss = H*CloBodyRat*CloThermEff*(SkinTemp - OpTemp)

$$h = h_c + h_r \quad \text{W/m}^2\text{°C} \quad (632)$$

$$h_c = 8.3\sqrt{v} \quad \text{W/m}^2\text{°C} \quad (633)$$

$$h_r = 3.87 + 0.031T_r \quad \text{W/m}^2\text{°C} \quad (634)$$

H = Hc + Hr

Hc = 8.3*SQRT(AirVel)

Hr = 3.87 + 0.031*RadTemp

$$T_o = (h_c T_a + h_r T_r) / (h_c + h_r) \quad \text{°C} \quad (635)$$

OpTemp = (Hc*AirTemp + Hr*RadTemp)/H

and

$$\text{For } E_{sw} \leq E_{max}, E_{sk} = E_{sw} + (1 - w_{rsw}) E_{diff} \quad \text{W/m}^2 \quad (636)$$

$$\text{For } E_{sw} > E_{max}, E_{sk} = E_{max} \quad \text{W/m}^2 \quad (637)$$

$$E_{diff} = 0.408(P_{sk} - P_a) \quad \text{W/m}^2 \quad (638)$$

$$E_{max} = 2.2h_c F_{pcl} (P_{sk} - P_a) \quad \text{W/m}^2 \quad (639)$$

EvapHeatLoss = SkinWetSweat*EvapHeatLossMax+(1. - SkinWetSweat)*EvapHeatLossDiff

SkinWetSweat = EvapHeatLossDrySweat/EvapHeatLossMax

EvapHeatLossDiff = 0.408*(SkinVapPress - VapPress)

EvapHeatLossMax = 2.2*Hc*(SkinVapPress - VapPress)*CloPermeatEff

Here, control signals, based on setpoint temperatures in the skin and core, are introduced into passive system equations and these equations are integrated numerically for small time increments or small increments in core and skin temperature. The control signals modulate the thermoregulatory mechanism and regulate the peripheral blood flow, the sweat rate, and the increase of metabolic heat by active muscle shivering. The development of the controlling

functions of skin conductance (KS), sweat rate (E_{sw}), and shivering (M_{shiv}) is based on their correlation with the deviations in skin and core temperatures from their setpoints.

$$KS = 5.3 + [6.75 + 42.45(T_{cr} - 36.98) + 8.15(T_{cr} - 35.15)^{0.8}(T_{sk} - 33.8)] / [1.0 + 0.4(32.1 - T_{sk})] \quad (640)$$

$$\begin{aligned} \text{SkinCndctDilation} &= 42.45 * \text{CoreSignalWarmMax} \& \\ &+ 8.15 * \text{CoreSignalSkinSens}^{0.8} * \text{SkinSignalWarmMax} \\ \text{SkinCndctConstriction} &= 1.0 + 0.4 * \text{SkinSignalColdMax} \\ \text{ThermCndct} &= 5.3 + (6.75 + \text{SkinCndctDilation}) / \text{SkinCndctConstriction} \end{aligned}$$

$$E_{sw} = \phi \times [260(T_{cr} - 36.9) + 26(T_{sk} - 33.8)] \exp[(T_{sk} - 33.8) / 8.5] / [1.0 + 0.05(33.37 - T_{sk})^{2.4}] \quad (641)$$

$$\begin{aligned} \text{WeighFac} &= 260 + 70 * \text{AcclPattern} \\ \text{SweatCtrlFac} &= 1. + 0.05 * \text{SkinSignalSweatColdMax}^{2.4} \\ \text{DrySweatRate} &= ((\text{WeighFac} * \text{CoreSignalSweatMax} \& \\ &+ 0.1 * \text{WeighFac} * \text{SkinSignalSweatMax}) \& \\ &* \text{EXP}(\text{SkinSignalSweatMax} / 8.5)) / \text{SweatCtrlFac} \end{aligned}$$

Where

$$\phi = 1.0 \quad w \leq 0.4 \quad (642)$$

$$\phi = 0.5 + 0.5 \exp[-5.6(w - 0.4)] \quad w > 0.4 \quad (643)$$

$$\text{SweatSuppFac} = 1.$$

$$\text{SweatSuppFac} = 0.5 + 0.5 * \text{EXP}(-5.6 * \text{SkinWetSignal})$$

$$M_{shiv} = 20(36.9 - T_{cr})(32.5 - T_{sk}) + 5(32.5 - T_{sk}) \quad \text{W/m}^2 \quad (644)$$

$$\text{ShivResponse} = 20 * \text{CoreSignalShivMax} * \text{SkinSignalShivMax} + 5 * \text{SkinSignalShivMax}$$

In KSU model, two new parameters are introduced and used in correlating thermal sensations with their associated physiological responses. In stead of correlating warm thermal sensations with skin wettedness, it is here correlated with a wettedness factor defined by

$$\varepsilon_{wsw} = (w_{rsw} - w_{rsw-o}) / (1.0 - w_{rsw-o}) \quad (645)$$

$$\text{SkinWetFac} = (\text{SkinWetSweat} - \text{SkinWetNeut}) / (1. - \text{SkinWetNeut})$$

Where

$$w_{wsw} = E_{sw} / E_{max}$$

$$w_{wsw-o} = 0.2 + 0.4 \{1.0 - \exp[-0.6(H / 58.2 - 1.0)]\}$$

$$\text{SkinWetSweat} = \text{DrySweatRate} / \text{EvapHeatLossMax}$$

$$\text{SkinWetNeut} = 0.02 + 0.4 * (1. - \text{EXP}(-0.6 * (\text{IntHeatProdMetMax} - 1.)))$$

and instead of correlating cold thermal sensation with the skin temperature, it is here correlated with a factor identified as vasoconstriction factor defined by

$$\varepsilon_{vc} = (KS_o - KS)/(KS_o - KS_{(-4)}) \quad (646)$$

$$\text{VasoconstrictFac} = (\text{ThermCndctNeut} - \text{ThermCndct}) / (\text{ThermCndctNeut} - \text{ThermCndctMin})$$

Thus, TSV in the cold is a function of a vasoconstriction factor (ε_{vc}) as:

$$TSV = -1.46 \times \varepsilon_{vc} + 3.75 \times \varepsilon_{vc}^2 - 6.17 \times \varepsilon_{vc}^3 \quad (647)$$

$$TSV = -1.46153 \times \text{VasoconstrictFac} + 3.74721 \times \text{VasoconstrictFac}^2 - 6.168856 \times \text{VasoconstrictFac}^3$$

and for the warm environments, TSV is defined as:

$$TSV = [5.0 - 6.56(RH - 0.5)] \times \varepsilon_{wsw} \quad (648)$$

$$TSV = (5. - 6.56 * (\text{RelHum} - 0.50)) * \text{SkinWetFac}$$

The KSU model's TSV was developed from experimental conditions in all temperature ranges and from clo levels between .05 clo to 0.7 clo and from activities levels of 1 to 6 mets (Berglund 1978).

Mean Radiant Temperature Calculation

There are three options to calculate mean radiant temperature in the thermal comfort models. One is the zone averaged MRT, another is the surface weighted MRT, and the other is angle factor MRT. The zone averaged MRT is calculated on the assumption that a person is in the center of a space, whereas the surface weighted MRT is calculated in consideration of the surface that a person is closest to, and the angle factor MRT is calculated based on angle factors between a person and the different surfaces in a space. Here, the surface weighted MRT is the average temperature of the selected surface and zone averaged MRT and is intended to represent conditions in the limit as a person gets closer and closer to a particular surface. In that limit, half of the person's radiant field will be dominated by that surface and the other half will be exposed to the rest of the zone. Note that the surface weighted MRT is only an approximation. The angle factor MRT is the mean temperature of the surrounding surface temperatures weighted according to the magnitude of the respective angle factors and allows the user to more accurately predict thermal comfort at a particular location within a space.

Table 56. Nomenclature and variable list for MRT calculation

Mathematical variable	Description	Units	Range	FORTTRAN variable
T_r	Mean radiant temperature	°C	-	RadTemp
T_{r-avg}	Zone averaged radiant temperature	°C	-	ZoneRadTemp
T_{surf}	Surface temperature	°C	-	SurfaceTemp
F_{surf}	Angle factor between person and surface	-	0~1	AngleFactor

Description of the model and algorithm

The zone averaged MRT is calculated without weighting any surface temperature of the space.

$$T_r = T_{r-avg}$$

RadTemp = MRT(ZoneNum)

The surface weighted MRT is the average temperature of the zone averaged MRT and the temperature of the surface that a person is closest to.

$$T_r = (T_{r-avg} + T_{surf}) / 2$$

ZoneRadTemp = MRT(ZoneNum)

SurfaceTemp = GetSurfaceTemp(People(PeopleNum)%SurfacePtr)

RadTemp = (ZoneRadTemp + SurfaceTemp)/2.0

The angle factor MRT is the mean value of surrounding surface temperatures weighted by the size of the respective angle factors between a person and each surface.

$$T_r = T_{surf-1}F_{surf-1} + T_{surf-2}F_{surf-2} + \dots + T_{surf-n}F_{surf-n}$$

SurfTempAngleFacSummed = SurfTempAngleFacSummed &
+ SurfaceTemp * AngleFactorList(AngleFacNum)%AngleFactor(SurfNum)

RadTemp = SurfTempAngleFacSummed

References

- ASHRAE. 1984. "High Intensity Infrared Radiant Heating", 1984 system Handbook, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 18.
- ASHRAE. 1985. "Physiological Principles for Comfort and Health," 1985 Fundamental Handbook, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 8.
- ASHRAE. 1993. "Physiological Principles and Thermal Comfort", 1993 ASHRAE Handbook-Fundamentals, American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta, GA, Chapter 8.
- Azer, N.Z., Hsu, S. 1977. "The prediction of Thermal Sensation from Simple model of Human Physiological Regulatory Response", ASHRAE Trans., Vol.83, Pt 1.
- Berglund, Larry. 1978. "Mathematical Models for Predicting the Thermal Comfort Response of Building Occupants", ASHRAE Trans., Vol.84.
- Doherty, T.J., Arens, E. 1988. "Evaluation of the Physiological Bases of Thermal Comfort Models", ASHRAE Trans., Vol.94, Pt 1.
- Du Bois, D. and E.F. 1916. "A Formula to Estimate Approximate Surface Area, if Height and Weight are Known", Archives of internal Medicine, Vol.17.
- Fanger, P.O. 1970. Thermal Comfort-Analysis and Applications in Environmental Engineering, Danish Technical Press, Copenhagen.
- Fanger, P.O. 1986. "Radiation and Discomfort", ASHRAE Journal. February 1986.
- Fanger P.O. 1967. "Calculation of Thermal Comfort: Introduction of a Basic Comfort Equation", ASHRE Trans., Vol.73, Pt 2.
- Fountain, Marc.E., Huizenga, Charlie. 1997 "A Thermal Sensation Prediction Tool for Use by the Profession", ASHRAE Trans., Vol.103, Pt 2.
- Gagge, A.P., Stolwijk, J. A. J., Nishi, Y. 1970. "An Effective Temperature Scale Based on a Simple Model of Human Physiological Regulatory Response", ASHRAE Trans., Vol.70, Pt 1.
- Gagge, A.P., Fobelets, A.P., Berglund, L. G. 1986. "A Standard Predictive Index of Human Response to the Thermal Environment", ASHRAE Trans., Vol.92, Pt 2.

- Hsu, S. 1977. "A Thermoregulatory Model for Heat Acclimation and Some of its Application", Ph. D. Dissertation, Kansas State University.
- Int-Hout, D. 1990. "Thermal Comfort Calculation / A Computer Model", ASHRAE Trans., Vol.96, Pt 1.
- ISO. 1983. "Determination of the PMV and PPD Indices and Specification of the Conditions for Thermal Comfort", DIS 7730, Moderate Thermal Environment, 1983.

Trombe Walls

Trombe walls are passive solar devices designed for thermal storage and delivery. It consists of a thick wall (150mm to 300mm) [8" to 16"] faced with a selective surface solar absorber, air gap, and high transmissivity glass pane. Trombe walls are usually South facing (in the Northern Hemisphere) for maximum sun exposure. An overhang above the wall is used to decrease exposure in the summer when the sun is high in the sky and heating is not required, yet still allows for full exposure in the winter when the sun is low in the sky and heating is desirable.

In EnergyPlus, there is no Trombe wall object per se; rather, it is composed of other existing EnergyPlus objects in the input file (except for a special key choice for Zone Inside Convection Algorithm in the Zone input object). This approach provides flexibility in specifying the various wall parameters and allows the freedom to explore unusual configurations. On the other hand, this approach puts more of a burden on the user to be sure that all parts of the Trombe wall are correctly specified; otherwise unexpected results may be obtained.

To simulate the Trombe wall, a very narrow zone is coupled to the desired surface via an interzone partition. The depth of the zone corresponds to the size of the air space usually 18mm to 150mm ($\frac{3}{4}$ " to 6"). In most cases the Trombe zone will be a sealed zone with no ventilation. The exterior wall of the Trombe zone contains a single or double-pane window. Optimally, the window covers nearly all of the wall area and has a very high transmissivity to allow the maximum amount of solar flux into the Trombe zone. Frames and dividers can be defined as usual for the window. The interior wall is usually constructed of very thick masonry materials with a solar absorber surface as the innermost layer of the wall. The absorber is a selective surface material with very high absorptivity and very low emissivity, e.g. copper with a special black surface treatment. It is important to make sure the Solar Distribution field in the **Building** object is set to FullInteriorAndExterior so that the majority of the solar flux is directed on the absorber surface and not just on the very small area of the Trombe zone floor. The Zone Inside Convection Algorithm for the Trombe's Zone object should also be set to TrombeWall to correctly model the air space. As is the case for all interzone partitions, the wall construction of the adjoining zone must be the mirror image of the wall construction in the Trombe zone. Finally, an overhang is optionally attached to the Trombe zone to control the amount of seasonal sun exposure. Since the user selects all of the Trombe wall parameters in the input file, there is considerable freedom to experiment with different materials, sizes, and configurations.

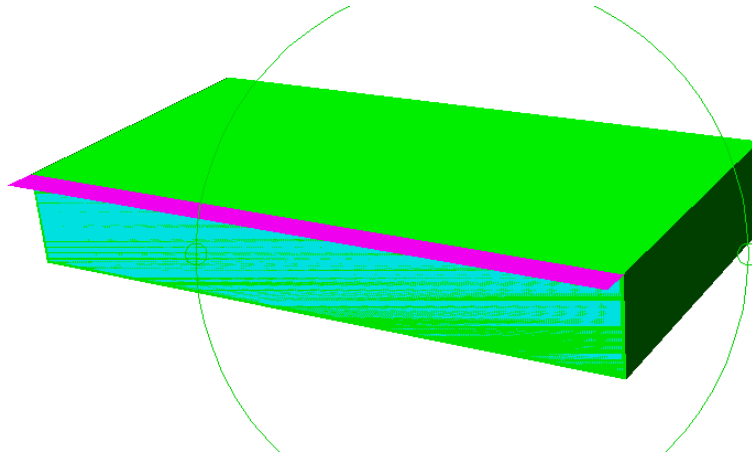


Figure 220. Building with Trombe Wall

Passive Trombe Wall

Passive Trombe walls perform without the assistance of any additional mechanical equipment. Most Trombe walls are passive Trombe walls. They can be either sealed or naturally ventilated.

For a sealed or unvented Trombe wall, the *Zone Inside Convection Algorithm* field in the Zone object should be set to "TrombeWall". This algorithm correctly calculates the convection coefficients for a narrow sealed vertical cavity based on the ISO 15099 standard. Refer to the "Trombe Wall Algorithm" subsection (under Interior Convection, above) for a complete description of the algorithm. The EnergyPlus modeling approach for the sealed passive Trombe wall has been validated with experimental data (Ellis 2003).

For a naturally ventilated Trombe wall, there is no built-in algorithm for calculating the correct convection coefficients on the inside of the cavity walls. One option is to use the "Detailed" convection algorithm. This algorithm takes into account some natural convection effects but is intended for a normal sized room. Therefore, some error may be incurred when used with a narrow cavity. Another option is to use the SurfaceProperty:ConvectionCoefficients object to schedule coefficients that have been determined beforehand by the user.

Input File

An input file (PassiveTrombeWall.idf) is provided to demonstrate a sample sealed Trombe wall implementation. In this file two separated fictional buildings are simulated for summer and winter design days in Zion, Utah. The buildings are identical in size and construction except that one has a Trombe wall and the other does not. The buildings have uncontrolled zones with no internal loads and heavy insulation. All floors use interzone partitions to disconnect them from the ground. The window on the Trombe zone is a 3 mm, low iron, single pane glazing with very high transmissivity (0.913 visible, 0.899 solar). The absorber surface is a Tabor solar absorber with an emittance of 0.05 and absorptance of 0.85.

Results

The resulting temperature profiles for winter and summer design days are plotted below.

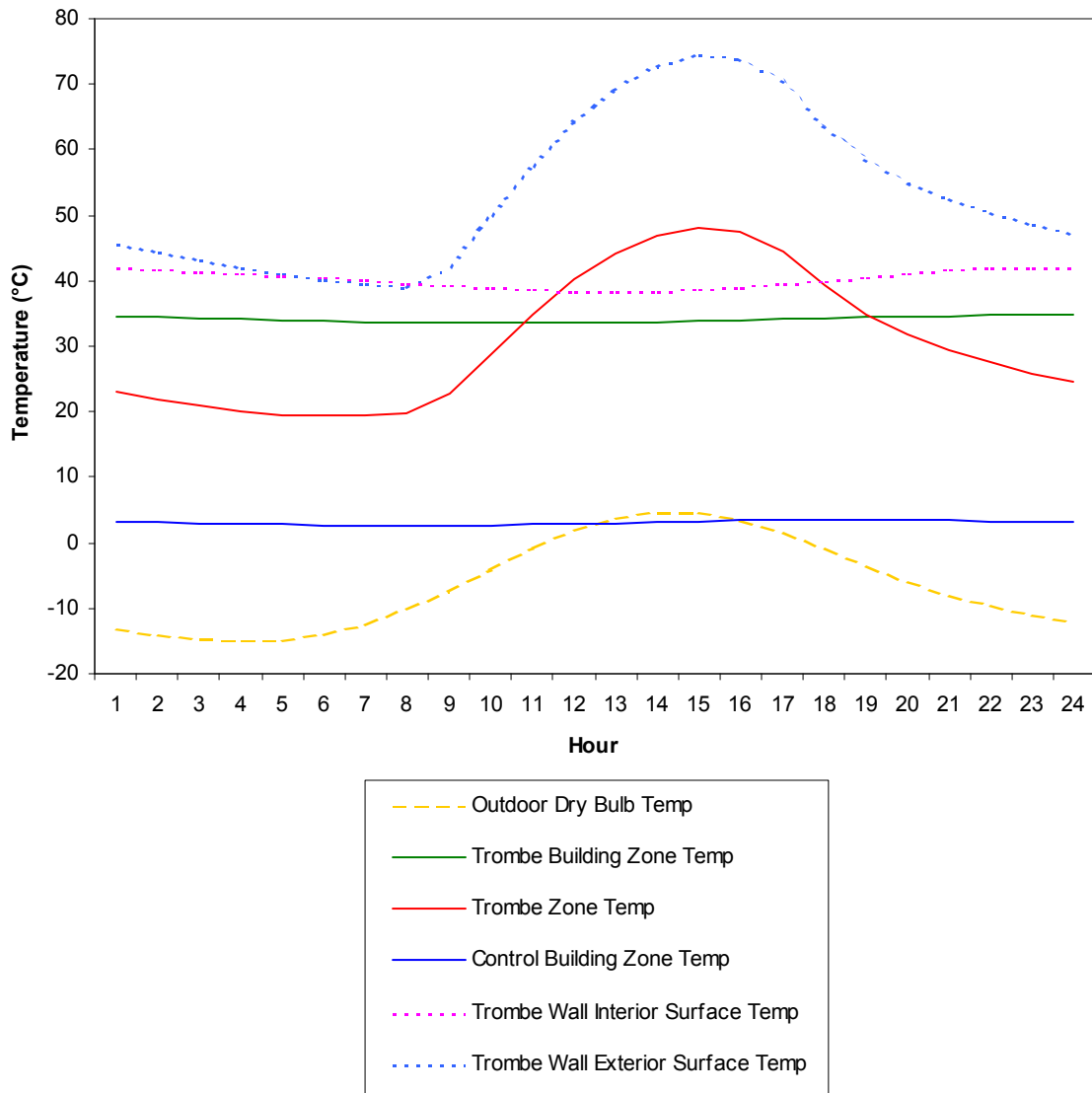


Figure 221. Passive Trombe Wall Winter

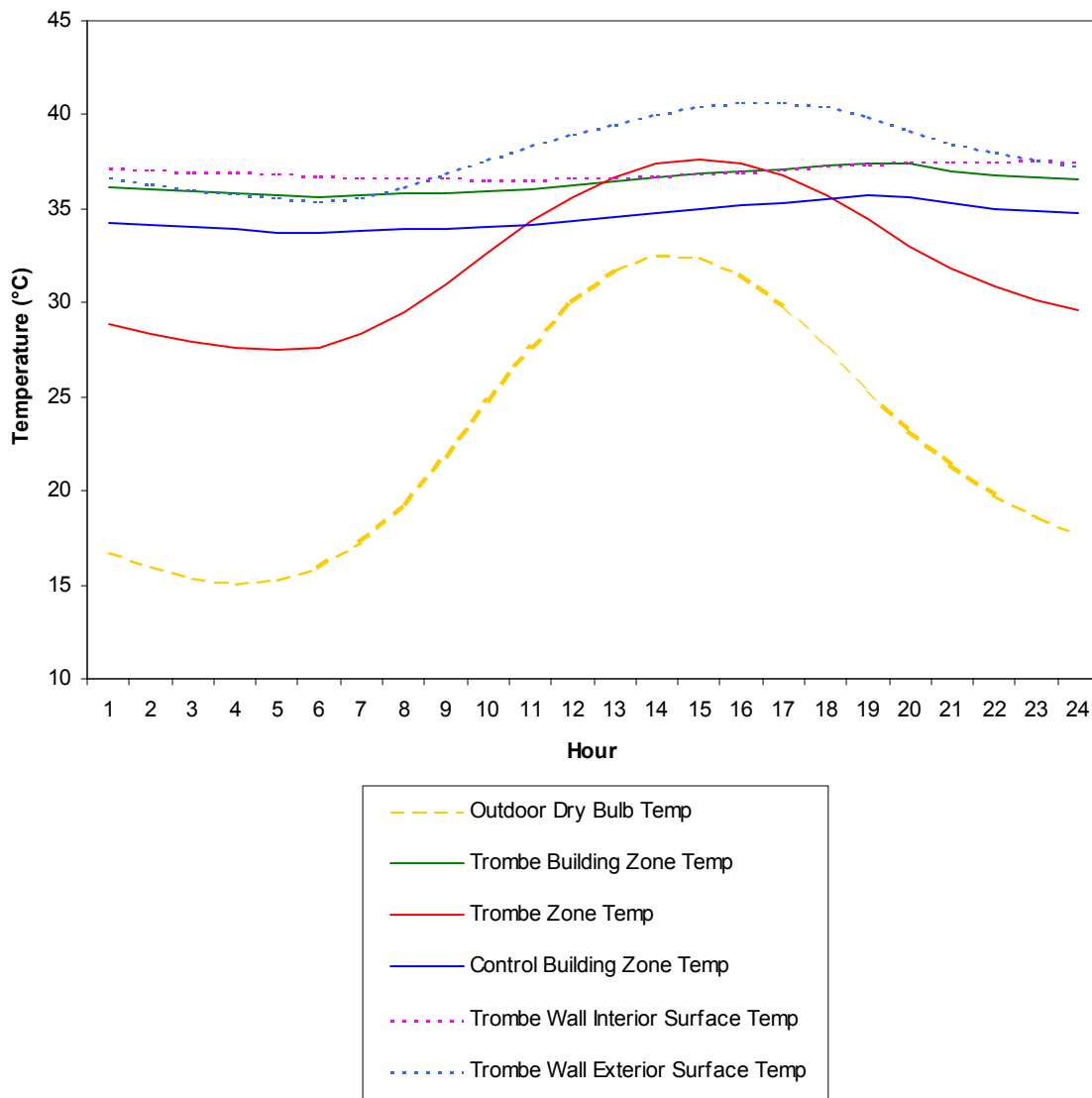


Figure 222. Passive Trombe Wall Summer

References

Ellis, Peter G. 2003. *Development and Validation of the Unvented Trombe Wall Model in EnergyPlus*, Master's Thesis, University of Illinois at Urbana-Champaign.

ISO 15099. 2000. "Thermal Performance of Windows, Doors, and Shading Devices-Detailed Calculations". International Standards Organization, Draft, July 18, 2000.

Active Trombe Wall

The active Trombe wall is the same as the passive Trombe wall with the addition of a simple fan system to circulate air between the Trombe zone and the main zone. The fan is scheduled to operate only during winter daylight hours to improve the heat transfer from the Trombe zone.

As with the passive Trombe wall, there is no EnergyPlus object for the active Trombe wall; it is simulated using a collection of other EnergyPlus objects. Like the passive Trombe wall,

the active Trombe wall uses a narrow zone coupled to the main zone with interzone partitions. However, the unique part of the active Trombe wall is that the Trombe zone is used to define a zone supply plenum object which allows the Trombe zone to be integrated into the air system. A constant volume fan is the main component of the air system. To make the zone connections, the Direct Air component is used.

For the active Trombe wall, there is no built-in algorithm for calculating the correct convection coefficients due to forced convection on the inside of the cavity walls. One approach is to use the SurfaceProperty:ConvectionCoefficients object to schedule coefficients that have been determined beforehand by the user.

Input File

An input file (ActiveTrombeWall.idf) is provided to demonstrate a sample active Trombe wall implementation. The building and Trombe wall in this file are identical to the ones described above for PassiveTrombeWall.idf. However, this input file adds a system in the form of a low flow rate ($0.1 \text{ m}^3/\text{s}$) constant volume fan and the necessary duct connections. The fan is scheduled to operate October through March from 10 AM to 8 PM.

Results

The resulting temperature profile for the winter design day is plotted below. The plot for the summer design day is not shown because it is identical to Figure 222 above since the fan is not scheduled to operate in the summer.

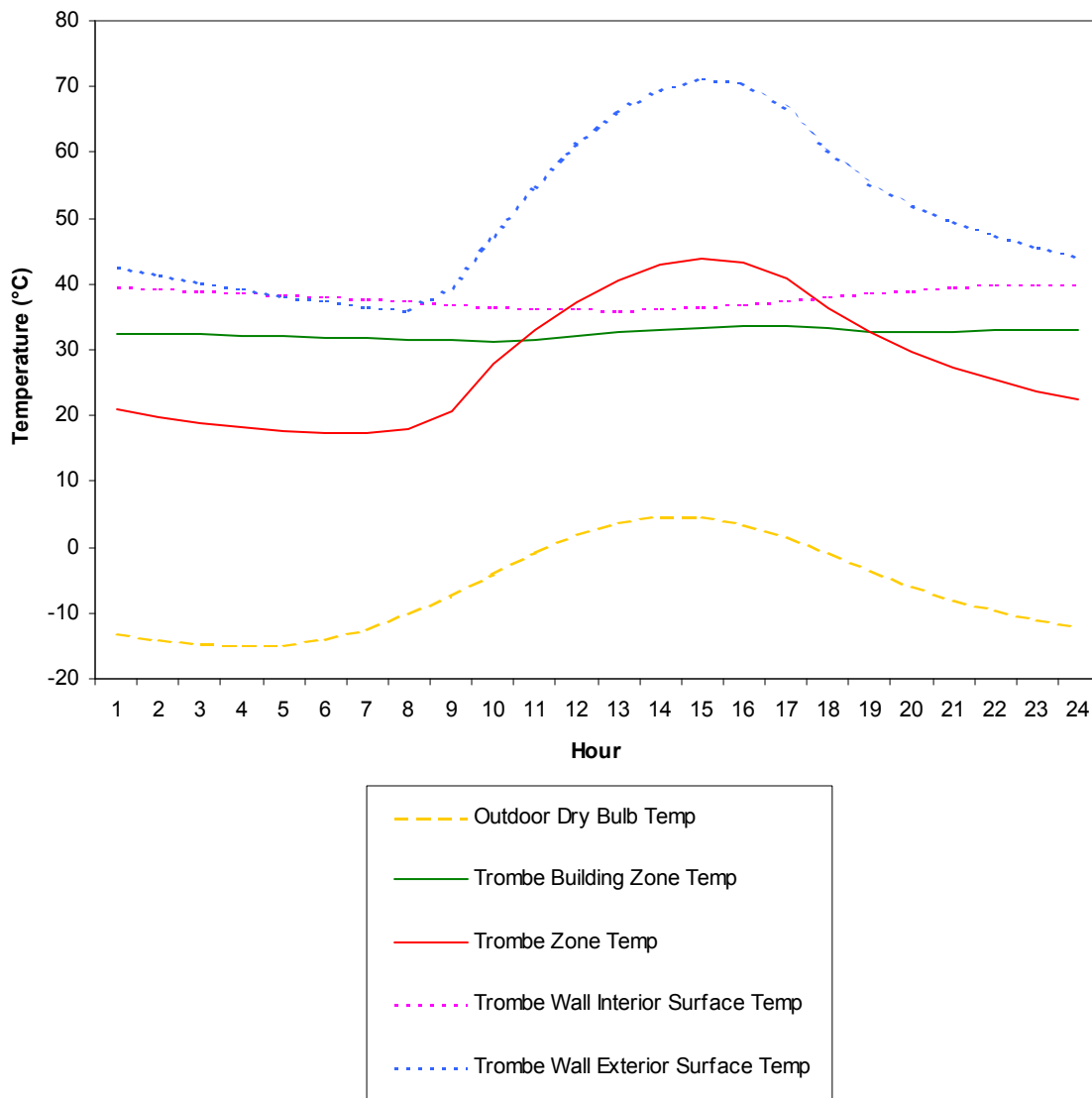


Figure 223. Active Trombe Wall Winter

Water Thermal Tanks (includes Water Heaters)

Water thermal tanks are devices for storing thermal energy in water. The most common types are water heaters, devices for storing and heating water. Typical water heater applications are for domestic hot water heating, low-temperature radiant space heating, and energy storage for solar hot water systems or waste heat recovery. In EnergyPlus, water heater objects can be coupled to a plant loop simulation or used stand-alone. There are also chilled water storage tanks that can be used to hold cold water.

Mixed Water Thermal Tank

The input object WaterHeater:Mixed provides a model that simulates a well-mixed water tank, i.e. non-stratified, and is appropriate for simulating many types of water heaters and storage tanks, including gas and electric residential water heaters, a variety of large commercial water heaters, and also instantaneous, tankless water heaters. This model is used for both the mixed water heater and the mixed chilled water storage tanks.

Energy Balance

The well-mixed assumption implies that all water in the tank is at the same temperature. To calculate the water temperature, the model analytically solves the differential equation governing the energy balance of the water tank:

$$\rho V c_p \frac{dT}{dt} = q_{net} \quad (649)$$

where

ρ = density of water

V = volume of the tank

c_p = specific heat of water

T = temperature of the tank water

t = time

q_{net} = net heat transfer rate to the tank water

The density and volume can be replaced with the total mass m of water in the tank to get:

$$m c_p \frac{dT}{dt} = q_{net} \quad (650)$$

The net heat transfer rate q_{net} is the sum of gains and losses due to multiple heat transfer pathways.

$$q_{net} = q_{heater} + q_{oncyclepara} + q_{offcyclepara} + q_{oncycleloss} + q_{offcycleloss} + q_{use} + q_{source} \quad (651)$$

where

q_{heater} = heat added by the heating element or burner

$q_{oncyclepara}$ = heat added due to on-cycle parasitic loads (zero when off)

$q_{offcyclepara}$ = heat added due to off-cycle parasitic loads (zero when on)

$q_{oncycleloss}$ = heat transfer to/from the ambient environment (zero when off)

$q_{offcycleloss}$ = heat transfer to/from the ambient environment (zero when on)

q_{use} = heat transfer to/from the use side plant connections

q_{source} = heat transfer to/from the source side plant connections

$q_{oncycleloss}$ and $q_{offcycleloss}$ are defined as:

$$q_{oncycleloss} = UA_{oncycle} (T_{amb} - T) \quad (652)$$

$$q_{offcycleloss} = UA_{offcycle} (T_{amb} - T) \quad (653)$$

where

$UA_{oncycle}$ = on-cycle loss coefficient to ambient environment (zero when off)

$UA_{offcycle}$ = off-cycle loss coefficient to ambient environment (zero when on)

T_{amb} = temperature of ambient environment

q_{use} , and q_{source} are defined as:

$$q_{use} = \varepsilon_{use} \dot{m}_{use} c_p (T_{use} - T) \quad (654)$$

$$q_{source} = \varepsilon_{source} \dot{m}_{source} c_p (T_{source} - T) \quad (655)$$

where

ε_{use} = heat exchanger effectiveness for the use side plant connections

\dot{m}_{use} = mass flow rate for the use side plant connections

T_{use} = inlet fluid temperature of the use side plant connections

ε_{source} = heat exchanger effectiveness for the source side plant connections

\dot{m}_{source} = mass flow rate for the source side plant connections

T_{source} = inlet fluid temperature of the use side plant connections

Incorporating all of these equations into the original differential equation,

$$mc_p \frac{dT}{dt} = q_{heater} + q_{oncycle} + q_{offcycle} + UA_{oncycle} (T_{amb} - T) + UA_{offcycle} (T_{amb} - T) \\ + \varepsilon_{use} \dot{m}_{use} c_p (T_{use} - T) + \varepsilon_{source} \dot{m}_{source} c_p (T_{source} - T) \quad (656)$$

Associating terms not dependent on temperature T and terms dependent on temperature T yields:

$$\frac{dT}{dt} = \left[\frac{1}{mc_p} \left(q_{heater} + q_{oncycle} + q_{offcycle} + UA_{oncycle} T_{amb} + UA_{offcycle} T_{amb} \right) \right] \\ + \left[\frac{-1}{mc_p} \left(UA_{oncycle} + UA_{offcycle} + \varepsilon_{use} \dot{m}_{use} c_p + \varepsilon_{source} \dot{m}_{source} c_p \right) \right] T \quad (657)$$

The differential equation now has the form

$$\frac{dT}{dt} = a + bT \quad (658)$$

where

$$a = \frac{1}{mc_p} \left(q_{heater} + q_{oncycle} + q_{offcycle} + UA_{oncycle} T_{amb} + UA_{offcycle} T_{amb} \right) \quad (659)$$

$$b = \frac{-1}{mc_p} \left(UA_{oncycle} + UA_{offcycle} + \varepsilon_{use} \dot{m}_{use} c_p + \varepsilon_{source} \dot{m}_{source} c_p \right) \quad (660)$$

The solution to the differential equation can be written in terms of a and b as:

$$T(t) = \left(\frac{a}{b} + T_i \right) e^{bt} - \frac{a}{b} \quad (661)$$

where

$T(t)$ = temperature of the tank water at time t

T_i = initial temperature of the tank water at time $t = 0$

However, if $b = 0$, the solution instead is:

$$T(t) = at + T_i \quad (662)$$

Since the control algorithm must sometimes calculate the time needed to reach a specified temperature, the equations above can also be rearranged to solve for t .

$$t = \frac{1}{b} \ln \left(\frac{\frac{a}{b} + T_f}{\frac{a}{b} + T_i} \right) \quad (663)$$

or, if $b = 0$,

$$t = \frac{T_f - T_i}{a} \quad (664)$$

where

T_f = final temperature of the tank water at time t .

In the special case where $b = 0$ and $a = 0$, and $T_f <> T_i$, the time t is infinity.

Water Heater Control Algorithm

For water heaters, control options allow the heater to cycle or modulate to meet the load. When cycling, the heater element or burner is either on or off. The heater remains fully on while heating the tank up to the setpoint temperature. When the setpoint is reached, the heater turns off. The heater remains off until the tank temperature falls below the "cut-in" temperature, i.e., the setpoint temperature minus the deadband temperature difference. The heater continuously cycles on and off to maintain the tank temperature within the deadband. Most storage-tank water heaters cycle.

When modulating, the heater power varies between the maximum and minimum heater capacities. The heater stays on as long as the required total demand is above the minimum capacity. Below the minimum capacity, the heater will begin to cycle on and off based on the deadband temperature difference. Most tankless/instantaneous water heaters modulate.

Within a time step, the differential equation is solved separately for when the heater element or burner is "on" (on-cycle) and when it is "off" (off-cycle). This approach allows ambient losses and parasitic loads to be divided into on-cycle and off-cycle effects and accounted for in detail.

An illustration of how the control algorithm cycles on and off is shown below. Ambient losses cool the tank temperature until the bottom of the deadband is reached (50 C) at which point the heater cycles on and reheats the tank back to the setpoint (60 C). A water draw causes hot water to be replaced with cold water from the water mains. The incoming cold water rapidly cools the tank. In this example the heater cannot keep up with the water draw and the tank temperature continues to drop until the water draw ends.

Although the instantaneous tank water temperature may vary considerably within a timestep (due to cycling, etc.), only the average temperature over the timestep is reported. The model calculates the average by piece-wise integration of the area under the instantaneous temperature curve for each unique set of conditions. The instantaneous temperature is preserved internally by the program and is propagated from the end of one timestep to the beginning of the next.

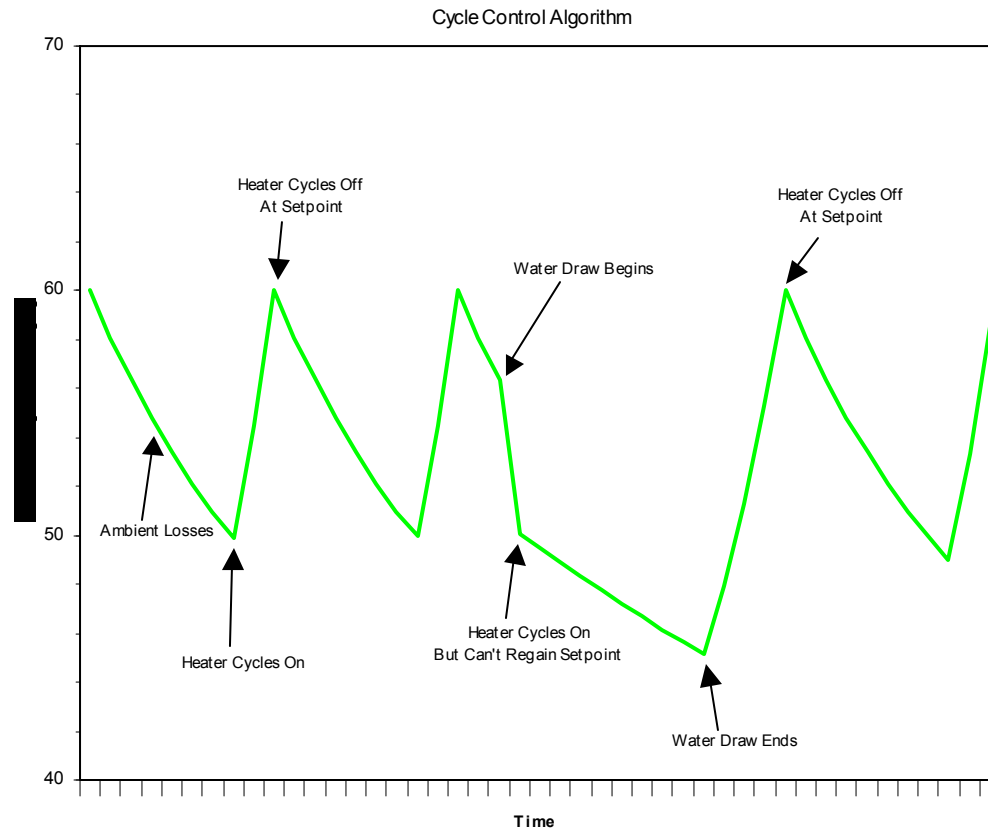


Figure 224. Water Heater Cycle Control Algorithm

Chilled Water Tank Control Algorithm

The input objects ThermalStorage:ChilledWater:Mixed and ThermalStorage:ChilledWater:Stratified provide chilled water tank models that do not include active cooling elements, there is only indirect cooling by remote devices such as a chiller. The tank's setpoint controls are used to determine if flow is to be requested through the source side of the tank. The setpoint and deadband control scheme is similar to the water heater but the logic is flipped around for cooling instead of heating. The setpoint temperature is the "cut-out" temperature and the setpoint plus deadband is the "cut-in" temperature. If the tank temperature (or tank sensing node for stratified tanks) is above the "cut-in" temperature, then flow is requested. If temperatures are below the "cut-out" temperature, then flow is not requested. The chilled water tanks also have separate availability schedules for the use side and source side for additional control options.

Standard Ratings

For water heaters, the industry standard ratings of Recovery Efficiency and Energy Factor are calculated according to the 10CFR430 test procedure. To emulate the test procedure, a 24-hour simulation of the water heater is performed internally using the specified test conditions:

- Setpoint Temperature = 57.2 C (135 F)
- Ambient Temperature = 19.7 C (67.5 F)
- Ambient Relative Humidity = 50% (used for heat pump water heaters)
- Inlet Temperature (Water Mains) = 14.4 C (58 F)

For heat pump water heaters, the water heater tank's heating element and water heater tank parasitic loads are disabled and the user-defined water heating capacity, energy use, and

parasitic loads for the heat pump are used to calculate Recovery Efficiency and Energy Factor.

The simulated test procedure performs six equal draws of approximately 0.041 m³ (10.7 gal) in each of the first six hours of the simulation. Each draw occurs over the first timestep of the hour.

The Recovery Efficiency is calculated when the water heater recovers to the setpoint after the first draw.

$$RE = \frac{m_1 c_p (57.2 - 14.4)}{E_1} \quad (665)$$

where

m_1 = water mass of the first draw

c_p = specific heat of water

E_1 = fuel energy consumed until the setpoint is recovered (including parasitics)

Note: When the standards rating for a heat pump water heater is calculated, the fuel energy consumed refers to the total energy consumed by the heat pump compressor, evaporator fan, condenser pump, and parasitic loads. It is assumed that the parasitic loads for a heat pump water heater do not contribute to heating the water (ref. Heat Pump Water Heater).

The Energy Factor is calculated at the end of the 24-hour simulation period.

$$EF = \frac{m_{total} c_p (57.2 - 14.4)}{E_{total}} \quad (666)$$

where

m_{total} = total water mass of all six draws

c_p = specific heat of water

E_{total} = total fuel energy consumed over 24 hours (including parasitics)

Under certain input parameters, the rating method will not succeed and a warning message will be generated. Problems occur when inputs do not allow the tank to recover to the setpoint temperature within the test period. This can occur if the maximum heater capacity is undersized, or if the deadband temperature difference is large enough that the first draw of the test does not trigger the heater to come on. In either case, the Recovery Efficiency test will not compute properly because recovery to the setpoint was not achieved.

References

10CFR430. *Title 10, Code of Federal Regulations, Part 430 - Energy Conservation Program for Consumer Products, Appendix E to Subpart B - Uniform Test Procedure for Measuring the Energy Consumption of Water Heaters.*

Heat Pump Water Heater

Overview

The input object WaterHeater:HeatPump provides a model for a heat pump water heater (HPWH) that is a compound object consisting of a water heater tank (e.g., WaterHeater:Mixed), a direct expansion (DX) "coil" (i.e., an air-to-water DX compression system which includes a water heating coil, air coil, compressor, and water pump), and a fan

to provide air flow across the air coil associated with the DX compression system. These objects work together to model a system which heats water using zone air, outdoor air, or a combination of zone and outdoor air as the primary heat source.

Numerous configurations of tank location, inlet air source, and DX coil compressor location can be modeled. The DX coil compressor may be located in a zone, outdoors, or the ambient temperature surrounding the compressor may be scheduled. The location of the compressor controls the operation of its crankcase heater. The water heater tank location is specified in the water heater tank object and is independent of the compressor location. In addition, the inlet air configuration may be specified in one of several ways. The heat pump water heater air coil and fan assembly may draw its inlet air from the zone and outdoor air using an optional mixer and splitter assembly as shown in the first figure below. When used, the mixer and splitter air streams are controlled by a single inlet air mixer schedule. When the HPWH draws its inlet air solely from a zone, the mixer/splitter assembly is not required as shown in the second figure below. In this case, the inlet air to the evaporator and fan assembly is made up entirely of zone air and the heat pump outlet air is directed back to the zone. The final figure illustrates a HPWH that draws its inlet air solely from outdoors and exhausts its outlet air outdoors as well. Each of these configurations may also be connected to a plant hot water loop (via the water heater tank use nodes).

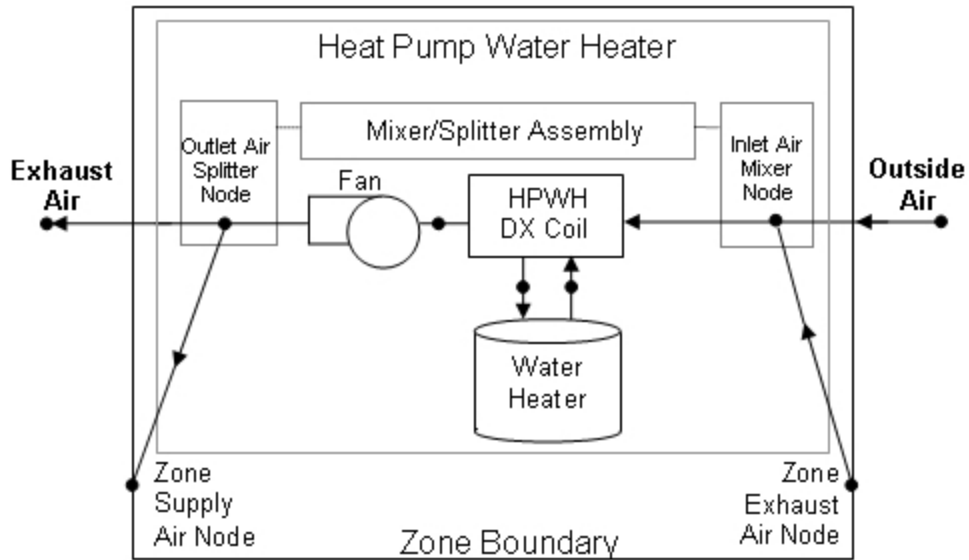


Figure 225. Schematic of a heat pump water heater using optional mixer/splitter nodes

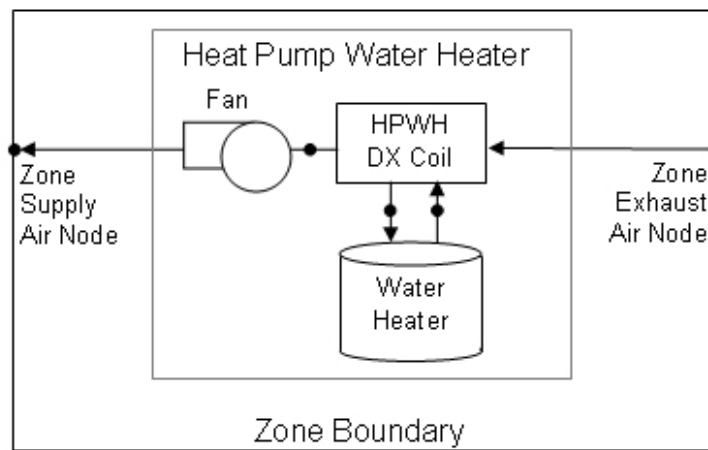


Figure 226. Schematic of a Heat Pump Water Heater with Inlet Air from a Zone

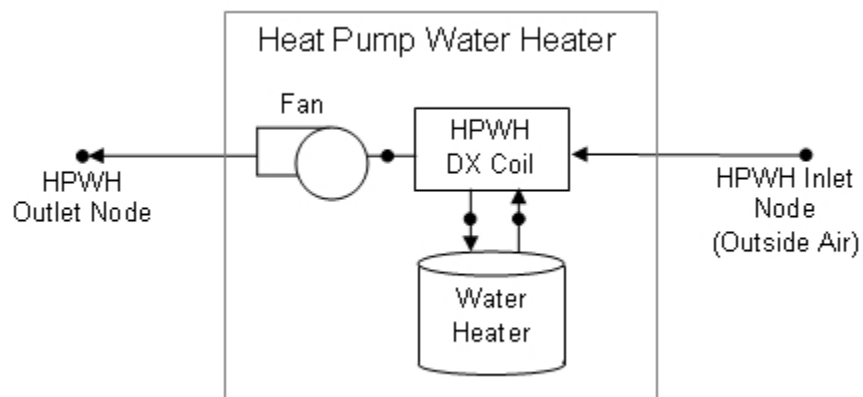


Figure 227. Schematic of a Heat Pump Water Heater with Inlet Air from Outdoors

Note: The water heater tank location shown in the figures above is completely independent of the heat pump water heater's inlet air configuration and its compressor location. The water heater tank may be located outdoors, in a zone, or the ambient temperature surrounding the tank can be scheduled as described in the mixed water heater section below.

Model Description

The heat pump water heater input requires a compressor setpoint temperature schedule and dead band temperature difference, which are independent from the setpoint temperature schedule and dead band temperature difference for the heater (element or burner) associated with the water heater tank. The cut-in temperature for the heat pump compressor is defined as the heat pump compressor's setpoint temperature minus its dead band temperature difference.

$$T_{HP, cut-in} = T_{HP, set\ point} - T_{HP, dead\ band}$$

where:

$T_{HP, cut-in}$ = Cut-in temperature for the heat pump compressor (°C)

$T_{HP, set\ point}$ = Heat pump compressor setpoint temperature (°C)

$T_{HP, dead\ band}$ = Heat pump compressor dead band temperature difference (°C)

In this model, the heat pump water heater's DX compression system is considered the primary heat source and the water tank's heater (element or burner) provides supplemental heat as necessary. Therefore, the cut-in temperature for the heat pump compressor (setpoint minus dead band temperature difference) is usually higher than the setpoint temperature for the heater (element or burner) in the associated water heater tank object. At times when the water heater tank setpoint temperature is greater than the cut-in temperature of the heat pump compressor, the heat pump compressor is disabled and the tank's heater is used to heat the water.

The simulation starts by first calculating the air conditions entering the air coil (evaporator)/fan assembly based on the inlet air configuration of the heat pump water heater and the presence of the optional mixer/splitter nodes. When the HPWH draws its inlet air from the zone and outdoors using the optional mixer/splitter nodes (i.e., Inlet Air Configuration = Zone and Outdoor Air), the inlet air conditions are calculated as follows:

$$Frac = GetScheduleValue(MixerInletAirSchedule)$$

$$T_{inlet} = T_{outdoor} (Frac) + T_{zone} (1 - Frac)$$

$$\omega_{inlet} = \omega_{outdoor} (Frac) + \omega_{zone} (1 - Frac)$$

where:

$Frac$ = current value of the inlet air mixer schedule (fraction of outdoor air, 0-1)

T_{inlet} = inlet air dry-bulb temperature to the HPWH evaporator/fan assembly (°C)

$T_{outdoor}$ = outdoor air dry-bulb temperature (°C)

T_{zone} = zone (exhaust) air dry-bulb temperature (°C)

ω_{inlet} = inlet air humidity ratio to the HPWH evaporator/fan assembly (kg/kg)

$\omega_{outdoor}$ = outdoor air humidity ratio (kg/kg)

ω_{zone} = zone (exhaust) air humidity ratio (kg/kg)

When the heat pump water heater draws its inlet air solely from the zone (i.e., Inlet Air Configuration = Zone Air Only), the inlet air conditions to the evaporator/fan assembly are simply set equal to the zone (exhaust) air conditions. If the heat pump water heater draws its inlet air solely from outdoors (i.e., Inlet Air Configuration = Outdoor Air Only), the inlet air conditions to the evaporator/fan assembly are simply set equal to the outdoor air conditions. When the inlet air to the heat pump water heater evaporator and fan assembly is scheduled (i.e., Inlet Air Configuration = Schedule), the inlet air conditions are determined directly from the user-supplied schedules as follows.

$$T_{inlet} = GetScheduleValue(InletAirTemperatureSchedule)$$

$$RH_{inlet} = GetScheduleValue(InletAirHumiditySchedule)$$

$$\omega_{inlet} = PsyWFnTdbRhPb(T_{inlet}, RH_{inlet}, OutBaroPress)$$

where:

RH_{inlet} = inlet air relative humidity to heat pump water heater evaporator/fan assembly (0-1)

$PsyWFnTdbRhPb$ = psychrometric function returning air humidity ratio given dry-bulb temperature, relative humidity, and barometric pressure

$OutBaroPress$ = outdoor barometric pressure (Pa)

For each simulation time step, the heat pump water heating capacity, energy use, and air-side/water-side mass flow rates are set to zero and the water heater tank is simulated with the heat pump compressor disabled when any of the following conditions apply:

- the HPWH is scheduled off by its availability schedule,
- the water heater tank setpoint temperature is greater than or equal to the heat pump compressor cut-in temperature,
- the inlet air dry-bulb temperature to the evaporator/fan assembly is less than the Minimum Inlet Air Temperature for Heat Pump Compressor Operation (as specified by the user in the HPWH input object), or
- the HPWH setpoint temperature is greater than or equal to the Maximum Temperature Limit (specified in the Water Heater:Mixed object).

Otherwise, simulation of the heat pump water heater is based on its current mode of operation. This mode of operation is either floating (heat pump compressor is off and tank water temperature has not fallen below the heat pump compressor cut-in temperature) or heating (tank water temperature dropped below the compressor cut-in temperature on a previous time step but was unable to reach the compressor setpoint temperature). Each mode is handled differently and they will be discussed separately.

Float Mode

When the heat pump water heater tank temperature is floating between the heat pump compressor's cut-in and cut-out temperatures at the end of the previous simulation time step, both the heat pump compressor and the water heater tank's heating element are disabled

and a resulting tank temperature is calculated. If the resulting tank temperature is below the heat pump compressor's cut-in temperature, the heat pump compressor part-load ratio is estimated using the ratio of the temperature differences shown below. The part-load ratio can not be less than zero or greater than one.

$$PLR = MIN \left(1.0, MAX \left(0.0, \left(\frac{T_{HP, cut-in} - T_{tank, float}}{T_{tank, initial} - T_{tank, float}} \right) \right) \right)$$

where:

PLR = part-load ratio of the heat pump water heater compressor

$T_{tank, float}$ = tank temperature in float mode when heating capacity is set to zero(°C)

$T_{tank, initial}$ = tank temperature at the beginning of the simulation time step (°C)

Since the pump and fan are assumed to cycle on and off with the heat pump compressor, the average condenser water and evaporator air mass flow rates for the simulation time step are calculated based on the PLR calculated above:

$$\dot{m}_{water, avg} = \dot{V}_{water} (\rho_{water}) (PLR)$$

$$\dot{m}_{air, avg} = \dot{V}_{air} (\rho_{air}) (PLR)$$

where:

$\dot{m}_{water, avg}$ = average condenser water mass flow rate for the time step (kg/s)

\dot{V}_{water} = condenser water volumetric flow rate, user input (m³/s)

ρ_{water} = density of condenser inlet water (kg/m³)

$\dot{m}_{air, avg}$ = average evaporator/fan air mass flow rate for the time step (kg/s)

\dot{V}_{air} = evaporator/fan air volumetric flow rate, user input (m³/s)

ρ_{air} = density of evaporator/fan inlet air (kg/m³)

The water tank temperature is then calculated based on heat pump operation at the part-load ratio estimated above and with the water tank's heating element enabled. If the resulting water tank temperature is above the heat pump compressor's setpoint (cut-out) temperature, then the part-load ratio is reduced and the water heater tank is simulated again. The process is performed iteratively until the part-load ratio of the heat pump compressor achieves the desired setpoint temperature (to the extent possible).

Heating Mode

When the HPWH is in heating mode at the end of the previous simulation time step (i.e., the heat pump compressor operated during the previous simulation time step but was unable to achieve the setpoint temperature), both the heat pump compressor and the water heater tank's heating element are enabled. The part-load ratio of the heat pump compressor is set to 1, and the condenser water and evaporator air mass flow rates are set to their maximum flow rates.

$$\dot{m}_{water, avg} = \dot{V}_{water} (\rho_{water})$$

$$\dot{m}_{air, avg} = \dot{V}_{air} (\rho_{air})$$

If the resulting tank temperature is above the heat pump compressor's setpoint (cut-out) temperature, the part-load ratio of the heat pump compressor is reduced and the water heater tank is simulated again. The process is performed iteratively until the part-load ratio of the heat pump compressor achieves the desired setpoint temperature (to the extent possible).

The air-side outlet conditions of the HPWH are calculated through simulation of the fan and DX coil with either a blow through or draw through fan placement (user selectable). If mixer/splitter nodes are used, the HPWH model splits the heat pump outlet air mass flow rate with exhaust air flow equaling the outdoor air flow, and the balance of the outlet air being sent to the zone supply air node (i.e., ensures that the heat pump water heater does not contribute to zone pressurization or depressurization). Calculations of heat pump water heating capacity, energy use, air-side performance, and water-side temperature difference are performed in the associated DX Coil object. See the engineering reference section for the Coil:WaterHeating:AirToWaterHeatPump object for details.

Model Outputs

After completing the float mode or heating mode calculations and the final part-load ratio has been determined, the output (report) variables are calculated as follows:

$$\text{Heat Pump Water Heater Compressor Part-Load Ratio} = PLR$$

$$\text{Heat Pump Water Heater On-Cycle Parasitic Electric Power (W)} = P_{parasitic, on} (PLR)$$

$$\text{Heat Pump Water Heater On-Cycle Parasitic Electric Consumption (J)} = P_{parasitic, on} (PLR)(3600)t_{sys}$$

$$\text{Heat Pump Water Heater Off-Cycle Parasitic Electric Power (W)} = P_{parasitic, off} (1 - PLR)$$

$$\text{Heat Pump Water Heater Off-Cycle Parasitic Electric Consumption (J)} = P_{parasitic, off} (1 - PLR)(3600)(t_{sys})$$

where:

$P_{parasitic, on}$ = on-cycle parasitic electric load, user input (W)

$P_{parasitic, off}$ = off-cycle parasitic electric load, user input (W)

t_{sys} = HVAC system simulation time step (hours)

Note: All heat pump water heater output variables, including off-cycle parasitic electric power and consumption, equal 0 when the heat pump water heater availability schedule equals 0 (i.e., the heat pump water heater is scheduled OFF).

Stratified Water Thermal Tank

The input objects WaterHeater:Stratified and ThermalStorage:ChilledWater:Stratified provide models for a stratified water thermal tank that divides the water tank into multiple nodes of equal volume. This model is used for both the stratified water heater and the stratified chilled water storage tank. The nodes are coupled by vertical conduction effects, internode fluid flow, and temperature inversion mixing. The object simultaneously solves the differential equations governing the energy balances on the nodes using the Forward-Euler numerical method. The system time step is divided into one second substeps that allow the simulation to capture events that occur on a very short time scale.

Energy Balance

Similar to the well-mixed model, the stratified model solves the same fundamental differential equation governing the energy balance on a mass of water:

$$mc_p \frac{dT}{dt} = q_{net}$$

where

m = mass of water

c_p = specific heat of water

T = temperature of water

t = time

q_{net} = net heat transfer rate

The difference for the stratified model is that it must solve the energy balance on n number of nodes simultaneously. Node 1 is at the top of the water tank and node n is at the bottom of the water tank.

$$m_n c_p \frac{dT_n}{dt} = q_{net,n}$$

where

m_n = mass of water for node n

c_p = specific heat of water

T_n = temperature of water for node n

t = time

$q_{net,n}$ = net heat transfer rate for node n

The net heat transfer rate q_{net} is the sum of gains and losses due to multiple heat transfer pathways.

$$q_{net,n} = q_{heater,n} + q_{oncyclepara,n} + q_{offcyclepara,n} + q_{oncycleloss,n} + q_{offcycleloss,n} \\ + q_{cond,n} + q_{use,n} + q_{source,n} + q_{flow,n} + q_{invmix,n}$$

where

$q_{heater,n}$ = heat added by Heater 1 or Heater 2

$q_{oncyclepara,n}$ = heat added due to on-cycle parasitic loads (zero when off)

$q_{offcyclepara,n}$ = heat added due to off-cycle parasitic loads (zero when on)

$q_{oncycleloss,n}$ = heat transfer to/from the ambient environment (zero when off)

$q_{offcycleloss,n}$ = heat transfer to/from the ambient environment (zero when on)

$q_{cond,n}$ = heat transfer due to conduction between the node above and below

$q_{use,n}$ = heat transfer to/from the use side plant connections

$q_{source,n}$ = heat transfer to/from the source side plant connections

$q_{flow,n}$ = heat transfer due to fluid flow from the node above and below

$q_{invmix,n}$ = heat transfer due to inversion mixing from the node above and below

$q_{oncycleloss,n}$ and $q_{offcycleloss,n}$ are defined as:

$$q_{oncycleloss,n} = UA_{oncycle,n} (T_{amb} - T_n)$$

$$q_{\text{offcycloss},n} = UA_{\text{offcyc},n} (T_{\text{amb}} - T_n)$$

where

$UA_{\text{oncyc},n}$ = on-cycle loss coefficient to ambient environment (zero when off)

$UA_{\text{offcyc},n}$ = off-cycle loss coefficient to ambient environment (zero when on)

T_{amb} = temperature of ambient environment

$q_{\text{cond},n}$ is defined as:

$$q_{\text{cond},n} = \frac{kA_{n+1}}{L_{n+1}} (T_{n+1} - T_n) + \frac{kA_{n-1}}{L_{n-1}} (T_{n-1} - T_n)$$

where

k = fluid thermal conductivity of water, 0.6 W/m-K

A_{n+1} = shared surface area between node n and node $n+1$

L_{n+1} = distance between the center of mass of node n and $n+1$

T_{n+1} = temperature of node $n+1$

A_{n-1} = shared surface area between node n and node $n-1$

L_{n-1} = distance between the center of mass of node n and $n-1$

T_{n-1} = temperature of node $n-1$

$q_{\text{use},n}$, and $q_{\text{source},n}$ are defined as:

$$q_{\text{use}} = \varepsilon_{\text{use}} \dot{m}_{\text{use}} c_p (T_{\text{use}} - T)$$

$$q_{\text{source}} = \varepsilon_{\text{source}} \dot{m}_{\text{source}} c_p (T_{\text{source}} - T)$$

where

ε_{use} = heat exchanger effectiveness for the use side plant connections

\dot{m}_{use} = mass flow rate for the use side plant connections

T_{use} = inlet fluid temperature of the use side plant connections

$\varepsilon_{\text{source}}$ = heat exchanger effectiveness for the source side plant connections

\dot{m}_{source} = mass flow rate for the source side plant connections

T_{source} = inlet fluid temperature of the use side plant connections

$q_{\text{flow},n}$ is defined as:

$$q_{\text{flow},n} = \dot{m}_{n+1} c_p (T_{n+1} - T_n) + \dot{m}_{n-1} c_p (T_{n-1} - T_n)$$

where

\dot{m}_{n+1} = mass flow rate from node $n+1$

\dot{m}_{n-1} = mass flow rate from node $n-1$

$q_{\text{invmix},n}$ is defined as:

$$q_{\text{invmix},n} = \dot{m}_{\text{invmix},n+1} c_p (T_{n+1} - T_n) + \dot{m}_{\text{invmix},n-1} c_p (T_{n-1} - T_n)$$

where

$\dot{m}_{invmix,n+1}$ = mass flow rate from node $n+1$ due to temperature inversion mixing

$\dot{m}_{invmix,n-1}$ = mass flow rate from node $n-1$ due to temperature inversion mixing

Inversion mixing occurs when the node below is warmer than the node above. The difference in temperatures drives a difference in density that causes the nodes to mix. Usually inversion mixing occurs very rapidly. In this algorithm, the inversion mixing rate is selected to be the maximum value that will provide a stable solution given the node mass and the substep interval:

$$\dot{m}_{invmix} = 0.5 * m_n / \Delta t$$

where

Δt = the substep time interval.

Numerical Solution

The system of simultaneous differential equations is solved using the Forward-Euler numerical method. The system time step is divided into one-second substeps. The new temperature for a given node is calculated using the following equation:

$$T_n = T_{n,old} + \frac{q_{net,n} \Delta t}{m_n c_p}$$

All node temperatures for $q_{net,n}$ are the old temperatures from the previous substep.

Before each system time step is calculated the following evaluations are made:

- Use and source inlet flow rates are applied to the inlet nodes
- Internode flow is determined and net flow rates are determined

Before each substep is calculated, the following evaluations are made:

- Thermostatic controls for heater 1 and heater 2 are evaluated to determine if the heater elements should turn on or off
- Node 1 temperature is compared against the maximum limit to determine if venting is necessary
- Adjoining node temperatures are compared to determine if there are any temperature inversions for which the inversion mixing rate should be used.

The solution continues looping through all substeps until the system time step is completed.

References

Duffie, J., and W. Beckman. 1980. *Solar Engineering of Thermal Processes*. John Wiley & Sons.

Newton, B. 1995. *Modeling of Solar Storage Tanks*. Master's Dissertation, University of Wisconsin-Madison.

Water Heating Sizing

Some inputs for water heaters can be autosized using the input object WaterHeater:Sizing. This section describes the sizing calculations for water heaters. There are six general methods for sizing tank volume and heating capacity.

Autosizing Tank Volume

The volume a water heater can be sized in the following ways depending on the design method chosen by the user.

- **Peak Draw.** The volume is determined from the loop design flow rate. The water heater is positioned on the supply side of a plant loop. After the plant sizing routines have run, the model obtains the design flow rate for all components on the demand side. The tank volume is then: $V = \dot{V}_{loop,des} * t_{draw}$,
- **Residential HUD-FHA Minimum.** The volume is determined from a set of rules defined by the table below. This is from Chapter 48 of 1999 ASHRAE Handbook HVAC Applications, American Society of Heating Refrigeration and Air-conditioning Engineers, Atlanta GA. (also used in the Building America Benchmark).

Table 57. Residential HUD-FHA Minimum

Residential HUD-FHA Minimum DHW Storage and Burner Capacity (ASHRAE 1999)												
# Bedrooms	1	2			3			4			5	6
# Bathrooms	All	≤1.5	2-2.5	≥3	≤1.5	2-2.5	≥3	≤1.5	2-2.5	≥3	All	All
Gas												
Storage (gal)	20	30	30	40	30	40	40	40	40	50	50	50
Burner (kBtu/hr)	27	36	36	36	36	36	38	36	38	38	47	50
Electric												
Storage (gal)	20	30	40	50	40	50	50	50	50	66	66	80
Burner (kW)	2.5	3.5	4.5	5.5	4.5	5.5	5.5	5.5	5.5	5.5	5.5	5.5

- **Per Person.** The tank volume is determined by summing the design level of people in the model and multiplying by a user-entered volume per person factor.
- **Per Floor Area.** The tank volume is determined by summing the floor area in all the zones in the model and multiplying by a user-entered volume per floor area factor.
- **Per Unit.** The tank volume is determined by multiplying a user-entered volume per unit and a user-entered number of units.
- **Per Solar Collector Area.** The tank volume is determined by summing the collector area in all the hot water solar collectors in the model and multiplying by a user-entered volume per collector area factor.

Autosizing Heater Capacity

The heater capacity can be sized in the following ways depending on the design method chosen by the user.

- **Peak Draw.** The heater capacity is determined from the tank volume, assumed start and finish temperatures and a user defined time for recovery. The heater capacity is then

$$q_{heat} = V \rho c_p (T_{finish} - T_{start})$$

where,

$$T_{finish} = 57.2C$$

$$T_{start} = 14.4C$$

- **Residential HUD-FHA Minimum.** The heater capacity is determined from a set of rules defined by the table above. This is from 1999 ASHRAE Handbook HVAC

Applications, Americal Society of Heating Refrigeration and Air-conditioning Engineers, Atlanta GA. (also used the Building America Benchmark).

- Per Person. The heater capacity is determined by summing the design level of people in the model and using a user-entered factor for recovery capacity per person. The heater capacity is then:

$$q_{heat} = \frac{N_{people} f_{RecoveryPerPerson} (T_{finish} - T_{start}) \rho c_p}{3600}$$

- Per Floor Area. The heater capacity is determined by summing the floor area in all the zones in the model and using a user-entered factor for recovery capacity per floor area. The heater capacity is then:

$$q_{heat} = \frac{A_{floors} f_{RecoveryPerFloorArea} (T_{finish} - T_{start}) \rho c_p}{3600}$$

- Per Unit. The heater capacity is determined from a user-entered Recovery Capacity per unit and a user-entered number of units. The heater capacity is then:

$$q_{heat} = \frac{N_{units} f_{RecoveryPerUnit} (T_{finish} - T_{start}) \rho c_p}{3600}$$

- Per Solar Collector Area. The water heater is assumed to be used for solar hot water storage and the heater capacity is set to zero.

Autosizing Tank Height

If the water heater is Stratified, then the geometry is important and the height of the tank can be scaled with changes in volume. For tank shapes of Vertical Cylinder, the user defined height aspect ratio, AR , is used to calculate a tank height, H , using

$$H = \left(\frac{4V AR^2}{\pi} \right)^{1/3}$$

Autosizing Plant Connection Flow Rates

When the water thermal tank is connected to a plant loop, it is convient to autosize the design volume flow rates through the plant connections. When the water thermal tank is connected to the supply side of plant loop and flow rates are autosized, the flow rate is the sum of the flow requests of all the various components on the demand side of that plant loop. When the water thermal tank is connected on the demand side of a plant loop (e.g. as for indirect water heating with a boiler) and flow rates are autosized, the design flow rates are calculated with the following equation,

$$\dot{V} = - \left(\frac{V}{t_{Recover} * 3600 * \varepsilon} \right) * Ln \left[\frac{(T_{PlantDesign} - T_{Setpoint})}{(T_{PlantDesign} - T_{start})} \right]$$

where

V = volume of the tank

$t_{Recover}$ = User parameter for the time it takes for the tank to recover from assumed starting temperature to an assumed setpoint temperature. For water heaters, the starting

temperature is 14.4°C and the final assumed setpoint temperature is 57.2°C. For chilled water tanks, the starting temperature is 14.4 °C and the final temperature is 9.0°C.

$$\mathcal{E} = \mathcal{E}_{use} \text{ OR } \mathcal{E}_{source}$$

$T_{PlantDesign}$ = the exit temperature specified in the Plant Sizing object

$T_{setpoint}$ = the final tank temperature of 57.2°C for heaters and 9.0°C for chilled water tanks.

T_{start} = the initial tank temperature of 14.4°C

If the demand side connections are autosized and a water heater's tank volume is autosized, then the problem cannot be readily solved in EnergyPlus because the demand side connection flows need to be reported earlier in the simulation and the tank volume is not yet available. This situation is resolved by using an interim, nominal tank volume for sizing the connections and the actual volume is calculated later in the simulation.

Water Systems

Water systems include a variety of components that simulate water consumption, production, and storage, including:

- Water Use Equipment and Connections
- Rainwater Collection
- Groundwater Well
- Water Storage Tanks

Water Mains Temperatures

The input object Site:WaterMainsTemperature provides a model for the temperature of water mains that deliver water to a building via underground pipes. The water mains temperatures are an important input for plant equipment that require make-up water from the water mains, such as for domestic hot water systems.

Water mains temperatures are a function of outdoor climate conditions and vary with time of year. A correlation has been formulated to predict water mains temperatures based on two weather inputs:

average annual outdoor air temperature (dry-bulb)

maximum difference in monthly average outdoor air temperatures

These values can be easily calculated from annual weather data using a spreadsheet.

The correlation was developed by Craig Christensen and Jay Burch and is described in Hendron et al. (2004). The correlation is:

$$T_{mains} = (T_{out,avg} + 6) + ratio * (\Delta T_{out,maxdiff} / 2) * SIN(0.986 * (day - 15 - lag) - 90)$$

where

T_{mains} = water mains temperature (°F)

$T_{out,avg}$ = average annual outdoor air temperature (°F)

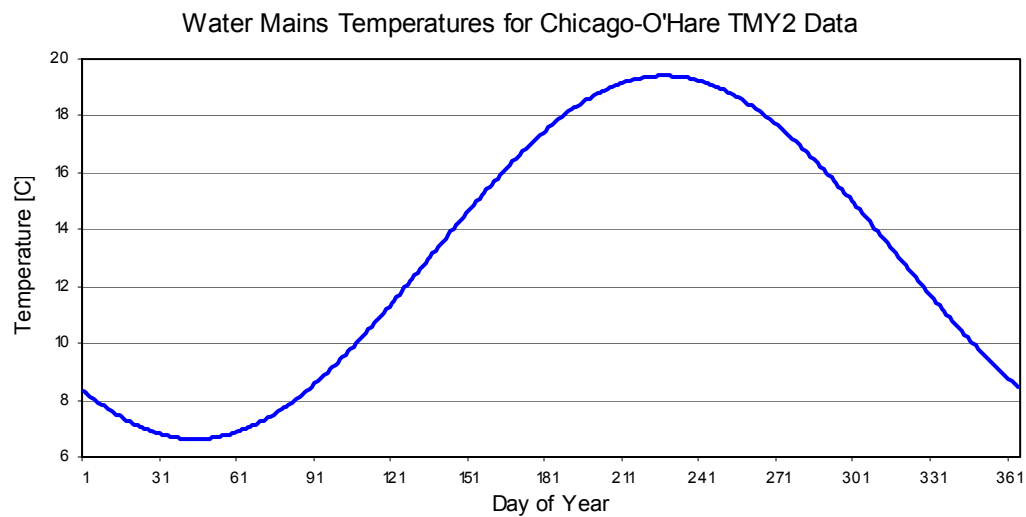
$\Delta T_{out,maxdiff}$ = maximum difference in monthly average outdoor air temperatures (°F)

day = Julian day of the year (1-365)

ratio = $0.4 + 0.01 * (T_{amb,avg} - 44)$

lag = $35 - 1.0 * (T_{amb,avg} - 44)$ (°F)

For the Chicago-O'Hare TMY2 weather file, $T_{out,avg} = 9.69$ C and $\Delta T_{out,maxdiff} = 28.1$ C. The resulting water mains temperature profile is graphed below.



References

Hendron, R., Anderson, R., Christensen, C., Eastment, M., and Reeves, P. 2004. "Development of an Energy Savings Benchmark for All Residential End-Uses", Proceedings of SimBuild 2004, IBPSA-USA National Conference, Boulder, CO, August 4 - 6, 2004.

Water Use Equipment and Connections

Essential parts of most water systems are the end uses where the water is utilized for a purpose. These end uses are characterized by familiar pieces of equipment such as sinks, showers, dishwashers, clothes washers, and toilets. End uses can also include outdoor uses such as landscape irrigation and agricultural irrigation.

Water end-use modeling in EnergyPlus is accomplished via two input objects:

- WaterUse:Equipment
- WaterUse:Connections

The WaterUse:Equipment object simulates all different types of water end uses. The WaterUse:Connections object manages multiple pieces of water equipment with a common supply and return, internally providing the services of a splitter and a mixer. The WaterUse:Connections object also allows water consuming equipment to be simulated in a closed plant loop by supplying makeup water from the water mains.

Unconnected Water Use Equipment

Common water equipment, such as sinks and showers, requires mixing of hot and cold water to provide a desired temperature at the tap. Water use equipment that is simulated in "unconnected" mode, i.e., without being referenced by a WaterUse:Connections object, presents a relatively simple modeling problem illustrated by the diagram below:

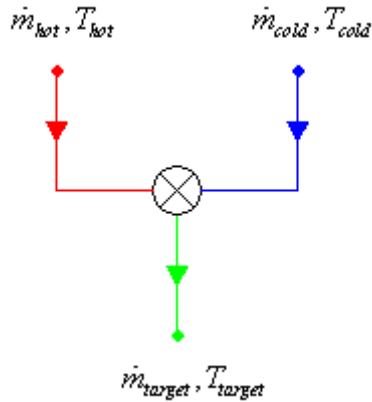


Figure 228. Hot and Cold Water Mixing

The variables are defined as:

\dot{m}_{hot} = hot water supply mass flow rate

T_{hot} = hot water supply temperature

\dot{m}_{cold} = cold water supply mass flow rate

T_{cold} = cold water supply temperature

\dot{m}_{target} = target mass flow rate at the tap

T_{target} = target temperature at the tap

The basic physics of the problem are described by two equations representing conservation of mass and conservation of energy respectively:

$$\dot{m}_{target} = \dot{m}_{hot} + \dot{m}_{cold} \quad (667)$$

$$\dot{m}_{target} T_{target} = \dot{m}_{hot} T_{hot} + \dot{m}_{cold} T_{cold} \quad (668)$$

The hot and cold water supply temperatures, T_{hot} and T_{cold} , are specified by the user with schedules (or indirectly from the water mains temperatures). The target flow rate and target temperature, \dot{m}_{target} and T_{target} , at the tap are also specified with schedules. The target conditions can be thought of as a way to simulate occupant behavior; a person at a sink will adjust the hot and cold flow rates to get the overall water flow and temperature that is desired.

The two equations above are easily solved to calculate the necessary hot and cold flow rates needed to attain the target conditions.

$$\dot{m}_{hot} = \dot{m}_{target} \left(\frac{T_{target} - T_{cold}}{T_{hot} - T_{cold}} \right) \quad (669)$$

$$\dot{m}_{cold} = \dot{m}_{target} - \dot{m}_{hot} \quad (670)$$

Even though hot and cold flow rates are unlimited in "unconnected" mode, it is still possible to fail to meet the target conditions if $T_{target} > T_{hot}$. In this case, the actual mixed water temperature at the tap, T_{mixed} , is set equal to T_{hot} . The target flow rate is always met.

Water equipment that omits schedules for the target temperature and/or hot water supply temperature implies that no hot water is needed. The result is all cold water at the target flow rate.

For "unconnected" water equipment, the heating rate and energy that is required to supply the hot water is calculated by the following equations.

$$Q = \dot{m}_{hot} c_p (T_{hot} - T_{cold}) \quad (671)$$

$$\Delta E = Q \Delta t \quad (672)$$

where

Q = heating rate

c_p = specific heat of water

ΔE = heating energy

Δt = time step interval

All heating is assigned to "Purchased Heating".

Zone Heat Gain from Water Use Equipment

Some water use equipment can be a significant source of sensible and latent heat gain when located in a zone. Showers, for example, add a noticeable latent gain as some of the hot water evaporates into the zone air (and at the same time adding some sensible gain). Other types of equipment may contribute greater sensible gain but less latent gain.

Calculating the amount of sensible and latent heat gain can be extremely complicated and depends on many variables. One set of variables is related to the initial conditions of the entering water and the undisturbed zone air, namely:

- Entering water temperature
- Entering water flow rate
- Ambient zone air temperature
- Ambient zone humidity
- Ambient zone barometric pressure

A second set of variables is tied to the specifics of the water equipment which describe the heat and mass transfer processes that occur as the water passes through the zone. These variables might include the velocity of the water in the zone air, the surface area of water exposed to the zone air, the dwell time of water in the zone, and possibly others. The complexity of these variables cannot be easily cast into a generalized EnergyPlus input object. Consider that in a shower, the velocity of water droplets is increasing as they fall, and the surface area of the spray is increasing. How to even determine the surface area of the spray at any point?

The approach taken here is to utilize the first set of initial condition variables and forego the overly-complex second set. The initial conditions can be used to calculate a maximum possible change in sensible and latent energy between the entering water conditions and the leaving water conditions at an infinite time later. Sensible and latent energy are calculated separately and represent the maximum heat gain, respectively, that could be added to the zone air. The user simply specifies a fraction of the maximum heat gain for sensible and latent that is actually added to the zone air. The fraction is specified with a schedule to

account for different modes of operation. The split between sensible and latent will vary depending on the type of equipment that is to be modeled. Typically, both fractions should be small numbers.

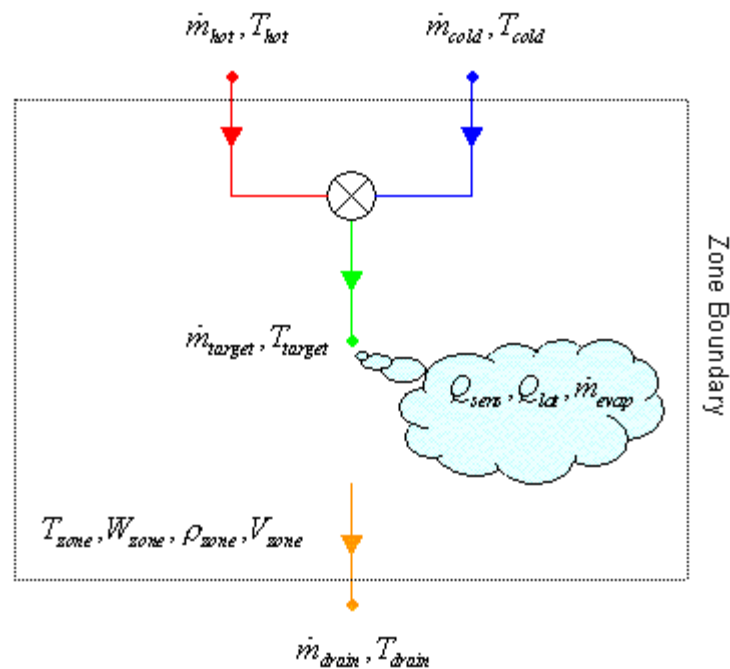


Figure 229. Zone Heat Gain from Water Use Equipment

The maximum sensible heat gain rate is calculated by assuming that all of the water flow is allowed to cool to the zone dry-bulb temperature. The actual sensible zone heat gain rate is found by multiplying the maximum by the user fraction.

$$Q_{sens} = f_{sens} \dot{m}_{target} c_p (T_{target} - T_{zone}) \quad (673)$$

$$\Delta E_{sens} = Q_{sens} \Delta t \quad (674)$$

where

Q_{sens} = sensible heat gain rate

f_{sens} = user fraction of maximum sensible heat

\dot{m}_{target} = target mass flow rate at the tap

c_p = specific heat of water

T_{target} = target temperature at the tap

T_{zone} = zone dry-bulb air temperature

ΔE_{sens} = sensible heat gain

Δt = time step interval

The maximum latent heat gain rate is calculated by assuming that the maximum amount of water is evaporated into the zone air during the time step. The actual latent zone heat gain rate is found by multiplying the maximum by the user fraction.

$$m_{evap,zonemax} = (W_{sat} - W_{zone}) \rho_{zone} V_{zone} \quad (675)$$

$$m_{evap,flowmax} = \dot{m}_{target} \Delta t \quad (676)$$

$$m_{evap,max} = MIN(m_{evap,zonemax}, m_{evap,flowmax}) \quad (677)$$

$$m_{evap} = f_{lat} m_{evap,max} \quad (678)$$

$$\dot{m}_{evap} = m_{evap} / \Delta t \quad (679)$$

$$Q_{lat} = h_{fg} \dot{m}_{evap} \quad (680)$$

$$\Delta E_{lat} = Q_{lat} \Delta t \quad (681)$$

where

$m_{evap,zonemax}$ = mass of water that can be absorbed by the zone air in one timestep

W_{sat} = fully-saturated zone humidity ratio (function of air temperature and pressure)

W_{zone} = zone humidity ratio

ρ_{zone} = zone dry-air density (function of air temperature and pressure)

V_{zone} = zone air volume

$m_{evap,flowmax}$ = mass of water available in the flow in one timestep

\dot{m}_{target} = target mass flow rate at the tap

Δt = time step interval

$m_{evap,max}$ = maximum mass of water that can be evaporated

m_{evap} = actual mass of water evaporated into the zone air

f_{lat} = user fraction of maximum latent heat

\dot{m}_{evap} = mass rate of water evaporation into the zone air

Q_{lat} = latent heat gain rate

h_{fg} = heat of vaporization for water

ΔE_{lat} = latent heat gain

Sensible and latent heat gains to the zone are lagged by one zone time step.

Note that this method allows the heat gain to vary dynamically according to changing water and zone conditions instead of the fixed or scheduled value that is traditionally used.

However, sensible and latent fractions may be difficult to estimate. Experimental measurement is probably the best way to determine the fractions.

A related result of any zone heat and moisture gain is that the water undergoes an equal heat and moisture loss. For a shower the water will generally leave the zone at a cooler temperature and slightly reduced flow rate when compared to the entering water conditions.

$$\dot{m}_{drain} = \dot{m}_{target} - \dot{m}_{evap} \quad (682)$$

$$T_{drain} = \frac{\dot{m}_{target} c_p T_{target} - Q_{sens} - Q_{lat}}{\dot{m}_{drain} c_p} \quad (683)$$

where

\dot{m}_{drain} = drainwater mass flow rate at the drain

\dot{m}_{target} = target mass flow rate at the tap

\dot{m}_{evap} = mass rate of water evaporation into the zone air

T_{drain} = water temperature at the drain

c_p = specific heat of water

T_{target} = target temperature at the tap

Q_{sens} = sensible heat gain rate

Q_{lat} = latent heat gain rate

The change in leaving water conditions has implications for drainwater heat recovery and storage tank conditions.

Connected Water Use Equipment

Water use equipment referenced by a WaterUse:Connections object is considered to be in "connected" mode. The WaterUse:Connections object manages multiple pieces of water equipment with a common supply and return, internally providing the services of a splitter and a mixer. The connections object also offers three internal configurations for drainwater heat recovery. External connections are available for coupling to plant loop nodes or water storage tanks for graywater reuse or reclamation.

Keep in mind that water use equipment in "connected" mode does not imply that it is connected to a plant loop. WaterUse:Equipment objects are never directly connected to a plant loop; they are connected indirectly through the WaterUse:Connections object. WaterUse:Connections can operate either within a plant loop or in stand-alone mode, i.e., without plant loop connections.

The WaterUse:Connections object overrides the hot and cold water supply temperatures of its subordinate WaterUse:Equipment objects based on the specified schedules, plant loop conditions, or water storage tank conditions. But the WaterUse:Equipment objects still maintain their individual target flow rates and target temperatures.

Solving the water subsystem described by the WaterUse:Connections object is appreciably more difficult than solving a single, unconnected piece of water use equipment. The subsystem is illustrated below.

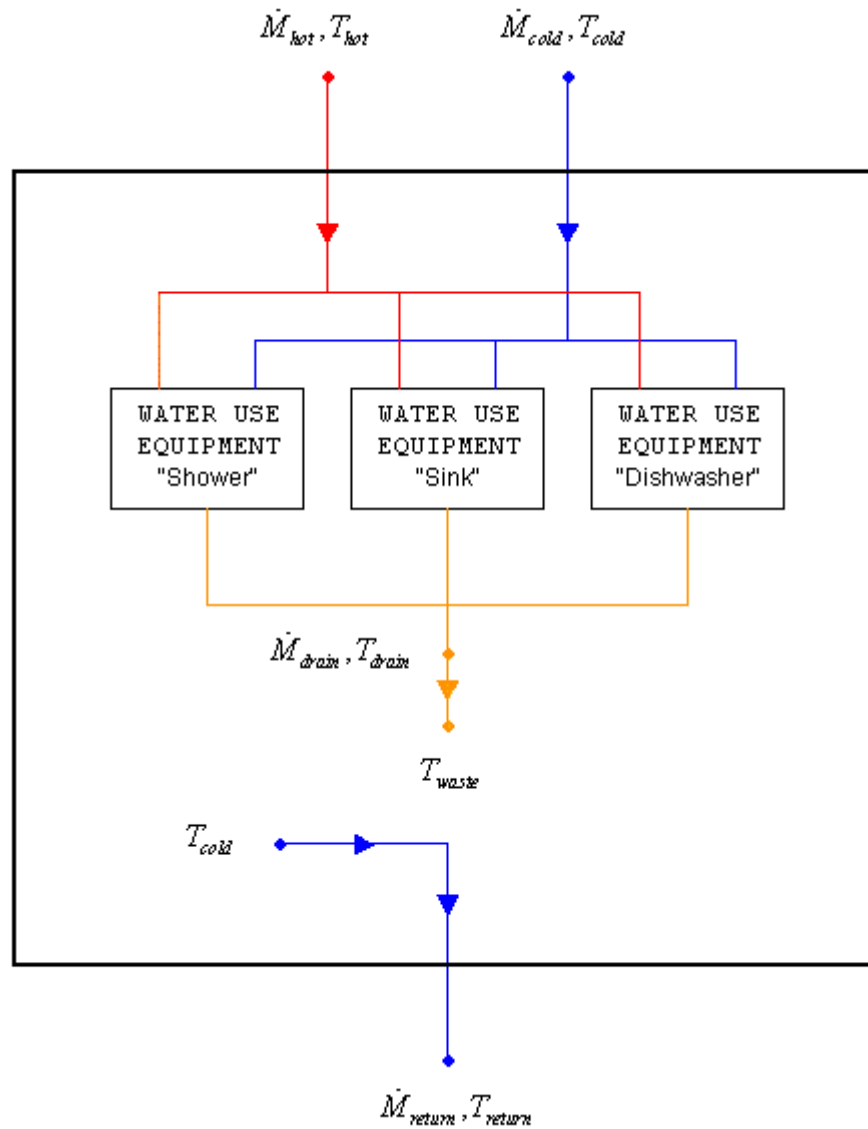


Figure 230. Water Use Connections Subsystem

The nomenclature here uses uppercase letters to indicate variables related to the connections object and lowercase letters to indicate variables related to the equipment objects. The variables are defined as:

\dot{M}_{hot} = hot water supply mass flow rate

T_{hot} = hot water supply temperature

\dot{M}_{cold} = cold water supply mass flow rate

T_{cold} = cold water supply temperature

\dot{M}_{drain} = drainwater mass flow rate at the drain

T_{drain} = water temperature at the drain

T_{waste} = wastewater temperature after the drain

\dot{M}_{return} = return water mass flow rate back to the plant loop

T_{return} = return temperature of makeup water back to the plant loop

Water Use Equipment Calculations

The solution method proceeds in steps:

- *InitConnectionsTemps*
- *CalcConnectionsFlowRates*
- *CalcConnectionsDrainTemp*
- *UpdateConnectionsNodes*

Initialize Connections Temperatures

Initializes the hot and cold supply water temperatures, T_{hot} and T_{cold} , for the common supply to all the equipment. If plant-coupled, T_{hot} is taken from the plant loop inlet node. If stand-alone, T_{hot} is taken from the schedule.

T_{cold} is taken from the supply water storage tank, if specified. Otherwise T_{cold} is taken from the schedule. If a schedule is not specified, T_{cold} is taken from the water mains.

Calculate Connections Flow Rates

First calls on each piece of equipment to simulate itself with the given supply water temperatures T_{hot} and T_{cold} . The desired hot and cold flow rates for $\dot{m}_{hot,i}$ and $\dot{m}_{cold,i}$ are calculated (as described earlier) and passed back to the connections object. The total flow rates for \dot{M}_{hot} and \dot{M}_{cold} are then calculated:

$$\dot{M}_{hot} = \sum_i \dot{m}_{hot,i} \quad \dot{M}_{cold} = \sum_i \dot{m}_{cold,i} \quad (684)$$

If plant-coupled, \dot{M}_{hot} is compared to the maximum flow rate available in the plant loop (after the first HVAC iteration). If $\dot{M}_{hot} > \dot{M}_{hot,max}$, the actual flow rate is reset to equal the maximum: $\dot{M}_{hot} = \dot{M}_{hot,max}$. New hot and cold flow rates, $\dot{m}'_{hot,i}$ and $\dot{m}'_{cold,i}$, at the equipment level are then recalculated, decreasing the hot flow rate and increasing the cold flow rate in order to still meet the target flow rate.

$$\alpha = \frac{\dot{M}_{hot,max}}{\dot{M}_{hot}} \quad (685)$$

$$\dot{m}'_{hot,i} = \alpha \dot{m}_{hot,i} \quad (686)$$

$$\dot{m}'_{cold,i} = \dot{m}_{target,i} - \dot{m}'_{hot,i} \quad (687)$$

A new mixed water temperature is also recalculated at the equipment level.

Although water storage tanks can also have a maximum flow rate on the cold side, the cold flow rate is not limited in this implementation.

Calculate Connections Drain Temperature

At this point zone heat gains are calculated for all equipment in zones. The final drainwater temperatures and flow rates are determined for each piece of equipment by subtracting the lost heat and moisture (see above). The total drainwater temperature and flow rate for the subsystem are calculated:

$$\dot{M}_{drain} = \sum_i \dot{m}_{drain,i} \quad T_{drain} = \frac{\sum_i \dot{m}_{drain,i} t_{drain,i}}{\dot{M}_{drain}} \quad (688)$$

In the case of no drainwater heat recovery, the subsystem wastewater temperature, T_{waste} , is equal to the drainwater temperature, T_{drain} . (For drainwater heat recovery, see below.) The wastewater temperature and flow rate are propagated to the reclamation water storage tank, if specified.

Update Connections Nodes

Finally, if plant-coupled, the return water temperature, T_{return} , is passed on to the plant outlet node. T_{return} is taken from the cold water supply schedule or the water mains. The return flow rate is equal to the hot water supply flow rate in order to preserve a closed plant loop:

$$\dot{M}_{return} = \dot{M}_{hot}$$

For "connected" water equipment, the heating rate and energy that is required to supply the hot water for individual water equipment is calculated by the following equations.

$$Q = \dot{m}_{hot} c_p (T_{hot} - T_{return}) \quad (689)$$

$$\Delta E = Q \Delta t \quad (690)$$

where

Q = heating rate

c_p = specific heat of water

ΔE = heating energy

Δt = time interval

The heating rate and energy for the WaterUse:Connections is the sum of the values for its constituent water equipment objects. If the WaterUse:Connections object is stand-alone, all heating is assigned to "Purchased Heating" by the individual water equipment objects. If the WaterUse:Connections object is coupled to a plant loop, all heating is supplied by the plant loop equipment.

Drainwater Heat Recovery

Drainwater heat recovery adds one more complication to the water connections subsystem. A heat exchanger is added between the drain and the waste outlet to recover some heat from the water before it leaves the system. The heat is usually recovered by preheating the incoming makeup water from the mains.

The heat exchanger can be modeled as "ideal", "counterflow", or "crossflow". One common type of drainwater heat exchanger is the gravity-film exchanger (GFX). The GFX can be approximated with a counterflow heat exchanger.

The destination of the preheated makeup water can be plumbed in three possible configurations: "plant", "equipment", "plant and equipment". In the "plant" configuration, all preheated water flow is returned to the plant loop to match the makeup water for the hot water supply flow. In the "equipment" configuration, all preheated water flow is directed internally within the WaterUse:Connections object to provide the makeup water for the cold water supply flow for the local water use equipment. In the "plant and equipment" configuration, the preheated water is split between both of the previous configurations. This is the only configuration where the flow rate is equal on both sides of the heat exchanger.

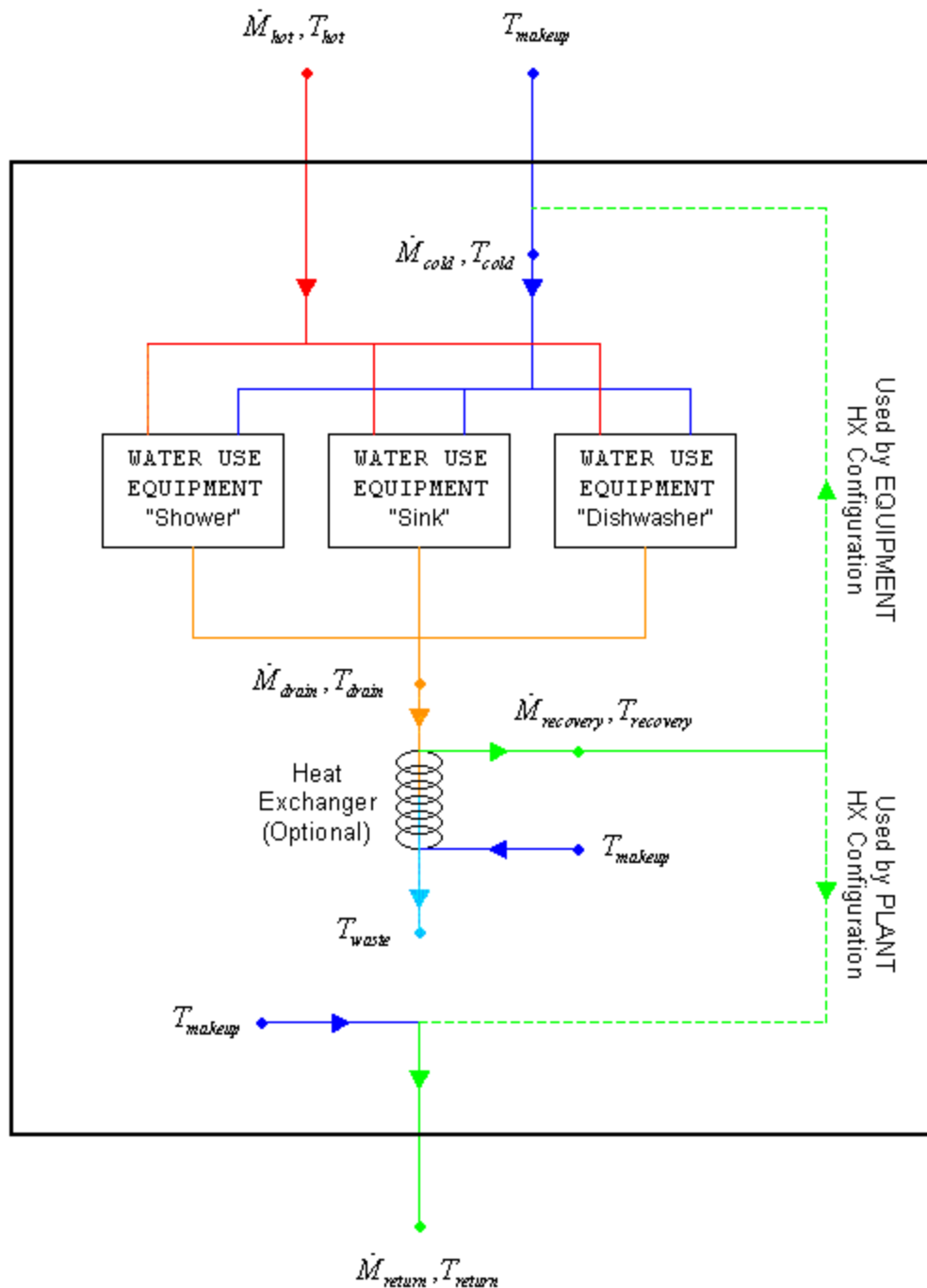


Figure 231. Water Use Connections Subsystem with Drainwater Heat Recovery

The new variables added here are defined as:

$\dot{M}_{recovery}$ = makeup water mass flow rate in the heat exchanger

$T_{recovery}$ = water temperature leaving the heat exchanger

T_{makeup} = cold makeup water temperature supplied to various flows

Another subroutine is inserted into the solution method:

- *InitConnectionsTemps*
- *CalcConnectionsFlowRates*
- *CalcConnectionsDrainTemp*
- ***CalcConnectionsHeatRecovery***
- *UpdateConnectionsNodes*

For the "equipment" and "plant and equipment" configurations, the solution requires iteration to solve because the preheated water leaving the heat exchanger is connected to the cold water supply water, creating a feedback loop.

Calculate Connections Heat Recovery

Based on the selected configuration, the flow rate through the heat exchanger is first determined. For the "plant" configuration, $\dot{M}_{recovery} = \dot{M}_{hot}$. For the "equipment" configuration, $\dot{M}_{recovery} = \dot{M}_{cold}$. For the "plant and equipment" configuration, $\dot{M}_{recovery} = \dot{M}_{hot} + \dot{M}_{cold}$.

The heat capacity rates on both sides of the heat exchanger are calculated and the minimum and maximum heat capacity rates, C_{min} and C_{max} , are determined.

$$C_{drain} = \dot{M}_{drain} c_p \quad (691)$$

$$C_{recovery} = \dot{M}_{recovery} c_p \quad (692)$$

$$C_{min} = MIN(C_{drain}, C_{recovery}) \quad (693)$$

$$C_{max} = MAX(C_{drain}, C_{recovery}) \quad (694)$$

$$R_c = \frac{C_{min}}{C_{max}} \quad (695)$$

where R_c = the heat capacity ratio.

The effectiveness ε is then calculated for the given type of heat exchanger. For the "ideal" heat exchanger, $\varepsilon = 1$.

For the "counterflow" and "crossflow" heat exchangers, the effectiveness is calculated using the effectiveness-NTU method, where:

$$NTU = \frac{UA}{C_{min}} \quad (696)$$

For the "counterflow" heat exchanger:

$$\varepsilon = \frac{NTU}{1 + NTU}, \text{ if } R_c = 1 \quad (697)$$

$$\varepsilon = \frac{1 - \text{EXP}(-NTU(1 - R_c))}{1 - R_c \text{EXP}(-NTU(1 - R_c))}, \text{ if } R_c < 1 \quad (698)$$

For the "crossflow" heat exchanger:

$$\varepsilon = 1 - \text{EXP}\left(\frac{NTU^{0.22}}{R_c} \left(\text{EXP}(-R_c NTU^{0.78}) - 1\right)\right) \quad (699)$$

The heat recovery rate is then calculated.

$$Q_{\text{recovery}} = \varepsilon C_{\text{min}} (T_{\text{drain}} - T_{\text{makeup}}) \quad (700)$$

The outlet water temperatures can then be calculated for both sides of the heat exchanger.

$$T_{\text{recovery}} = T_{\text{makeup}} + \frac{Q_{\text{recovery}}}{\dot{M}_{\text{recovery}} c_p} \quad (701)$$

$$T_{\text{waste}} = T_{\text{drain}} - \frac{Q_{\text{recovery}}}{\dot{M}_{\text{drain}} c_p} \quad (702)$$

At each iteration the difference between T_{recovery} and T_{cold} is compared and checked for convergence.

$$\Delta T_{\text{error}} = |T_{\text{recovery}} - T_{\text{cold}}| \quad (703)$$

Once the temperatures have converged, the solution is complete. T_{waste} is then passed on to a storage tank, if specified.

Zone Controls

Thermostatic Zone Control

The input object ZoneControl:Thermostat provides a way for the zone to be controlled to a specified temperature. ZoneControl:Thermostat references a control type schedule and one or more control type objects which in turn reference one or more setpoint schedules.

The control type schedule and the list of control type/name pairs are directly related. The schedule defines the type of control that is to be used during for each hour. Valid control types are

- 0 - Uncontrolled (No specification or default)
- 1 - Single Heating Setpoint
- 2 - Single Cooling SetPoint
- 3 - Single Heating/Cooling Setpoint
- 4 - Dual Setpoint (Heating and Cooling) with deadband

If the schedule referenced in the ZoneControl statement has a value of 4 for a particular hour, this indicates that during that hour "dual setpoint with deadband control" is to be used. The specific "dual setpoint with deadband" control object to be used is specified in the list of control type/name pairs. Then the specific control type objects reference the thermostat setpoint temperature schedule to be used. Because only one control can be specified for each control type in a ZoneControl statement, there are only four pairs possible in a particular ZoneControl type/name list. This is because individual controls can be defined hourly, thus giving the user a full range of flexibility. Since putting in the name of the control type directly in the schedule would be very cumbersome, the control types are assigned a number which is used in the hourly schedule profile.

For more information see ZoneControl:Thermostat in the InputOutput Reference and Zone Thermostats in the Engineering Documentation.

Zone Thermostats

The schema for the current set of four zone thermostats is given below. In each case, the keyword is accompanied by an identifying name and either one or two schedule names (depending on whether the control is a single or dual setpoint control). The schedule defines a temperature setpoint for the control type. The schedule would be defined through the standard schedule syntax described earlier in this document. For an uncontrolled zone no thermostat is specified or necessary. See the Input Output Reference for more details.

The control type schedule and the list of control type/name pairs are directly related. The schedule defines the type of control that is to be used during for each hour. Valid Control Types are

Control Type Value	Control Type Name
0	Uncontrolled (No specification or default)
1	Single Heating Setpoint (input object ThermostatSetpoint:SingleHeating)
2	Single Cooling SetPoint (input object ThermostatSetpoint:SingleCooling)
3	Single Heating/Cooling Setpoint (input object ThermostatSetpoint:SingleHeatingOrCooling)
4	Dual Setpoint (Heating and Cooling) with deadband (input object ThermostatSetpoint:DualSetpoint)

For the Uncontrolled case no heating or cooling requirement is calculated for the system to meet.

```

CASE (0)
! Uncontrolled Zone
LoadToHeatingSetPoint = 0.0
LoadToCoolingSetPoint = 0.0
ZoneSysEnergyDemand (ZoneNum) %TotalOutputRequired = 0.0

```

For the Single Heating Setpoint there would be a heating only thermostat. The setpoint can be scheduled and varied throughout the simulation but only heating is allowed with this control type.

```

CASE (SingleHeatingSetPoint)
! Determine zone load based on
! Qload + Qsys = 0 and Qsys = mCp(Tsys-Tzone)
! System Load Sign Convention:
!   - -> Cooling required to reach setpoint
!   + -> Heating required to reach setpoint

LoadToHeatingSetPoint = (TempDepZnLd(ZoneNum) * TempZoneThermostatSetPoint(ZoneNum) -
                        TempIndZnLd(ZoneNum))

IF ((ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired - 1.0) .LT. 0.0) &
    DeadBandOrSetback(ZoneNum) = .TRUE.

```

For the Single Cooling Setpoint there would be a cooling only thermostat. The setpoint can be scheduled and varied throughout the simulation but only cooling is allowed with this control type.

```

CASE (SingleCoolingSetPoint)

LoadToCoolingSetPoint = (TempDepZnLd(ZoneNum) * TempZoneThermostatSetPoint(ZoneNum) -
                        TempIndZnLd(ZoneNum))

ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = LoadToCoolingSetPoint

IF ((ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired + 1.0) .GT. 0.0) &
    DeadBandOrSetback(ZoneNum) = .TRUE.

```

For the Single Heat Cool Setpoint there would be a cooling only thermostat there would be a heating and cooling thermostat. The setpoint can be scheduled and varied throughout the simulation for both heating and cooling. With this control type only 1 setpoint profile is needed or used.

```

CASE (SingleHeatCoolSetPoint)

LoadToHeatingSetPoint = (TempDepZnLd(ZoneNum) * TempZoneThermostatSetPoint(ZoneNum) -
                        TempIndZnLd(ZoneNum))
LoadToCoolingSetPoint = (TempDepZnLd(ZoneNum) * TempZoneThermostatSetPoint(ZoneNum) -
                        TempIndZnLd(ZoneNum))

! Possible combinations:
! 1/ LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint > 0 --> Heating required
! 2/ LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint < 0 --> Not Feasible
! 3/ LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint < 0 --> Cooling Required
! 4/ LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint > 0 --> Dead Band Operation
IF (LoadToHeatingSetPoint .GT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
    ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = LoadToHeatingSetPoint
    ZoneSetPoint = ZoneThermostatSetPointLo(ZoneNum)
ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .LT. 0.0) THEN
    ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = LoadToCoolingSetPoint
    ZoneSetPoint = ZoneThermostatSetPointHi(ZoneNum)
ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
    ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = 0.0
    IF (Zone(ZoneNum)%SystemZoneNodeNumber > 0) THEN
        ZoneSetPoint = Node(Zone(ZoneNum)%SystemZoneNodeNumber)%Temp
    END IF
    DeadBandOrSetback(ZoneNum) = .TRUE.
END IF

```

For Dual Setpoint with DeadBand there would be a heating and cooling thermostat. For this case both a heating and cooling setpoint can be scheduled for any given time period. The setpoint can be scheduled and varied throughout the simulation for both heating and cooling.

```

CASE (DualSetPointWithDeadBand)

LoadToHeatingSetPoint = (TempDepZnLd(ZoneNum) * ZoneThermostatSetPointLo(ZoneNum) -
TempIndZnLd(ZoneNum))
LoadToCoolingSetPoint = (TempDepZnLd(ZoneNum) * ZoneThermostatSetPointHi(ZoneNum) -
TempIndZnLd(ZoneNum))

! Possible combinations:
! 1/ LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint > 0 --> Heating required
! 2/ LoadToHeatingSetPoint > 0 & LoadToCoolingSetPoint < 0 --> Not Feasible
! 3/ LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint < 0 --> Cooling Required
! 4/ LoadToHeatingSetPoint < 0 & LoadToCoolingSetPoint > 0 --> Dead Band Operation
IF (LoadToHeatingSetPoint .GT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = LoadToHeatingSetPoint
ZoneSetPoint = ZoneThermostatSetPointLo(ZoneNum)
ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .LT. 0.0) THEN
ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = LoadToCoolingSetPoint
ZoneSetPoint = ZoneThermostatSetPointHi(ZoneNum)
ELSE IF (LoadToHeatingSetPoint .LT. 0.0 .AND. LoadToCoolingSetPoint .GT. 0.0) THEN
ZoneSysEnergyDemand(ZoneNum)%TotalOutputRequired = 0.0
IF (Zone(ZoneNum)%SystemZoneNodeNumber > 0) THEN
ZoneSetPoint = Node(Zone(ZoneNum)%SystemZoneNodeNumber)%Temp
END IF
DeadBandOrSetback(ZoneNum) = .TRUE.
END IF

```

Operative Temperature Control

The input object ZoneControl:Thermostat:OperativeTemperature provides a model where the behavior of the zone thermostats described above can be modified to accommodate operative temperature control. The ZoneControl:Thermostat:OperativeTemperature input object is used to modify the behavior of any of the thermostatic control types described above. When this object is used, the zone can be controlled by a weighted mixture of radiant and air temperatures (rather than just zone air temperatures).

In EnergyPlus, operative temperature, T_{OP} , is defined as ,

$$T_{OP} = \gamma T_{MRT} + (1 - \gamma) T_{drybulb}$$

where,

γ is the radiative fraction,

T_{MRT} is the mean radiant temperature for the thermal zone, and

$T_{drybulb}$ is the mean zone air temperature.

The radiative fraction γ is selected by the user and can be scheduled to vary during the simulation. A typical value is 0.5. The maximum value needs to be less than 0.9 and the minimum is 0.0. A value of 0.0 is the same as controlling on only zone air temperature. If air velocities are higher than 0.2 m/s, then lower values for radiative fraction might apply. Niu and Burnett (1998) cite International Standard ISO 77300 in recommending the values for this fraction listed in the following table.

Table 58. Operative Temperature - Radiative Fraction vs Air Velocity

Air Velocity (m/s)	< 0.2	0.2 to 0.6	0.6 to 1.0
γ	0.5	0.4	0.3

The control is accomplished using the load to setpoint formulation that is fundamental to HVAC system control in EnergyPlus. Although real systems will not know the loads and operate on temperature-based control, EnergyPlus needs to be able to function at zone timesteps up to one hour and this is too long of a timeframe for true temperature-based zone

control. Load-based control is an important method of determining the average HVAC system response to zone conditions over the time step. For operative temperature control, the same methods and routines are used for calculating loads to setpoints but the setpoints are shifted to emulate operative temperature control rather than air temperature control.

The user-defined setpoint for T_{OP} is an input in degrees Celsius. T_{MRT} is continually modeled by the full zone heat balance method and includes all the interactions between thermal loads and HVAC systems. A new setpoint for the zone's $T_{drybulb}$ is obtained for each zone timestep by rearranging the definition of T_{OP} to put $T_{drybulb}$ on the left hand side.

$$T_{drybulb,setpoint} = \frac{T_{OP,setpoint} - \gamma T_{MRT}}{(1 - \gamma)}$$

This adjustment to the target zone air drybulb temperature is made every timestep for every zone immediately after the setpoints are obtained from the schedule.

Reference: J. Niu and J. Burnett. 1998. Integrating Radiant/Operative Temperature Controls into Building Energy Simulations. ASHRAE Transactions Vol. 104. Part 2. page 210. ASHRAE. Atlanta, GA.

Humidistat

The input object ZoneControl:Humidistat provides a way for the zone to be controlled to a single relative humidity setpoint schedule, or dual humidity schedules (humidifying and dehumidifying with deadband). The schedules consist of relative humidities, expressed as a percentage (0-100), to be used for the zone moisture prediction calculation. Only one control statement can be specified for each zone. Individual relative humidity values can be defined for every time step, thus giving the user a full range of flexibility. For a single setpoint humidistat, if the control relative humidity is below the calculated load and the equipment specified can humidify then that equipment will try and meet the requirement. The opposite is true if the calculated value is above the setpoint and the equipment can dehumidify. For a dual setpoint humidistat, if the zone relative humidity is below the **humidifying** relative humidity setpoint and the equipment specified can humidify then that equipment will try and meet the zone's humidification load. The opposite is true if the zone relative humidity is above the **dehumidifying** relative humidity setpoint and the equipment can dehumidify.

If the ZoneControl:Humidistat is used by a furnace or unitary system then no other objects are required. The signal from the humidistat is used directly by that component. If the ZoneControl:Humidistat is used to control a Humidifier or used in conjunction with the Controller:Simple object with control variable "TemperatureAndHumidityRatio", then either the "SetpointManager:SingleZone:Humidity:Minimum" or "SetpointManager:SingleZone:Humidity:Maximum" objects are required to determine a setpoint for those components to meet for the single setpoint humidistat. For a dual setpoint humidistat, the SetpointManager:SingleZone:Humidity:Minimum and SetpointManager:SingleZone:Humidity:Maximum objects are required to determine the setpoints for the corresponding humidification and dehumidification components. Note that the "SetpointManager:Scheduled" object can also be used to directly set humidity ratio setpoints on the exit node of the humidifier component.

For the single setpoint humidistat case, the model takes into account all of the moisture gains and/or losses from sources except the HVAC system contribution, and then calculates a moisture removal or addition rate based on the provided setpoint value, like the temperature predictor. The algorithm uses a 3rd Order derivative to predict zone moisture addition or removal to smooth the changes using the zone air capacitance. Positive values of moisture load mean that this amount of moisture must be added to the zone to reach the setpoint. Negative values represent the amount of moisture that must be removed by the system.

$$\text{MoistLoadHumidSetPoint} = \text{massflow} \cdot \text{HumidityRatio} = \text{kg air/sec} \cdot \text{kg H}_2\text{O/kg air} = \text{kg H}_2\text{O/sec}$$

This routine also calculates the zone required humidity ratio, W_{setpoint} , which is required and takes into account the zone multiplier. At this time the W_{sp} is not stored in a derived type to make it available for SetpointManagers. It was setup to be analogous to the temperature thermostat.

For the dual setpoint humidistat case, the model takes into account all of the moisture gains and/or losses from sources except the HVAC system contribution, and then calculates moisture addition and removal rates based on the provided humidifying and dehumidifying setpoint values, like the temperature predictor. The algorithm uses a 3rd Order derivative to predict zone moisture addition and removal to smooth the changes using the zone air capacitance. The program calculates the humidifying (addition) and dehumidifying (removal) moisture loads based on the specified humidifying and dehumidifying setpoints, respectively. The calculated humidifying moisture load is passed to subroutine CalcSingZoneMinHumSetPoint in the SetpointManager module to calculate the requested supply air humidity ratio at the setpoint node (Ref. SetpointManager:SingleZone:Humidity:Minimum object). The calculated dehumidifying moisture load is passed to subroutine CalcSingZoneMaxHumSetPoint in the SetpointManager module to calculate the requested supply air humidity ratio at the setpoint node (Ref. SetpointManager:SingleZone:Humidity:Maximum object). At the same time, the dehumidifying load is also passed directly to a furnace or unitary system if it is defined for the simulation, and that system will operate to meet this load based on its configuration and control settings.

Thermal Comfort Zone Control

The input object ZoneControl:Thermostat:ThermalComfort provides a way for the zone to be controlled to a specified temperature based on a selected thermal comfort model. ZoneControl:Thermostat:ThermalComfort references a thermal comfort control type schedule and one or more thermal comfort control type objects which in turn reference one or more setpoint schedules.

The thermal comfort control type schedule and the list of thermal comfort control type/name pairs are directly related. The schedule defines the type of thermal comfort control that is to be used during for each simulation time step. Valid Control Types are

Control Type	Control Type Name/Object
0	No thermal comfort control
1	ThermostatSetpoint:ThermalComfort:Fanger:SingleHeating
2	ThermostatSetpoint:ThermalComfort:Fanger:SingleCooling
3	ThermostatSetpoint:ThermalComfort:Fanger:SingleHeatingOrCooling
4	ThermostatSetpoint:ThermalComfort:Fanger:DualSetpoint

If the schedule referenced in the ZoneControl statement has a value of 4 for a particular time step, this indicates that during that time step "ThermostatSetpoint:ThermalComfort:Fanger:DualSetpoint" control is to be used. The specific "ThermostatSetpoint:ThermalComfort:Fanger:DualSetpoint" control object to be used is specified in the list of thermal comfort control type/name pairs. Then the specific thermal comfort control type objects reference the thermal comfort PMV setpoint schedule to be used. Because only one thermal comfort control can be specified for each thermal comfort control type in a ZoneControl statement, there are only four pairs possible in a particular ZoneControl type/name list. This is because individual thermal comfort controls can be defined for specific times, thus giving the user a full range of flexibility. Since putting in the name of the thermal comfort control type directly in the schedule would be very cumbersome, the thermal comfort control types are assigned a number which is used in the schedule profile.

For more information see ZoneControl:Thermostat:ThermalComfort in the InputOutput Reference and Zone Fanger Thermal Comfort in the Engineering Documentation.

Zone Control Based on Fanger Thermal Comfort

The syntax for the current set (4) of zone thermal comfort control types is given below. In each case, the keyword is accompanied by an identifying name and either one or two schedule names (depending on whether the control type is a single or dual setpoint control). The schedule defines a PMV setpoint for the control type. The schedule would be defined through the standard schedule syntax described earlier in this document. For an uncontrolled thermal comfort zone, no Fanger thermal comfort object is specified or necessary. See the Input Output Reference for more details.

The control type schedule and the list of control type/name pairs are directly related. The schedule defines the type of control that is to be used during for each hour. Valid Control Types are

Control Type	Control Type Name/Object
0	No thermal comfort control
1	ThermostatSetpoint:ThermalComfort:Fanger:SingleHeating
2	ThermostatSetpoint:ThermalComfort:Fanger:SingleCooling
3	ThermostatSetpoint:ThermalComfort:Fanger:SingleHeatingOrCooling
4	ThermostatSetpoint:ThermalComfort:Fanger:DualSetpoint

For the no thermal comfort control (uncontrolled) case, the control will revert to thermostat control if specified. If the thermal comfort control is specified as “uncontrolled” for a particular period and thermostat control is not specified in the input, then conditions will float.

For the ThermostatSetpoint:ThermalComfort:Fanger:SingleHeating case there would be a heating only thermal comfort setpoint temperature. The setpoint is calculated based on the selected thermal comfort model and varied throughout the simulation but only heating is allowed with this thermal comfort control type.

```
CASE (Single Thermal Comfort Heating Setpoint:Fanger)
TempControlType (ZoneNum) = SingleHeatingSetpoint
TempZoneThermostatSetPoint (ZoneNum) = Calculated Zone Setpoint from Fanger heating setpoint PMV
```

For the ThermostatSetpoint:ThermalComfort:Fanger:SingleCooling case there would be a cooling only thermal comfort setpoint temperature. The setpoint is calculated based on the selected thermal comfort model and varied throughout the simulation but only cooling is allowed with this thermal comfort control type.

```
CASE (Single Thermal Comfort Cooling Setpoint:Fanger)

TempControlType (ZoneNum) = SingleCoolingSetPoint
TempZoneThermostatSetPoint (ZoneNum) = Calculated Zone Setpoint from Fanger cooling setpoint PMV
```

For the ThermostatSetpoint:ThermalComfort:Fanger:SingleHeatingOrCooling there would be heating and cooling thermal comfort zone control objects. The setpoint is calculated based on the selected thermal comfort model and varied throughout the simulation for both heating and cooling. With this thermal comfort control type only 1 setpoint profile is needed or used.

```
CASE (Single Thermal Comfort Heating Cooling Setpoint:Fanger)

TempControlType (ZoneNum) = SingleHeatCoolSetPoint
TempZoneThermostatSetPoint (ZoneNum) = Calculated Zone Setpoint from Fanger heating and cooling
setpoint PMV
```

For ThermostatSetpoint:ThermalComfort:Fanger:DualSetpoint there would be heating and cooling thermal comfort control objects. For this case both a heating and cooling setpoint can be calculated based on the selected thermal comfort model for any given time period. The

thermal comfort setpoint temperature can be varied throughout the simulation for both heating and cooling.

```
CASE (Dual Thermal Comfort Setpoint with Deadband:Fanger)
```

```
TempControlType (ZoneNum) = DualSetPointWithDeadBand
```

```
ZoneThermostatSetPointLo (ZoneNum) = Calculated Zone Setpoint from Fanger heating setpoint PMV
```

```
ZoneThermostatSetPointHi (ZoneNum) = Calculated Zone Setpoint from Fanger cooling setpoint PMV
```

Zone Equipment and Zone Forced Air Units

Air Distribution Terminal Unit

Overview

The ZoneHVAC:AirDistributionUnit is a special piece of zone equipment – it connects centrally conditioned air with a zone. It encompasses the various types of air terminal units in EnergyPlus: *AirTerminal:DualDuct:ConstantVolume*, *AirTerminal:SingleDuct:VAV:Reheat*, etc. It is a generalized component that accesses the calculations for the different types of air terminal unit.

Model

The air distribution function is encapsulated in the module *ZoneAirEquipmentManager*. The object and module function only to invoke the individual air terminal unit simulations.

Inputs and Data

The data for this unit consists of the unit name, the air outlet node name (which should be the same as one of the zone inlet nodes), the type of air terminal unit (air distribution equipment), and the name of the air terminal unit.

All input data for air distribution units is stored in the array *AirDistUnit* in data module *DataDefineEquip*.

Calculation

There is no calculation associated with ZoneHVAC:AirDistributionUnit.

Simulation and Control

SimZoneAirLoopEquipment in module *ZoneAirEquipmentManager* calls the individual air terminal unit simulations.

References

No specific references.

Simple Duct Leakage Model

Overview

The input object ZoneHVAC:AirDistributionUnit also provides access to a model for duct leakage that can be a significant source of energy inefficiency in forced-air HVAC systems. Evaluating duct leakage energy losses can involve considerable user effort and computer resources if an airflow network is defined through a detailed description of the system components and airflow paths (including leakage paths). A nonlinear pressure-based solver is used to solve for pressures and flow rates in the network. By making certain assumptions and approximations for certain well defined configurations, however, it is possible to obtain accurate results with a simple mass and energy balance calculation and thus avoid the input and calculation costs of doing a full pressure-based airflow network simulation.

The Simple Duct Leakage Model (SDLM) assumes a central VAV air conditioning system with a constant static pressure setpoint. The model assumes that the leaks are in the supply

ducts and that the system returns air through a ceiling plenum that contains the ducts. Thus, the ducts leak into the return plenum, and this part of the supply does not reach the conditioned zones. With the additional assumptions described below, it is possible to model this configuration with heat and mass balance equations and avoid the use of a nonlinear pressure-based solver. In the EnergyPlus context, this means that use of AirflowNetwork is avoided and the leakage calculations are obtained in the course of the normal thermal simulation.

Principles and Description

Constant Flow Rate

The airflow rate through a duct leak is a function of the pressure difference between the duct and the surrounding space:

$$\dot{V}_{leak} = C_1 \cdot \Delta p_{duct-space}^n \quad (704)$$

The exponent n is 0.5 for leaks that look like orifices (holes that are large relative to the thickness of the duct wall); for leaks that resemble cracks (e.g., lap joints), n is approximately 0.6 to 0.65.

For a duct with constant flow rate and a linear pressure drop through the duct, the average static pressure in the duct will equal half of the duct static pressure drop. Assuming turbulent flow in the duct, the duct pressure drop is proportional to the square of the airflow through the duct. This can be expressed as:

$$\Delta p_{duct-space} = \frac{\Delta p_{duct}}{2} = C_2 \left(\frac{\dot{V}_{duct}}{2} \right)^2 \quad (705)$$

Combining equations (704) and (705) and assuming the leaks are large holes (n equals 0.5), gives:

$$\dot{V}_{leak} = C_1 \cdot \Delta p_{duct-space}^{0.5} = C_3 \cdot \dot{V}_{duct} \quad (706)$$

where

$$C_3 = C_1 \cdot (C_2 / 2)^{0.5} \quad (707)$$

Thus the leakage fraction C_3 remains constant regardless of the duct flow rate or static pressure. This result depends on the following assumptions:

- the duct airflow is turbulent;
- the duct pressure varies linearly along the duct;
- the average duct pressure approximates the pressure drop across the duct;
- the leaks are large and have pressure exponent 0.5.

Effects of Constant Pressure Upstream and Variable Flow and Pressure Downstream

Commonly VAV systems maintain a constant static pressure at some point in the duct system upstream of the VAV terminal units. That is, airflow rate will vary depending on the cooling requirement, but a constant pressure will be maintained at the static pressure sensor. Consequently, the leakage flow for a leak upstream of the VAV boxes will be approximately constant. Or to put it another way, the leakage fraction will vary in proportion to the flow rate.

For leaks downstream of the VAV terminal units, the airflow through the duct and the pressure in the downstream duct will vary as the box damper modulates in response to the differential between the room temperature and the thermostat setpoint. In this case, the situation is similar to the constant flow case: for an orifice-like leak, the pressure difference across the leak will vary linearly with the air speed (or flow rate); i.e., the leakage fraction will be approximately constant.

SDLM

For SDLM, our leakage model is then:

1. for leaks upstream of the terminal units, the leakage flow rate will be constant;
2. for leaks downstream of the terminal units, the leakage fraction will be constant.

This model assumes, in addition to the assumptions given above, that the VAV system is controlled to a constant static pressure setpoint. In EnergyPlus SDLM is not currently applicable to systems using static pressure reset. Using SDLM would require knowledge of static pressure as a function of system air flow rate.

Inputs and Data

User data for the SDLM is entered through The ZoneHVAC:AirDistributionUnit (ADU) object. There are 2 data items per ADU:

1. the upstream nominal leakage fraction;
2. the downstream fixed leakage fraction.

Both inputs are leakage fractions. Input (1) is the leakage fraction at design flow rate, which together can be used to determine the constant leakage flow rate upstream of the VAV boxes; this leakage fraction varies with the flow rate. Input (2) is a fixed leakage fraction and is constant as the flow rate varies.

Implementation

The various zone mass flow rates are related in the following manner.

$$\dot{m}_{s,us} = \dot{m}_{tu} + \dot{m}_{lk,us} \quad (708)$$

$$\dot{m}_{tu} = \dot{m}_{lk,ds} + \dot{m}_{s,z} \quad (709)$$

$$\dot{m}_{lk,us} = Frac_{us} \cdot \dot{m}_{s,us,max} \quad (710)$$

$$\dot{m}_{lk,ds} = Frac_{ds} \cdot \dot{m}_{tu} \quad (711)$$

Here

$\dot{m}_{s,us}$ is the constant zone supply air mass flow rate upstream of the leaks [kg/s];

\dot{m}_{tu} is the air mass flow rate through the terminal unit [kg/s];

$\dot{m}_{lk,us}$ is the upstream leakage air mass flow rate [kg/s];

$\dot{m}_{lk,ds}$ is the downstream leakage air mass flow rate [kg/s];

$\dot{m}_{s,us,max}$ is the maximum upstream supply air mass flow rate (program input) [kg/s];

$\dot{m}_{s,z}$ is the supply air mass flow rate delivered to the zone [kg/s];

$Frac_{us}$ is the design upstream leakage fraction (program input);

$Frac_{ds}$ is the constant downstream leakage fraction (program input);

\dot{m}_{tu} is calculated in the VAV terminal unit model in the usual manner: the mass flow rate is varied to meet the zone load. The limits on the mass flow rate variation are set by the $\dot{m}_{MaxAvail}$ and $\dot{m}_{MinAvail}$ values stored at the terminal unit's air inlet node. To account for upstream leakage the maximum air mass flow rate available is reset to:

$$\dot{m}'_{MaxAvail} = \dot{m}_{MaxAvail} - \dot{m}_{lk,us} \quad (712)$$

Downstream leakage must also be accounted for because not all of \dot{m}_{tu} will reach the zone. This is done by having \dot{m}_{tu} meet an adjusted zone load:

$$\dot{Q}_{z,adjusted} = \frac{1}{1 - Frac_{ds}} \dot{Q}_z \quad (713)$$

Here \dot{Q}_z [watts] is the actual zone load (met by $\dot{m}_{s,z}$) and $\dot{Q}_{z,adjusted}$ is the load used in the VAV terminal unit model to obtain \dot{m}_{tu} .

Once \dot{m}_{tu} is known, all the other flow rates can be calculated. $\dot{m}_{s,us}$ is assigned to the air distribution unit's air inlet node and $\dot{m}_{s,z}$ is assigned to the unit's air outlet node. Thus, air mass flow is not conserved through the unit: the two air leakage flow rates disappear. These two vanished flow rates are stored in the air distribution unit data structure. When the downstream return air plenum mass and energy balances are calculated, the leakage flow rate data is accessed and added back in as inlets to the return air plenum. Thus, the overall air system preserves a mass balance.

References

Wray, C.P. 2003. "Duct Thermal Performance Models for Large Commercial Buildings", Lawrence Berkeley National Laboratory Report to the California Energy Commission. LBNL-53410.

Wray, C.P. and N.E. Matson. 2003. "Duct Leakage Impacts on VAV System Performance in California Large Commercial Buildings", Lawrence Berkeley National Laboratory Report to the California Energy Commission. LBNL-53605.

Wray, C.P., R.C. Diamond, and M.H. Sherman. 2005. "Rationale for Measuring Duct Leakage Flows in Large Commercial Buildings". Proceedings – 26th AIVC Conference, Brussels, Belgium, September. LBNL-58252.

Fan Coil Unit

Overview

The input object ZoneHVAC:FourPipeFanCoil provides a model for a 4 pipe fan coil zonal hydronic unit that can supply heating and cooling to a zone. It contains a hot water coil, a chilled water coil, and a fan. It can supply a fixed amount of outdoor air, but can not operate in an economizer mode. The fan runs at constant speed – control is achieved by throttling the hot or cold water flow. The fan coil configuration and control is rather limited. The fan position is always *blow-through*, control is always by varying the water flow, never by holding the water flow constant and cycling the fan.

Model

The 4 pipe fan coil unit is modeled as a compound component consisting of 4 sub-components: an outdoor air mixer, a fan, a cooling coil, and a heating coil. In terms of EnergyPlus objects these are:

1. *OutdoorAir:Mixer*
2. *Fan:ConstantVolume*
3. *Coil:Cooling:Water*, *Coil:Cooling:Water:DetailedGeometry*, or *CoilSystem:Cooling:Water:HeatExchangerAssisted*
4. *Coil:Heating:Water*

The unit is a forward model: its inputs are defined by the state of its inlets: namely its 2 air streams – recirculated and outdoor air. The outputs of the model are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The unit data and simulation are encapsulated in the module *FanCoilUnits*.

Inputs and Data

The user describes the 4 pipe fan coil unit by inputting the names of the outdoor air mixer, the fan, the heating coil, and the cooling coil. The cooling coil type must also be specified.

The unit is connected to the overall HVAC system by specifying node names for the unit air inlet (for recirculated air) node, air outlet node, outdoor air node, relief node, inlet hot water node, and inlet chilled water node. The individual components comprising the fan coil must also be input and connected together properly. Specifically the outdoor air mixer mixed air node must be the same as the fan inlet node; the fan outlet node must be the same as the cooling coil air inlet node; the cooling coil air outlet node must be the same as the heating coil air inlet node; and the heating coil air outlet node must be the same as the unit air outlet node; the outdoor air mixer inlet nodes must match the unit inlet nodes; and the outdoor air mixer relief node must match the unit relief node.

The user needs to also specify (unless the unit is autosized) various maximum flow rates: the supply air flow rate, the outdoor air inlet flow rate, the maximum (and minimum) chilled water flow rate, and the maximum (and minimum) hot water flow rate. Heating and cooling convergence tolerances need to be specified or defaulted. And there is an on/off availability schedule for the unit.

All the input data for the fan coil unit is stored in the array *FanCoil*.

Calculation

Given the needed inputs, the output is calculated in subroutine *Calc4PipeFanCoil*. The temperature, humidity ratio and flow rate of the recirculated and outdoor air streams are taken from the inlet air nodes. The inlet hot and chilled water flow rates have been set by local controllers – temperatures are taken from the inlet water nodes. Then

1. The outdoor air mixer is simulated (Call *SimOAMixer*);
2. the fan is simulated (Call *SimulateFanComponents*);
3. the cooling coil is simulated (Call *SimulateWaterCoilComponents* or *SimHXAssistedCoolingCoil*);
4. the heating coil is simulated (Call *SimulateWaterCoilComponents*).

The load met (sensible cooling or heating) is calculated and passed back to the calling routine:

$$\dot{Q}_{sens,out} = \dot{m}_{tot} \cdot (\text{PsyHFnTdbW}(T_{out}, W_{in}) - \text{PsyHFnTdbW}(T_{in}, W_{in}))$$

where *PsyHFnTdbW* is the EnergyPlus function for calculating the specific enthalpy of air given the drybulb temperature and the humidity ratio. The subscript *in* indicates the conditions at the inlet recirculated air node.

Simulation and Control

From the result of the zone simulation we have the current heating/cooling demand on the unit $\dot{Q}_{z,req}$. The first step is to decide whether the unit is on for the current time step. If the load is less than 1 watt or the flow rate is less than .001 kg/s, the unit is off. If the availability schedule is off, the mass flow rate is set to zero, so the second condition holds. When the unit is off there will be no air flow through the unit and outlet conditions will be equal to inlet conditions.

$\dot{Q}_{z,req}$ is not the demand on the heating or cooling coil. To obtain the actual coil load, we need to calculate the unit output with no heating or cooling by the coils ($\dot{Q}_{unit,no\ hc}$). We obtain this by calling *Calc4PipeFanCoil* with the water flow rates set to zero. Then the coil loads are calculated:

$$\dot{Q}_{hc} = \dot{Q}_{z,hsp} - \dot{Q}_{unit,no\ hc}$$

$$\dot{Q}_{cc} = \dot{Q}_{z,csp} - \dot{Q}_{unit,no\ hc}$$

where \dot{Q}_{hc} is the heating coil load, $\dot{Q}_{z,hsp}$ is the current zone load to the heating setpoint, \dot{Q}_{cc} is the cooling coil load, and $\dot{Q}_{z,csp}$ is the current zone load to the cooling setpoint.

If the unit is on and $\dot{Q}_{cc} < 0$ and the thermostat type is not “single heating setpoint”, *ControlCompOutput* is called with the control node set to the cold water inlet node. *ControlCompOutput* is a general component control routine. In this case calls *Calc4PipeFanCoil* repeatedly while varying the cold water flow rate and minimizing $(\dot{Q}_{sens,out} - \dot{Q}_{z,csp}) / \dot{Q}_{z,csp}$ to within the cooling convergence tolerance. Similarly if the unit is on and $\dot{Q}_{hc} > 0$ and the thermostat type is not “single cooling setpoint”, *ControlCompOutput* is called with the control node set to the hot water inlet node. *ControlCompOutput* varies the hot water flow rate to minimize $(\dot{Q}_{sens,out} - \dot{Q}_{z,hsp}) / \dot{Q}_{z,hsp}$ to within the heating tolerance. *ControlCompOutput* executes a slow but safe interval halving algorithm to do its minimization. Once control is achieved, the total cooling/heating output is calculated:

$$\dot{Q}_{tot,out} = \dot{m} \cdot (\text{PsyHFnTdbW}(T_{out}, W_{out}) - \text{PsyHFnTdbW}(T_{in}, W_{in}))$$

References

No specific references.

Window Air Conditioner

Overview

The input object ZoneHVAC:WindowAirConditioner provides a model for a window air conditioner unit that is a packaged unit that supplies cooling to a zone (it is part of zone equipment, not part of the air loop equipment). It contains a fan, a DX cooling coil, and an outdoor air inlet. The coil meets the cooling load by cycling on/off. The fan can operate continuously or cycle on/off in conjunction with the coil.

Model

The window air conditioner is modeled as a compound component consisting of 3 sub-components: an outdoor air mixer, a fan, and a DX coil. In terms of EnergyPlus objects these are OutdoorAir:Mixer, Fan:ConstantVolume or Fan:OnOff, and Coil:Cooling:DX:SingleSpeed or CoilSystem:Cooling:DX:HeatExchangerAssisted. The unit is a forward model: its inputs are

defined by the state of its inlets: namely its 2 air streams – recirculated and outdoor air. The outputs of the model are the conditions of the outlet air stream: flow rate, temperature and humidity ratio. The model is also an averaged model: the performance of the unit is averaged over the time step. That is, the unit is assumed to cycle on/off during the time step and this on/off cycling is averaged over the simulation time step. The unit data and simulation are encapsulated in the module *WindowAC*.

Inputs and Data

The user describes the window air conditioner by inputting the names of the outdoor air mixer, the fan, and the cooling coil. The user can also choose fan placement – blow through or draw through; fan operation – cycling or continuous; and cooling coil type – normal DX or DX with heat exchanger assistance.

The connectivity of the unit needs to be specified: a recirculated (inlet) air node (same as a zone exhaust node); an air outlet node (same as a zone inlet node); an outdoor air inlet node; and a relief air node. The individual components comprising the window air conditioner must of course also be input and connected together properly. For instance, for a blow through fan configuration the outdoor air mixer mixed air node must be the same as the fan inlet node; the fan outlet node must be the same as the coil inlet node; the coil outlet node must be the same as the unit outlet node; the outdoor air mixer inlet nodes must match the unit inlet nodes; and the outdoor air mixer relief node must match the unit relief node.

The user also specifies the air conditioner flow rate delivered to the zone (when cycled on) and the outdoor air flow rate. The user also needs to specify an availability schedule for the unit (this is an on/off schedule).

Note that there is no input specifying the unit's design cooling capacity. This is an input in the DX coil object and is not repeated here.

All the input data for the window air conditioner is stored in the array *WindAC*.

Calculation

Given the needed inputs, the output is calculated in subroutine *CalcCyclingWindowAC*. The temperature, humidity ratio and flow rate of the recirculated and outdoor air streams are taken from the inlet air nodes. The part load ratio is specified by the calling routine. Then

- The outdoor air mixer is simulated (Call *SimOAMixer*);
- For blow-through fan position:
- the fan is simulated (Call *SimulateFanComponents*);
- the coil is simulated (Call *SimDXCoil* or *SimHXAssistedCoolingCoil*).

For draw-through fan position, the simulation order of the fan and coil is reversed. Note that data is never explicitly passed between the sub-components. This is all handled automatically by the node connections and the data stored on the nodes.

Simulation and Control

From the result of the zone simulation we have the heating/cooling demand on the unit $\dot{Q}_{z,req}$. The first step is to decide whether the unit is on for the current time step. For a unit with a cycling fan, the entire unit is assumed to be off if there is no cooling load, the cooling load is very small (less than 1 watt), the unit is scheduled off, or the zone temperature is in the deadband. For a unit with a continuous flow the fan operates if the unit is scheduled on, whether or not there is a cooling demand. The coil however only operates if there is a cooling demand and the zone temperature is not in the deadband.

If the unit is determined to be on, the next step is to find the unit part load fraction that will satisfy the cooling load. This is done in *ControlCycWindACOutput*. In this routine *CalcCyclingWindowAC* is first called with part load fraction equal to 0, then with part load fraction equal to 1. These calls establish the minimum and maximum cooling output possible

by the unit given the current conditions. An initial estimate of the part load fraction is then made:

$$PLF = (\dot{Q}_{z,req} - \dot{Q}_{out,min}) / |\dot{Q}_{out,max} - \dot{Q}_{out,min}|$$

Since the unit's cooling output is a nonlinear function of the part load fraction, this *PLF* will not give exactly the desired $\dot{Q}_{z,req}$. To obtain the exact *PLF* that will give $\dot{Q}_{z,req}$, it is necessary to iteratively call *CalcCyclingWindowAC*, varying *PLF* until the desired cooling output is obtained, within the error tolerance specified by the user in the input.

Once *PLF* is determined, *ControlCycWindACOutput* is exited. One last call to *CalcCyclingWindowAC* is made to establish the final outlet conditions at the unit's air outlet node. Finally, the inlet and outlet node conditions are used to calculate the reporting variables: sensible and total cooling output.

$$\dot{Q}_{sens,out} = \dot{m} \cdot (\text{PsyHFnTdbW}(T_{out}, W_{in}) - \text{PsyHFnTdbW}(T_{in}, W_{in}))$$

$$\dot{Q}_{sens,out} = \dot{m} \cdot (\text{PsyHFnTdbW}(T_{out}, W_{out}) - \text{PsyHFnTdbW}(T_{in}, W_{in}))$$

where *PsyHFnTdb* is the EnergyPlus function giving enthalpy as a function of temperature and humidity ratio.

References

No specific references.

Packaged Terminal Air Conditioner

Overview

The input object *ZoneHVAC:PackagedTerminalAirConditioner* provides a model for a packaged terminal air conditioner (PTAC) that is a compound object made up of other components. Each PTAC consists of an outdoor air mixer, direct expansion (DX) cooling coil, heating coil (gas, electric, hot water, or steam) and a supply air fan. While the figure below shows the PTAC with draw through fan placement, blow through fan placement can also be modeled by positioning the supply air fan between the outdoor air mixer and DX cooling coil. The packaged terminal air conditioner coordinates the operation of these components and is modeled as a type of zone equipment (Ref. *ZoneHVAC:EquipmentList* and *ZoneHVAC:EquipmentConnections*).

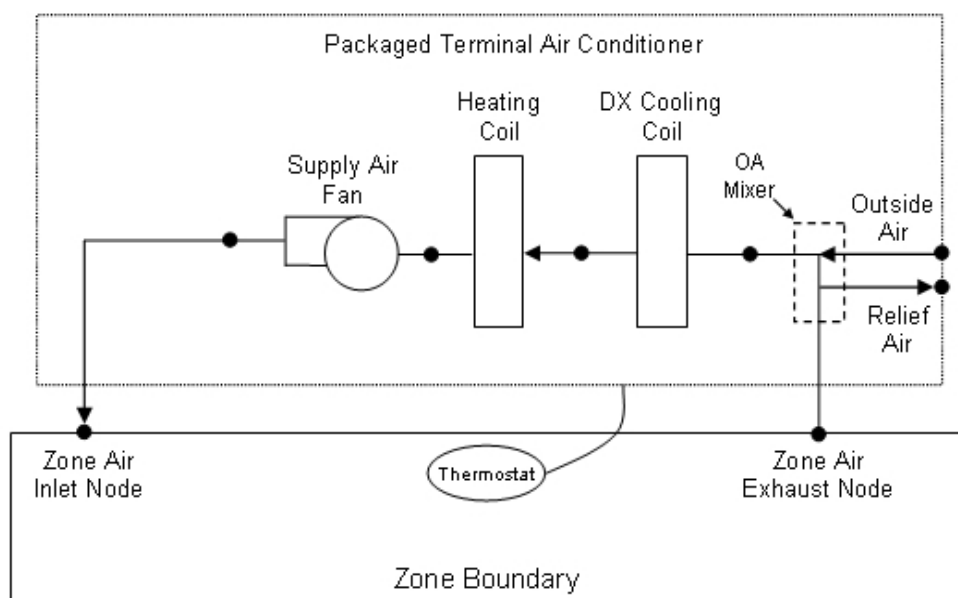


Figure 232. Schematic of a Packaged Terminal Air Conditioner with Draw Through Fan Placement

The PTAC conditions a single zone and is controlled by a thermostat located in that zone. The PTAC operates to meet the zone sensible cooling or sensible heating requirements as dictated by the thermostat schedule. The model calculates the required part-load ratio for the air conditioner's coils and the supply air fan to meet the cooling/heating requirements. The heating or cooling energy provided by the PTAC is delivered to the zone via the zone air inlet node.

The PTAC is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). Supply air fan operation is coordinated with the use of a supply air fan operating mode schedule. Schedule values of 0 denote cycling fan operation (AUTO fan). Schedule values other than 0 denote continuous fan operation (fan ON). Fan:OnOff must be used to model AUTO fan (i.e. if schedule values of 0 occur in the supply air fan operating mode schedule), while Fan:OnOff or Fan:ConstantVolume can be used to model fan ON (i.e. if schedule values of 0 do not occur in the supply air fan operating mode schedule). The supply air fan operating mode schedule specified for the PTAC overrides the operating mode specified in the DX cooling coil object.

Output variables reported by the PTAC object include the supply air fan part-load ratio, the air conditioner's part-load ratio (cooling or heating), and the electric consumption of the PTAC. Additional output variables report the total zone heating rate and the total zone cooling rate provided by the air conditioner. The sensible and latent components of zone cooling are also available as output variables. Reporting of other variables of interest for the PTAC (DX coil cooling rate, coil heating rate, crankcase heater power, fan power, etc.) is done by the individual system components (fan, DX cooling coil and heating coil).

Model Description

As described previously, the PTAC conditions a single zone and is controlled by a zone thermostat (ZoneControl:Thermostatic). Each simulation time step, EnergyPlus performs a zone air heat balance to determine if cooling or heating is required to meet the thermostat setpoints, excluding any impacts from PTAC operation. PTAC performance is then modeled with all heating/cooling coils off but the supply air fan operates as specified by the user. If the zone air heat balance plus the impact of PTAC operation with coils off results in no requirement for heating or cooling by the PTAC coils, or if the PTAC is scheduled off (via its availability schedule), then the PTAC coils do not operate and the air conditioner's part-load ratio output variable is set to 0. If the model determines that cooling or heating is required and

the PTAC is scheduled to operate, the model calculates the average air flow rate through the unit and the part-load ratio of the cooling and heating coils in order to meet the thermostat setpoint temperature.

The remainder of this section describes the calculations performed during the latter situation, when cooling or heating coil operation is required. For any HVAC simulation time step, the PTAC can only be cooling or heating, not both. Because the PTAC cycles its coil(s) on and off to meet the required load, the coil(s) operate for a portion of the time step and are off for the rest of the time step. If the user specifies continuous fan operation (i.e. supply air fan operating mode schedule value is greater than 0), then the supply air fan continues to operate at a user-specified flow rate even during periods when the coils cycle off. If the user specifies AUTO fan operation (i.e. supply air fan operating mode schedule value is equal to 0), then the supply air fan cycles on and off with the coils. The model accounts for these variations in air flow through the PTAC within a simulation time step when it determines the total cooling or heating energy delivered to the zone, the average supply air conditions and air flow rate, and the energy consumed by the air conditioner.

Cooling Operation

If EnergyPlus determines that the air conditioner must supply cooling to the zone in order to meet the zone air temperature setpoint, then the model first calculates the PTAC's sensible cooling rate to the zone under two conditions: when the unit runs at full-load (steady-state) conditions and when the DX cooling coil is OFF. If the supply air fan cycles on/off with the compressor, then the sensible cooling rate is zero when the cooling coil is OFF. However if the fan is configured to run continuously regardless of coil operation, then the sensible cooling rate will not be zero when the cooling coil is OFF. Calculating the sensible cooling rate involves modeling the supply air fan (and associated fan heat), the outdoor air mixer, and the DX cooling coil. The heating coil is also modeled, but only to pass the air properties and mass flow rate from its inlet node to its outlet node. For each of these cases (full load and DX cooling coil OFF), the sensible cooling rate delivered to the zone by the PTAC is calculated as follows:

$$\dot{Q}_{cooling,max} = \left(\dot{m}_{SA,full\ load} \right) \left(h_{out,full\ load} - h_{zone\ air} \right)_{HRmin}$$

$$\dot{Q}_{cooling,min} = \left(\dot{m}_{SA,coil\ off} \right) \left(h_{out,coil\ off} - h_{zone\ air} \right)_{HRmin}$$

where:

$\dot{Q}_{cooling,max}$ = maximum PTAC sensible cooling rate with cooling coil ON, W

$\dot{m}_{SA,full\ load}$ = supply air mass flow rate at full-load (steady-state) conditions, kg/s

$h_{out,full\ load}$ = enthalpy of air exiting the PTAC at full-load conditions, J/kg

$h_{zone\ air}$ = enthalpy of zone (exhaust) air, J/kg

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the PTAC exiting air or the zone (exhaust) air

$\dot{Q}_{cooling,min}$ = minimum PTAC sensible cooling rate with cooling coil OFF, W

$\dot{m}_{SA,coil\ off}$ = supply air mass flow rate with the cooling coil OFF, kg/s

$h_{out,coil\ off}$ = enthalpy of air exiting the PTAC with the cooling coil OFF, J/kg

With the calculated PTAC sensible cooling rates and the zone sensible cooling load to be met, the compressor part-load ratio for the PTAC is approximately equal to:

$$PartLoadRatio = MAX \left(0.0, \frac{ABS \left(\dot{Q}_{zone,cooling} - \dot{Q}_{cooling,min} \right)}{ABS \left(\dot{Q}_{cooling,max} - \dot{Q}_{cooling,min} \right)} \right)$$

where:

PartLoadRatio = compressor part-load ratio required to meet the zone load

$\dot{Q}_{zone,cooling}$ = required zone sensible cooling rate to meet setpoint, W

Since the part-load performance of the DX cooling coil is frequently non-linear (Ref: DX Cooling Coil Model), and the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan), the actual part-load ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the individual PTAC component models) until the PTAC's cooling output (including on/off cycling effects) matches the zone cooling load requirement.

If the PTAC has been specified with cycling fan/cycling coil (AUTO fan), then the user-defined supply air flow rate during cooling operation (volumetric flow rate converted to mass flow rate) is multiplied by the final PartLoadRatio value to determine the average supply air mass flow rate for the HVAC system simulation time step. For this case, the air conditions (temperature, humidity ratio and enthalpy) at nodes downstream of the cooling coil represent the full-load (steady-state) values when the coil is operating. If the supply air fan is specified to run continuously (fan ON), then the supply air mass flow rate is calculated as the average of the air mass flow rate when the compressor is on and the air mass flow rate when the compressor is off. In this case, the air conditions at nodes downstream of the cooling coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and mixed inlet air conditions when the coil is OFF). Additional discussion regarding the calculation of the average supply air flow and supply air conditions is provided later in this section.

Heating Operation

Calculations of the PTAC's sensible heating rate delivered to the zone at full load and with the heating coil OFF are identical to the calculations described above for cooling operation.

$$\dot{Q}_{heating,max} = \left(\dot{m}_{SA,full\ load} \right) \left(h_{out,full\ load} - h_{zone\ air} \right)_{HRmin}$$

$$\dot{Q}_{heating,min} = \left(\dot{m}_{SA,coil\ off} \right) \left(h_{out,coil\ off} - h_{zone\ air} \right)_{HRmin}$$

where:

$\dot{Q}_{heating,max}$ = maximum PTAC sensible heating rate with heating coil ON, W

$\dot{Q}_{heating,min}$ = minimum PTAC sensible heating rate with heating coil OFF, W

With the calculated PTAC sensible heating rates and the zone sensible heating load to be met, the heating coil part-load ratio for the PTAC is approximately equal to:

$$PartLoadRatio = MAX \left(0.0, \frac{ABS \left(\dot{Q}_{zone,heating} - \dot{Q}_{heating,min} \right)}{ABS \left(\dot{Q}_{heating,max} - \dot{Q}_{heating,min} \right)} \right)$$

where:

PartLoadRatio = heating coil part-load ratio required to meet the zone load

$\dot{Q}_{zone,heating}$ = required zone sensible heating rate to meet setpoint, W

Iterative calculations (successive modeling of the individual PTAC component models) are used to determine the final heating part-load ratio to account for the non-linear performance of the heating coil at part-load conditions and the variation in supply air fan heat for the case of cycling fan/cycling coil (AUTO fan). If heating coil operation at full load is unable to meet the entire zone heating load (e.g., the heating coil capacity is insufficient or the coil is scheduled OFF), the air conditioner's part-load ratio is set to 1 to meet the zone heating load to the extent possible.

Average Air Flow Calculations

The packaged terminal air conditioner operates based on user-specified (or autosized) air flow rates. The PTAC's supply air flow rate during cooling operation may be different than the supply air flow rate during heating operation. In addition, the supply air flow rate when no cooling or heating is required but the supply air fan remains ON can be different than the air flow rates when cooling or heating is required. The outdoor air flow rates can likewise be different in these various operating modes. The model takes these different flow rates into account when modeling the air conditioner, and the average air flow rate for each simulation time step is reported on the inlet/outlet air nodes of the various PTAC components in proportion to the calculated part-load ratio of the coil.

The average supply air and outdoor air mass flow rates through the air conditioner for the HVAC simulation time step are calculated based on the part-load ratio of the DX cooling coil or heating coil (whichever coil is operating) as follows:

$$\dot{m}_{SA,avg} = \dot{m}_{SA,coil\ on}(PartLoadRatio) + \dot{m}_{SA,coil\ off}(1 - PartLoadRatio)$$

$$\dot{m}_{OA,avg} = \dot{m}_{OA,coil\ on}(PartLoadRatio) + \dot{m}_{OA,coil\ off}(1 - PartLoadRatio)$$

where:

$\dot{m}_{SA,avg}$ = average supply air mass flow rate during the time step, kg/s

$\dot{m}_{SA,coil\ on}$ = supply air mass flow rate when the coil is ON, kg/s

PartLoadRatio = part-load ratio of the coil (heating or cooling)

$\dot{m}_{SA,coil\ off}$ = supply air mass flow rate when the coil is OFF, kg/s

$\dot{m}_{OA,avg}$ = average outdoor air mass flow rate during the time step, kg/s

$\dot{m}_{OA,coil\ on}$ = average outdoor air mass flow rate when the coil is ON, kg/s

$\dot{m}_{OA, coil\ off}$ = average outdoor air mass flow rate when the coil is OFF, kg/s

The supply air and outdoor air flow rates when the DX cooling coil or the heating coil is ON are specified by the user (i.e., supply air volumetric flow rate during cooling operation, supply air volumetric flow rate during heating operation, outdoor air volumetric air flow rate during cooling operation, and outdoor air volumetric air flow rate during heating operation) and are converted from volumetric to mass flow rate. If the user has specified cycling fan operation (i.e. supply air fan operating mode schedule value is equal to 0), then the supply air and outdoor air mass flow rates when the coil is OFF are zero. If the user has specified constant fan operation (i.e. supply air fan operating mode schedule value is greater than 0), then the user-defined air flow rates when no cooling or heating is needed are used when the coil is OFF.

There is one special case. If the supply air fan operating mode schedule value specifies constant fan operation and the user also specifies that the supply air volumetric flow rate when no cooling or heating is needed is zero (or field is left blank), then the model assumes that the supply air and outdoor air mass flow rates when the coil is OFF are equal to the corresponding air mass flow rates when the cooling or heating coil was last operating (ON).

Calculation of Outlet Air Conditions

When the supply air fan cycles on and off with the PTAC coils (AUTO fan), the calculated outlet air conditions (temperature, humidity ratio, and enthalpy) from the heating coil or the DX cooling coil at full-load (steady-state) operation are reported on the appropriate coil outlet air node. The air mass flow rate reported on the air nodes is the average air mass flow rate proportional to the part-load ratio of the coil (see Average Air Flow Calculations above).

When the supply air fan operates continuously while the PTAC coils cycle on and off (fan ON), the air mass flow rate reported on the air nodes is the average air mass flow rate proportional to the part-load ratio of the coil (see Average Air Flow Calculations above). Since the air flow rate can be different when the coil is ON compared to when the coil is OFF, then the average outlet air conditions from the heating coil or the DX cooling coil are reported on the appropriate coil outlet air node.

For hot water or steam coils, the water or steam mass flow rate is also proportional to the part-load ratio of the coil regardless of the supply air fan operating mode. Refer to the sections in the document that describe the heating and DX cooling coils for further explanation on how they report their outlet air (and water or steam) conditions.

Calculation of Zone Heating and Cooling Rates

At the end of each HVAC simulation time step, this compound object reports the heating or cooling rate and energy delivered to the zone, as well as the electric power and consumption by the air conditioner. In terms of thermal energy delivered to the zone, the sensible, latent and total energy transfer rate to the zone is calculated as follows:

$$\dot{Q}_{Total} = \left(\dot{m}_{SA, avg} \right) (h_{out, avg} - h_{zone\ air})$$

$$\dot{Q}_{Sensible} = \left(\dot{m}_{SA, avg} \right) (h_{out, avg} - h_{zone\ air})_{HRmin}$$

$$\dot{Q}_{Latent} = \dot{Q}_{Total} - \dot{Q}_{Sensible}$$

where:

- \dot{Q}_{Total} = total energy transfer rate to the zone, W
 $\dot{Q}_{Sensible}$ = sensible energy transfer rate to the zone, W
 \dot{Q}_{Latent} = latent energy transfer rate to the zone, W
 $\dot{m}_{SA, avg}$ = average mass flow rate of the supply air stream, kg/s
 $h_{out, avg}$ = enthalpy of the air being supplied to the zone, J/kg

Since each of these energy transfer rates can be calculated as positive or negative values, individual reporting variables are established for cooling and heating and only positive values are reported. The following calculations are representative of what is done for each of the energy transfer rates:

IF ($\dot{Q}_{Total} < 0.0$) THEN

$$\dot{Q}_{TotalCooling} = \text{ABS}(\dot{Q}_{Total})$$

$$\dot{Q}_{TotalHeating} = 0.0$$

ELSE

$$\dot{Q}_{TotalCooling} = 0.0$$

$$\dot{Q}_{TotalHeating} = \dot{Q}_{Total}$$

where:

$\dot{Q}_{TotalCooling}$ = output variable 'Packaged Terminal Air Conditioner Total Zone Cooling Rate, W'

$\dot{Q}_{TotalHeating}$ = output variable 'Packaged Terminal Air Conditioner Total Zone Heating Rate, W'

In addition to heating and cooling rates, the heating and cooling energy supplied to the zone is also calculated for the time step being reported. The following example for total zone cooling energy is representative of what is done for the sensible and latent energy as well as the heating counterparts.

$$Q_{TotalCooling} = \dot{Q}_{TotalCooling} * TimeStepSys * 3600.$$

where:

$Q_{TotalCooling}$ = output variable 'Packaged Terminal Air Conditioner Total Zone Cooling Energy, J'

$TimeStepSys$ = HVAC system simulation time step, hr

Packaged Terminal Heat Pump

Overview

The input object ZoneHVAC:PackagedTerminalHeatPump provides a model for a packaged terminal heat pump (PTHP) that is a compound object made up of other components. Each PTHP consists of an outdoor air mixer, direct expansion (DX) cooling coil, DX heating coil, supply air fan, and a supplemental heating coil. While the figure below shows the PTHP with draw through fan placement, blow through fan placement can also be modeled by moving the supply air fan before the DX cooling coil. The packaged terminal heat pump coordinates the operation of these components and is modeled as a type of zone equipment (Ref. ZoneHVAC:EquipmentList and ZoneHVAC:EquipmentConnections).

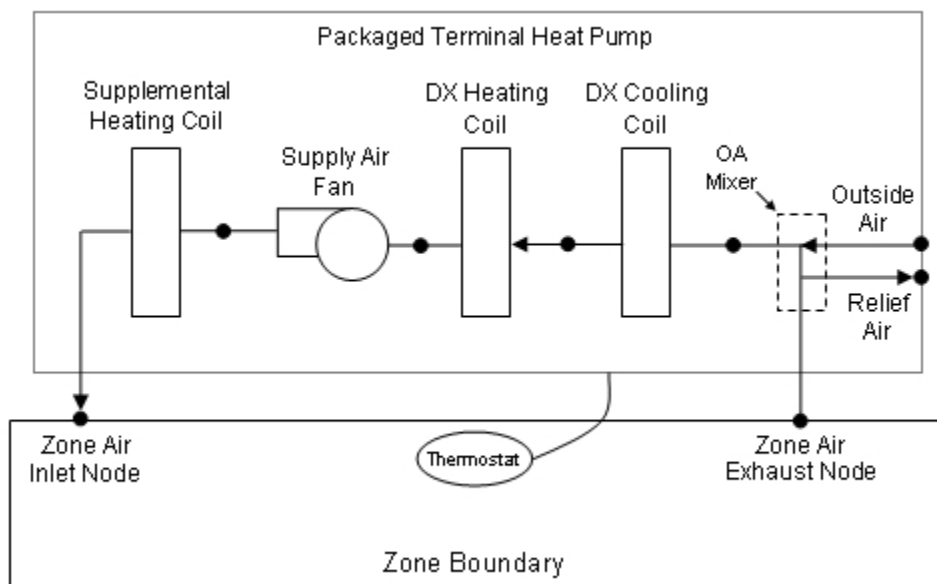


Figure 233. Schematic of a Packaged Terminal Heat Pump (Draw Through Fan Placement)

The PTHP conditions a single zone and is controlled by a thermostat located in that zone. The PTHP operates to meet the zone sensible cooling or sensible heating requirements as dictated by the thermostat schedule. The model calculates the required part-load ratio for the heat pump's coils and the supply air fan to meet the cooling/heating requirements. The heating or cooling energy provided by the PTHP is delivered to the zone via the zone air inlet node.

The PTHP is able to model supply air fan operation in two modes: cycling fan – cycling coil (i.e., AUTO fan) and continuous fan – cycling coil (i.e., fan ON). Fan:OnOff must be used to model AUTO fan, while Fan:OnOff or Fan:ConstantVolume can be used to model fan ON.

Output variables reported by the PTHP object include the supply air fan part-load ratio, the compressor part-load ratio, and the electric consumption of the PTHP. Additional output variables report the total zone heating rate and the total zone cooling rate provided by the heat pump. The sensible and latent components of zone cooling are also available as output variables. Reporting of other variables of interest for the PTHP (DX coil cooling rate, DX coil heating rate, crankcase heater power, fan power, etc.) is done by the individual system components (fan, DX cooling coil, DX heating coil, and supplemental heating coil).

Model Description

As described previously, the PTHP conditions a single zone and is controlled by a zone thermostat (ZoneControl:Thermostat). Each simulation time step, EnergyPlus performs a zone air heat balance to determine if cooling or heating is required to meet the thermostat setpoints, excluding any impacts from PTHP operation. PTHP performance is then modeled with all heating/cooling coils off but the supply air fan operates as specified by the user. If the zone air heat balance plus the impact of PTHP operation with coils off results in no requirement for heating or cooling by the PTHP coils, or if the PTHP is scheduled off (via its availability schedule), then the PTHP coils do not operate and the compressor part-load ratio output variable is set to 0. If the model determines that cooling or heating is required and the PTHP is scheduled to operate, the model calculates the average air flow rate through the unit and the part-load ratio of the cooling and heating coils in order to meet the thermostat setpoint temperature.

The remainder of this section describes the calculations performed during the latter situation, when cooling or heating coil operation is required. For any HVAC simulation time step, the PTHP can only be cooling or heating, not both. Because the PTHP cycles its coil(s) on and off to meet the required load, the coil(s) operate for a portion of the time step and are off for the rest of the time step. If the user specifies continuous fan operation (supply air fan operating mode schedule value > 0), then the supply air fan continues to operate at a user-specified flow rate even during periods when the coils cycle off. If the user specifies AUTO fan operation (supply air fan operating mode schedule value = 0), then the supply air fan cycles on and off with the coils. The model accounts for these variations in air flow through the PTHP within a simulation time step when it determines the total cooling or heating energy delivered to the zone, the average supply air conditions and air flow rate, and the energy consumed by the heat pump.

Cooling Operation

If EnergyPlus determines that the heat pump must supply cooling to the zone in order to meet the zone air temperature setpoint, then the model first calculates the PTHP's sensible cooling rate to the zone under two conditions: when the unit runs at full-load (steady-state) conditions and when the DX cooling coil is OFF. If the supply air fan cycles on/off with the compressor, then the sensible cooling rate is zero when the cooling coil is OFF. However if the fan is configured to run continuously regardless of coil operation, then the sensible cooling rate will not be zero when the cooling coil is OFF. Calculating the sensible cooling rate involves modeling the supply air fan (and associated fan heat), the outdoor air mixer, and the DX cooling coil. The DX heating coil and the gas or electric supplemental heating coil are also modeled, but only to pass the air properties and mass flow rate from their inlet nodes to their outlet nodes. For each of these cases (full load and DX cooling coil OFF), the sensible cooling rate delivered to the zone by the PTHP is calculated as follows:

$$\dot{Q}_{cooling,max} = \left(\dot{m}_{SA,full\ load} \right) \left(h_{out,full\ load} - h_{zone\ air} \right)_{HRmin}$$

$$\dot{Q}_{cooling,min} = \left(\dot{m}_{SA,coil\ off} \right) \left(h_{out,coil\ off} - h_{zone\ air} \right)_{HRmin}$$

where:

$\dot{Q}_{cooling,max}$ = maximum PTHP sensible cooling rate with cooling coil ON, W

$\dot{m}_{SA,full\ load}$ = supply air mass flow rate at full-load (steady-state) conditions, kg/s

$h_{out,full\ load}$ = enthalpy of air exiting the PTHP at full-load conditions, J/kg

$h_{zone\ air}$ = enthalpy of zone (exhaust) air, J/kg
 HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the PTHP exiting air or the zone (exhaust) air

$\dot{Q}_{cooling,min}$ = minimum PTHP sensible cooling rate with cooling coil OFF, W

$\dot{m}_{SA,coil\ off}$ = supply air mass flow rate with the cooling coil OFF, kg/s

$h_{out, coil\ off}$ = enthalpy of air exiting the PTHP with the cooling coil OFF, J/kg

With the calculated PTHP sensible cooling rates and the zone sensible cooling load to be met, the compressor part-load ratio for the PTHP is approximately equal to:

$$PartLoadRatio = MAX \left(0.0, \frac{ABS \left(\dot{Q}_{zone,cooling} - \dot{Q}_{cooling,min} \right)}{ABS \left(\dot{Q}_{cooling,max} - \dot{Q}_{cooling,min} \right)} \right)$$

where:

$PartLoadRatio$ = compressor part-load ratio required to meet the zone load

$\dot{Q}_{zone,cooling}$ = required zone sensible cooling rate to meet setpoint, W

Since the part-load performance of the DX cooling coil is frequently non-linear (Ref: DX Cooling Coil Model), and the supply air fan heat varies based on cooling coil operation for the case of cycling fan/cycling coil (AUTO fan), the actual part-load ratio for the cooling coil compressor and fan are determined through iterative calculations (successive modeling of the individual PTHP component models) until the PTHP's cooling output (including on/off cycling effects) matches the zone cooling load requirement within the cooling convergence tolerance that is specified.

If the PTHP is specified to operate with cycling fan/cycling coil (AUTO fan), then the user-defined supply air flow rate during cooling operation (volumetric flow rate converted to mass flow rate) is multiplied by the final PartLoadRatio value to determine the average supply air mass flow rate for the HVAC system simulation time step. For this case, the air conditions (temperature, humidity ratio and enthalpy) at nodes downstream of the cooling coil represent the full-load (steady-state) values when the coil is operating. If the supply air fan is specified to operate continuously (fan ON), then the supply air mass flow rate is calculated as the average of the air mass flow rate when the compressor is on and the air mass flow rate when the compressor is off. In this case, the air conditions at nodes downstream of the cooling coil are calculated as the average conditions over the simulation time step (i.e., the weighted average of full-load conditions when the coil is operating and mixed inlet air conditions when the coil is OFF). Additional discussion regarding the calculation of the average supply air flow and supply air conditions is provided later in this section.

Heating Operation

Calculations of the PTHP's sensible heating rate delivered to the zone at full load and with the DX heating coil OFF are identical to the calculations described above for cooling operation.

$$\dot{Q}_{heating,max} = \left(\dot{m}_{SA,full\ load} \right) \left(h_{out,full\ load} - h_{zone\ air} \right)_{HRmin}$$

$$\dot{Q}_{heating,min} = \left(\dot{m}_{SA,coil\ off} \right) \left(h_{out,coil\ off} - h_{zone\ air} \right)_{HRmin}$$

where:

$\dot{Q}_{heating,max}$ = maximum PTHP sensible heating rate with DX heating coil ON, W

$\dot{Q}_{heating,min}$ = minimum PTHP sensible heating rate with DX heating coil OFF, W

With the calculated PTHP sensible heating rates and the zone sensible heating load to be met, the compressor part-load ratio for the PTHP is approximately equal to:

$$PartLoadRatio = MAX \left(0.0, \frac{ABS \left(\dot{Q}_{zone,heating} - \dot{Q}_{heating,min} \right)}{ABS \left(\dot{Q}_{heating,max} - \dot{Q}_{heating,min} \right)} \right)$$

where:

PartLoadRatio = compressor part-load ratio required to meet the zone load

$\dot{Q}_{zone,heating}$ = required zone sensible heating rate to meet setpoint, W

Iterative calculations (successive modeling of the individual PTHP component models) are used to determine the final heating part-load ratio to account for the non-linear performance of the DX heating coil at part-load conditions and the variation in supply air fan heat for the case of cycling fan/cycling coil (AUTO fan). If DX heating coil operating at full load is unable to meet the entire zone heating load (e.g., the DX heating coil capacity is insufficient or the coil is scheduled OFF, or the outdoor temperature is below the PTHP's minimum outdoor dry-bulb temperature for compressor operation), the supplemental heating coil is activated to meet the remaining zone heating load to the extent possible.

Average Air Flow Calculations

The packaged terminal heat pump operates based on user-specified (or autosized) air flow rates. The PTHP's supply air flow rate during cooling operation may be different than the supply air flow rate during heating operation. In addition, the supply air flow rate when no cooling or heating is required but the supply air fan remains ON can be different than the air flow rates when cooling or heating is required. The outdoor air flow rates can likewise be different in these various operating modes. The model takes these different flow rates into account when modeling the heat pump, and the average air flow rate for each simulation time step is reported on the inlet/outlet air nodes of the various PTHP components in proportion to the calculated part-load ratio of the DX coil compressor.

The average supply air and outdoor air mass flow rates through the heat pump for the HVAC simulation time step are calculated based on the part-load ratio of the DX cooling coil or DX heating coil (whichever coil is operating) as follows:

$$\dot{m}_{SA,avg} = \dot{m}_{SA,comp\ on} (PartLoadRatio) + \dot{m}_{SA,comp\ off} (1 - PartLoadRatio)$$

$$\dot{m}_{OA,avg} = \dot{m}_{OA,comp\ on} (PartLoadRatio) + \dot{m}_{OA,comp\ off} (1 - PartLoadRatio)$$

where:

• $\dot{m}_{SA, avg}$ = average supply air mass flow rate during the time step, kg/s

• $\dot{m}_{SA, comp on}$ = supply air mass flow rate when the DX coil compressor is ON, kg/s

PartLoadRatio = part-load ratio of the DX coil compressor (heating or cooling)

• $\dot{m}_{SA, comp off}$ = supply air mass flow rate when the DX coil compressor is OFF, kg/s

• $\dot{m}_{OA, avg}$ = average outdoor air mass flow rate during the time step, kg/s

• $\dot{m}_{OA, comp on}$ = average outdoor air mass flow rate when the DX coil compressor is ON, kg/s

• $\dot{m}_{OA, comp off}$ = average outdoor air mass flow rate when the DX coil compressor is OFF, kg/s

The supply air and outdoor air flow rates when the DX cooling or DX heating coil compressor is ON are specified by the user (e.g., supply air volumetric flow rate during cooling operation, supply air volumetric flow rate during heating operation, outdoor air volumetric air flow rate during cooling operation, and outdoor air volumetric air flow rate during heating operation) and are converted from volumetric to mass flow rate. If the user has specified cycling fan operation (supply air fan operating mode schedule value = 0), then the supply air and outdoor air mass flow rates when the DX compressor is OFF are zero. If the user has specified constant fan operation (supply air fan operating mode schedule value > 0), then the user-defined air flow rates when no cooling or heating is needed are used when the DX compressor is OFF.

There is one special case. If the user has specified constant fan operation (supply air fan operating mode schedule value > 0) and they specify that the supply air volumetric flow rate when no cooling or heating is needed is zero (or field is left blank), then the model assumes that the supply air and outdoor air mass flow rates when the DX coil compressor is OFF are equal to the corresponding air mass flow rates when the compressor was last operating (ON).

Calculation of Outlet Air Conditions

When the supply air fan cycles on and off with the PTHP coils (AUTO fan), the calculated outlet air conditions (temperature, humidity ratio, and enthalpy) from the DX heating or DX cooling coil at full-load (steady-state) operation are reported on the appropriate coil outlet air node. The air mass flow rate reported on the air nodes is the average air mass flow rate proportional to the part-load ratio of the DX coil compressor (see Average Air Flow Calculations above).

When the supply air fan operates continuously while the PTHP coils cycle on and off (fan ON), the air mass flow rate reported on the air nodes is the average air mass flow rate proportional to the part-load ratio of the DX coil compressor (see Average Air Flow Calculations above). Since the air flow rate can be different when the coil is ON compared to when the coil is OFF, then the average outlet air conditions from the DX heating or DX cooling coil are reported on the appropriate coil outlet air node.

Refer to the sections in the document that describe the DX heating and DX cooling coils for further explanation on how they report their outlet air conditions.

Calculation of Zone Heating and Cooling Rates

At the end of each HVAC simulation time step, this compound object reports the heating or cooling rate and energy delivered to the zone, as well as the electric power and consumption by the heat pump. In terms of thermal energy delivered to the zone, the sensible, latent and total energy transfer rate to the zone is calculated as follows:

$$\dot{Q}_{Total} = \left(\dot{m}_{SA, avg} \right) (h_{out, avg} - h_{zone air})$$

$$\dot{Q}_{Sensible} = \left(\dot{m}_{SA, avg} \right) (h_{out, avg} - h_{zone air})_{HRmin}$$

$$\dot{Q}_{Latent} = \dot{Q}_{Total} - \dot{Q}_{Sensible}$$

where:

\dot{Q}_{Total} = total energy transfer rate to the zone, W

$\dot{Q}_{Sensible}$ = sensible energy transfer rate to the zone, W

\dot{Q}_{Latent} = latent energy transfer rate to the zone, W

$\dot{m}_{SA, avg}$ = average mass flow rate of the supply air stream, kg/s

$h_{out, avg}$ = enthalpy of the air being supplied to the zone, J/kg

Since each of these energy transfer rates can be calculated as positive or negative values, individual reporting variables are established for cooling and heating and only positive values are reported. The following calculations are representative of what is done for each of the energy transfer rates:

IF ($\dot{Q}_{Total} < 0.0$) THEN

$$\dot{Q}_{TotalCooling} = \text{ABS}(\dot{Q}_{Total})$$

$$\dot{Q}_{TotalHeating} = 0.0$$

ELSE

$$\dot{Q}_{TotalCooling} = 0.0$$

$$\dot{Q}_{TotalHeating} = \dot{Q}_{Total}$$

where:

$\dot{Q}_{TotalCooling}$ = output variable 'Packaged Terminal Heat Pump Total Zone Cooling Rate, W'

$\dot{Q}_{TotalHeating}$ = output variable 'Packaged Terminal Heat Pump Total Zone Heating Rate, W'

In addition to heating and cooling rates, the heating and cooling energy supplied to the zone is also calculated for the time step being reported. The following example for total zone

cooling energy is representative of what is done for the sensible and latent energy as well as the heating counterparts.

$$Q_{TotalCooling} = \dot{Q}_{TotalCooling} * TimeStepSys * 3600.$$

where:

$Q_{TotalCooling}$ = output variable 'Packaged Terminal Heat Pump Total Zone Cooling Energy, J'

$TimeStepSys$ = HVAC system simulation time step, hr

Energy Recovery Ventilator

The input object ZoneHVAC:EnergyRecoveryVentilator provides a model for a stand alone energy recovery ventilator (ERV) that is a single-zone HVAC component used for exhaust air heat recovery (see figure below). This compound object consists of three required components: a generic air-to-air heat exchanger (see object HeatExchanger:AirToAir:SensibleAndLatent), a supply air fan, and an exhaust air fan (see object Fan:OnOff). An optional controller (see object ZoneHVAC:EnergyRecoveryVentilator:Controller) may be used to simulate economizer (free cooling) operation.

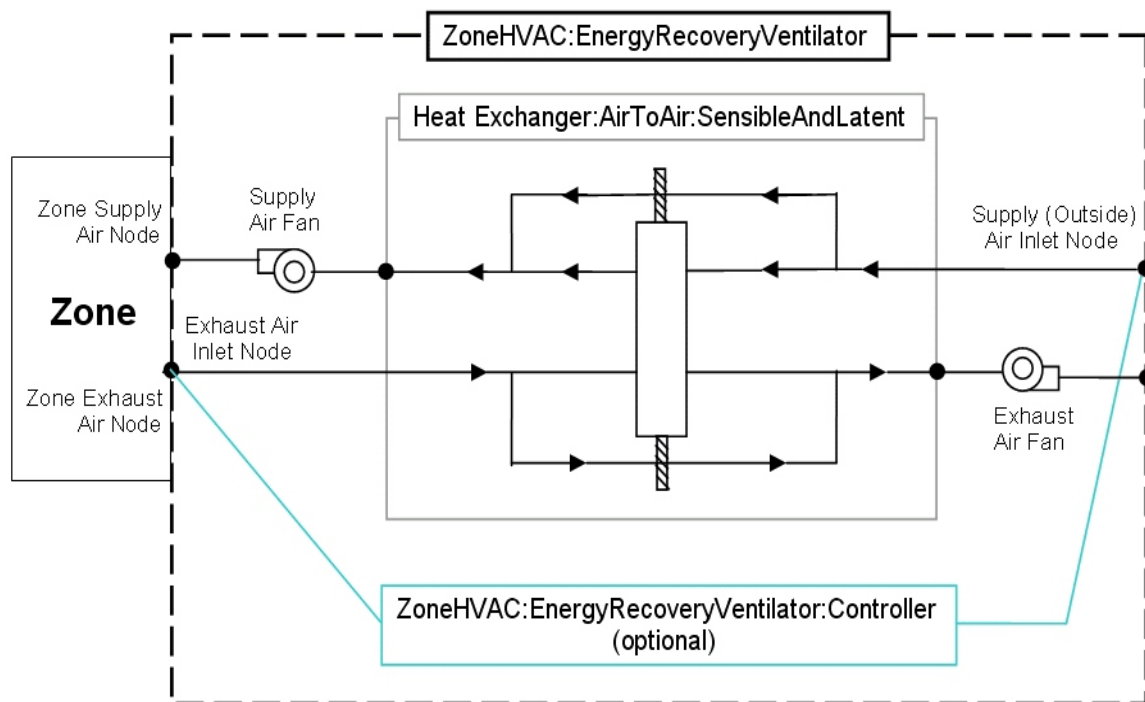


Figure 234. Schematic of the Energy Recovery Ventilator:Stand Alone compound object

This compound object models the basic operation of supply and exhaust air fans and an air-to-air heat exchanger. The stand alone ERV operates whenever the unit is scheduled to be available (Availability schedule). The stand alone ERV object can be used in conjunction with an economizer feature whereby heat exchange is suspended whenever free cooling is available (i.e., air flow is fully bypassed around a fixed-plate heat exchanger or the rotation of a rotary heat exchanger is stopped).

To model a stand alone ERV connected to a single zone, the input data file should include the following objects:

- ZoneHVAC:EnergyRecoveryVentilator

- HeatExchanger:AirToAir:SensibleAndLatent
- Fan:OnOff (supply air)
- Fan:OnOff (exhaust air)
- ZoneHVAC:EnergyRecoveryVentilator:Controller (if economizer [free cooling] operation is desired)
- SetpointManager:Scheduled (if supply air outlet temperature control is used, Ref. HeatExchanger:AirToAir:SensibleAndLatent)
- ZoneHVAC:EquipmentConnections
- ZoneHVAC:EquipmentList
- OutdoorAir:NodeList

Model Description

The purpose of this compound component is to simply call the individual component models and optional controller for each energy recovery ventilator. Since this equipment is not associated with an air loop, the compound object sets the supply and exhaust air mass flow rates through the ventilator. This compound object is also used to report the total, sensible and latent energy supplied to the zone, as well as the total electrical energy consumed by all of the individual components (supply air fan, exhaust air fan and heat exchanger parasitics).

During each simulation time step, the air mass flow rate at the supply air and exhaust air inlets is set based on the stand alone ERV's availability schedule and the specified volumetric air flow rates as follows:

IF (availability schedule value > 0) THEN

$$\dot{m}_{Supply} = \rho \dot{V}_{Supply}$$

$$\dot{m}_{Exhaust} = \rho \dot{V}_{Exhaust}$$

ELSE

$$\dot{m}_{Supply} = \dot{m}_{Exhaust} = 0.0$$

where:

\dot{m}_{Supply} = mass flow rate of the supply air stream, kg/s

$\dot{m}_{Exhaust}$ = mass flow rate of the exhaust air stream, kg/s

ρ = density of dry air at local barometric pressure (adjusted for altitude) and 20 °C, kg/m³

\dot{V}_{Supply} = volumetric flow rate of the supply air stream, m³/s

$\dot{V}_{Exhaust}$ = volumetric flow rate of the exhaust air stream, m³/s

With the supply and exhaust inlet air mass flow rates set, the compound object then calls the generic air-to-air heat exchanger model to determine its supply air and exhaust air exiting conditions based on the inputs specified in the heat exchanger object. The supply air and exhaust air fans are then modeled to determine the final conditions of the air streams exiting

the stand alone energy recovery ventilator. The heat exchanger and fan models are described in detail elsewhere in this document (reference: HeatExchanger:AirToAir:SensibleAndLatent and Fan:OnOff).

The sensible heat transfer rate to the zone by the stand alone ventilator is then calculated as follows:

$$\dot{Q}_{Sensible} = \dot{m}_{Supply} (h_{SupplyOutlet} - h_{ExhaustInlet})_{HRmin}$$

where:

$\dot{Q}_{Sensible}$ = sensible energy transfer rate to the zone, W

\dot{m}_{Supply} = mass flow rate of the supply air stream, kg/s

$h_{SupplyOutlet}$ = enthalpy of the air being supplied to the zone, J/kg

$h_{ExhaustInlet}$ = enthalpy of the air being exhausted from the zone through the ventilator, J/kg

HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the supply air outlet or the exhaust air inlet

The resulting sensible energy transfer rate is passed to the zone equipment manager and added to the zone load to be met by other heating or cooling equipment. Since the stand alone ERV is intended to reduce the outdoor air load through heat exchange and not meet that load completely, the stand alone heat exchanger must be modeled first in the list of zone equipment. This is accomplished by setting the stand alone ERV priority for cooling and heating higher than that of other zone cooling or heating equipment (reference: ZoneHVAC:EquipmentList).

When economizer (free cooling) operation is desired, a controller is coupled to the stand alone ERV by providing the name of the controller object in the ERV controller input field. This controller determines when the air-side economizer is active (i.e., air flow is fully bypassed around a fixed-plate heat exchanger or the rotation of a rotary heat exchanger is stopped) based on the controller inputs (Ref. ZoneHVAC:EnergyRecoveryVentilator:Controller).

At the end of each HVAC simulation time step, this compound object reports the heating or cooling rate and energy delivered to the zone, as well as the electric power and consumption by the ventilator. In terms of thermal energy delivered to the zone, the sensible, latent and total energy transfer rate to the zone is calculated as follows:

$$\dot{Q}_{Total} = \dot{m}_{Supply} (h_{SupplyOutlet} - h_{ExhaustInlet})$$

$$\dot{Q}_{Sensible} = \dot{m}_{Supply} (h_{SupplyOutlet} - h_{ExhaustInlet})_{HRmin}$$

$$\dot{Q}_{Latent} = \dot{Q}_{Total} - \dot{Q}_{Sensible}$$

where:

\dot{Q}_{Total} = total energy transfer rate to the zone, W

$\dot{Q}_{Sensible}$ = sensible energy transfer rate to the zone, W

- \dot{Q}_{Latent} = latent energy transfer rate to the zone, W
 \dot{m}_{Supply} = mass flow rate of the supply air stream, kg/s
 $h_{SupplyOutlet}$ = enthalpy of the air being supplied to the zone, J/kg
 $h_{ExhaustInlet}$ = enthalpy of the air being exhausted from the zone through the ventilator, J/kg
 HR_{min} = enthalpies evaluated at a constant humidity ratio, the minimum humidity ratio of the supply air outlet or the exhaust air inlet

Since each of these energy transfer rates can be calculated as positive or negative values, individual reporting variables are established for cooling and heating and only positive values are reported. The following calculations are representative of what is done for each of the energy transfer rates:

$$\begin{aligned}
 &\text{IF } (\dot{Q}_{Total} < 0.0) \text{ THEN} \\
 &\quad \dot{Q}_{TotalCooling} = \text{ABS}(\dot{Q}_{Total}) \\
 &\quad \dot{Q}_{TotalHeating} = 0.0 \\
 &\text{ELSE} \\
 &\quad \dot{Q}_{TotalCooling} = 0.0 \\
 &\quad \dot{Q}_{TotalHeating} = \dot{Q}_{Total}
 \end{aligned}$$

where:

$$\begin{aligned}
 \dot{Q}_{TotalCooling} &= \text{output variable 'Stand Alone ERV Zone Total Cooling Rate, W'} \\
 \dot{Q}_{TotalHeating} &= \text{output variable 'Stand Alone ERV Zone Total Heating Rate, W'}
 \end{aligned}$$

In addition to heating and cooling rates, the heating and cooling energy supplied to the zone is also calculated for the time step being reported. The following example for total cooling energy is representative of what is done for the sensible and latent energy as well as the heating counterparts.

$$Q_{TotalCooling} = \dot{Q}_{TotalCooling} * TimeStepSys * 3600.$$

where:

$$\begin{aligned}
 Q_{TotalCooling} &= \text{output variable 'Stand Alone ERV Zone Total Cooling Energy, J'} \\
 TimeStepSys &= \text{HVAC system simulation time step, hr}
 \end{aligned}$$

Unit Heater

(Note: Some of this information also appears in the Input Output Reference for EnergyPlus. It is repeated here for clarity.)

The input object ZoneHVAC:UnitHeater provides a model for unit heaters that are zone equipment units which are assembled from other components and are a simplification of unit ventilators. They contain only a fan and a heating coil. These components are described elsewhere in this document. The unit heater input simply requires the names of these components, which have to be described elsewhere in the input. The input also requires the name of an availability schedule, maximum airflow rate, and maximum and minimum hot water volumetric flow rates. The unit is connected to the zone inlet and exhaust nodes by specifying unit inlet and outlet node names. Note that the unit air inlet node should be the same as a zone exhaust node and the unit outlet node should be the same as a zone inlet node.

Controls

While the control of the heating coil is similar to the fan coil units and the unit ventilator, the overall control of the unit heater is much different. There are four different modes in which a unit heat can operate based on the user input:

OFF: In this mode, the unit has been scheduled off. All flow rates are set to zero, and the temperatures are set to zone conditions.

NO LOAD OR COOLING/ON-OFF FAN CONTROL: In this mode, the unit is available, but there is no heating load. With On-Off fan control, the fan will only run when there is a heating load. Since there is no heating load in this mode, all flow rates are set to zero, and the temperatures are set to zone conditions. Since the unit heater is designed only to provide heating, the presence of a cooling load signifies that the unit should not be running.

NO LOAD OR COOLING/CONTINUOUS FAN CONTROL: In this mode, the unit is available and the fan is controlled to be running continuously. If it is scheduled to be available, the fan runs and circulates air to the space. While no direct heating is provided, any heat added by the fan is introduced into the space with the circulation of the air. If the fan is scheduled off, the fan will not run (this is identical to on-off control with no load).

HEATING: In this mode, the unit and fan are on/available, and there is a heating load. The heating coil is modulated (constant fan speed) to meet the heating load. Control of the heating coil and its flow rate is identical to the fan coil unit and unit ventilator. The flow rate of air through the unit is controlled by the user input and schedules.

Unit Ventilator

(Note: Some of this information also appears in the Input Output Reference for EnergyPlus. It is repeated here for clarity.)

The input object ZoneHVAC:UnitVentilator provides a model for unit ventilators that are zone equipment units which are assembled from other components. They contain a built-in outdoor air mixer, a fan, a heating coil, and a cooling coil. These components are described elsewhere in this document, except the built-in outdoor air mixer which is contained within the unit ventilator statement. The unit ventilator input simply requires the names of these other three components, which have to be described elsewhere in the input. The input also requires the name of an availability schedule, maximum airflow rate, outdoor air control information (control type and schedules), an outdoor airflow rate, and maximum and minimum hot and cold water mass flow rates. The unit is connected to the zone inlet and exhaust nodes and the outdoor air by specifying unit inlet, outlet, outdoor air and exhaust (relief) air node names. Note that the unit air inlet node should be the same as a zone exhaust node and the unit outlet node should be the same as a zone inlet node. In general, the unit ventilator input is very similar to the fan coil unit input, and the unit is connected to a hot water loop (demand side) through its hot water coil and to a chilled water loop (demand side) through its cooling coil.

Controls and Outdoor Air

The main difference between the fan coil and unit ventilator input is that the unit ventilator has a built-in outdoor air mixer with its own specialized controls. The outdoor air control type can be selected from one of the following options: “variable percent”, “fixed temperature” or “fixed amount”. In fixed temperature control, the amount of outdoor air is varied between the minimum outdoor air fraction (specified by a schedule) and 100% outdoor air to obtain a mixed air temperature as close as possible to the temperature schedule defined in the input. Variable percent control will also vary the amount of outdoor air between the minimum and maximum fractions (both specified in input by the user) to meet the load without the use of a coil if possible. In fixed amount control, the outdoor air flow rate is fixed to the specified value by the user. In this control strategy, the maximum outdoor air flow rate and schedule are automatically set to be equal to the minimum outdoor air flow rate and schedule. These control types are based on the 2004 ASHRAE Systems and Equipment Handbook (pp. 31.1-31.3) description of unit ventilator systems.

The unit is controlled to meet the zone (remaining) heating or cooling demand. If there is a heating demand, the cooling coil is off and the hot water flow through the heating coil is throttled to meet the demand. The hot water control node must be specified (same as the hot water coil inlet node) as well as maximum and minimum possible hot water volumetric flow rates. If there is a cooling demand from the zone, the hot water coil is off and the chilled water flow through the cooling coil is throttled to meet the load. The cooling coil control node must be specified (same as the cooling coil inlet node) and the maximum and minimum chilled water volumetric flow rates must be given. Finally both heating and cooling require a convergence tolerance, which is the tolerance denoting how closely the fan coil unit will meet the heating or cooling load. The tolerance is always relative to the zone load.

Overall, control of the unit must consider the outdoor air. Here is a more detailed description of the overall unit control:

OFF: Unit is schedule off or there is no load on it. All flow rates are set to zero and the temperatures are set to zone conditions (except for the outdoor air inlet). Outdoor air requirements will not override this condition.

HEATING/NO COIL/VARIABLE PERCENT: The unit is on, there is a heating load, no heating coil is present or it has been scheduled off, and variable percent outdoor air control type has been specified. In this case, the variable percent outdoor air controls what happens with the outdoor air. If the outside temperature is greater than the return temperature, then the outdoor air is set to the maximum as defined by the user input. If the outdoor air temperature is less than the return temperature from the zone, then the outdoor air is set to the minimum outdoor air flow rate as defined by the user. Since a coil is not present to further condition the supply air, the zone simply receives whatever temperature air results from the outdoor air controls.

HEATING/NO COIL/FIXED TEMPERATURE: The unit is on, there is a heating load, no heating coil is present or it has been scheduled off, and fixed temperature has been specified. The unit ventilator tries to use outdoor air as best as possible to meet the temperature goal. If it cannot meet this goal because the temperature goal is not between the zone return temperature and the outdoor air temperature, then the unit ventilator will either use the maximum or minimum outdoor air flow rate.

HEATING/NO COIL/FIXED AMOUNT: The unit is on, there is a heating load, no heating coil is present or it has been scheduled off, and fixed amount control has been specified. The unit ventilator fixes the outdoor air flow rate as defined by the user and sets the maximum and minimum outdoor air flow rate to be equal in order to avoid the variation of outdoor air flow rate between the maximum and minimum values. Since a coil is not present to further condition the supply air, the zone simply receives whatever temperature air results from the outdoor air controls.

HEATING/WITH COIL/VARIABLE PERCENT: The unit is on, there is a heating load, and variable percent control is specified. The outdoor air fraction is set to the minimum outdoor air

fraction (schedule based), and the heating coil is activated. The heating coil attempts to meet the remaining load on the zone being served by the unit ventilator.

HEATING/WITH COIL/FIXED AMOUNT: The unit is on, there is a heating load, a heating coil is present and is scheduled on, and fixed amount control has been specified. The unit ventilator fixes the outdoor air flow rate as defined by the user and sets the maximum and minimum outdoor air flow rate to be equal in order to avoid the variation of outdoor air flow rate between the maximum and minimum values. The heating coil then attempts to meet any remaining zone heating load.

COOLING/NO COIL/VARIABLE PERCENT: The unit is on, there is a cooling load, no coil is present or it has been scheduled off, and variable percent outdoor air control type has been specified. In this case, the variable percent outdoor air controls what happens with the outdoor air. If the outside temperature is greater than the return temperature, then the outdoor air is set to the minimum as defined by the user input. If the outdoor air temperature is less than the return temperature from the zone, then the outdoor air is set to the maximum outdoor air flow rate as defined by the user. This may be somewhat simplistic in that it could result in overcooling of the space. However, since a temperature goal was not established, this is the best that can be done by the simulation. Since a coil is not present to further condition the supply air, the zone simply receives whatever temperature air results from the outdoor air controls.

COOLING/NO COIL/FIXED TEMPERATURE: The unit is on, there is a cooling load, no cooling coil is present or it has been scheduled off, and fixed temperature has been specified. The unit ventilator tries to use outdoor air as best as possible to meet the temperature goal. If it cannot meet this goal because the temperature goal is not between the zone return temperature and the outdoor air temperature, then the unit ventilator will either use the maximum or minimum outdoor air flow rate in the same fashion as the variable percent outdoor air control.

COOLING/NO COIL/FIXED AMOUNT: The unit is on, there is a cooling load, no cooling coil is present or it has been scheduled off, and fixed amount control has been specified. The unit ventilator fixes the outdoor air flow rate as defined by the user and sets the maximum and minimum outdoor air flow rate to be equal in order to avoid the variation of outdoor air flow rate between the maximum and minimum values. Since a coil is not present to further condition the supply air, the zone simply receives whatever temperature air results from the outdoor air controls.

COOLING/WITH COIL/VARIABLE PERCENT: The unit is on, there is a cooling load, a coil is present and is scheduled on, and variable percent outdoor air control type has been specified. In this case, the percentage of outdoor air is set to the minimum flow outdoor air flow rate. The coil then attempts to meet any remaining zone load.

COOLING/WITH COIL/FIXED TEMPERATURE: The unit is on, there is a cooling load, a cooling coil is present and is scheduled on, and fixed temperature has been specified. The unit ventilator tries to use outdoor air as best as possible to meet the temperature goal. If it cannot meet this goal because the temperature goal is not between the zone return temperature and the outdoor air temperature, then the unit ventilator will either use the maximum or minimum outdoor air flow rate in the same fashion as the fixed temperature outdoor air control for the “no coil” conditions. The cooling coil then attempts to meet any remaining zone load.

COOLING/WITH COIL/FIXED AMOUNT: The unit is on, there is a cooling load, a cooling coil is present and is scheduled on, and fixed amount control has been specified. The unit ventilator fixes the outdoor air flow rate as defined by the user and sets the maximum and minimum outdoor air flow rate to be equal in order to avoid the variation of outdoor air flow rate between the maximum and minimum values. The cooling coil then attempts to meet any remaining zone cooling load.

Note: the unit ventilator controls are strictly temperature based and do not factor humidity into the equation (not an enthalpy economy cycle but rather a simple return air economy cycle). In

addition, temperature predictions are not strict energy balances here in the control routine though in the mixing routine an energy balance is preserved.

Ventilated Slab

Model Overview

The input object ZoneHVAC:VentilatedSlab provides a model for ventilated slab systems that in general use outdoor air to “precool” slabs with colder nighttime air. This method of precooling the thermal mass of a space can be very effective when nighttime temperatures are low and the mass of the system is high enough to provide a significant amount of cooling potential during the day to counteract internal heat gains within a zone. Nearly all ventilated slabs are simple systems such as that shown in the right side of Figure 235. The fan is shown in a blow through position, but the model will allow either a blow or draw through configuration.

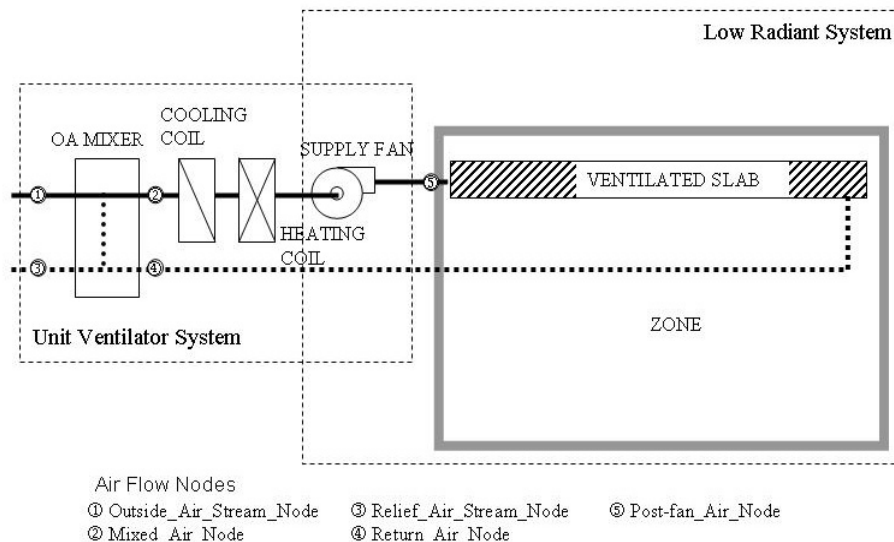


Figure 235. Basic System for the Ventilated Slab Module

It should be noted that in Figure 235 the use of “unit ventilator” and “low temperature radiant system” do not imply that the user must also specify these elements in the input file. The ventilated slab model combines aspects of these two existing EnergyPlus features into a single model that the user specifies through the input shown in the Input/Output Reference.

The ventilated slab system has been implemented in a fashion that is similar to the “unit ventilator” system in EnergyPlus. The unit ventilator is a system that allows the user to bring in outdoor air (ventilation) that may or may not be tempered with a heating or cooling coil as shown the left side of Figure 235. The air can be delivered to the slab only (Figure 236), to the slab then to the space (Figure 237), and to several slabs in different areas in series (Figure 238). The model essentially combines the functionality of the low temperature radiant system (using air as a transport medium rather than water) and the unit ventilator. In some cases, the system may not meet all the zone heating and cooling load because it is operated not by setpoint of the zone but control temperature range and coil outlet air temperature set by user input. Note that no coils are shown in Figure 238 for diagram simplicity but the implementation of the system shown in Figure 238 includes coils as in Figure 236 and Figure 237.

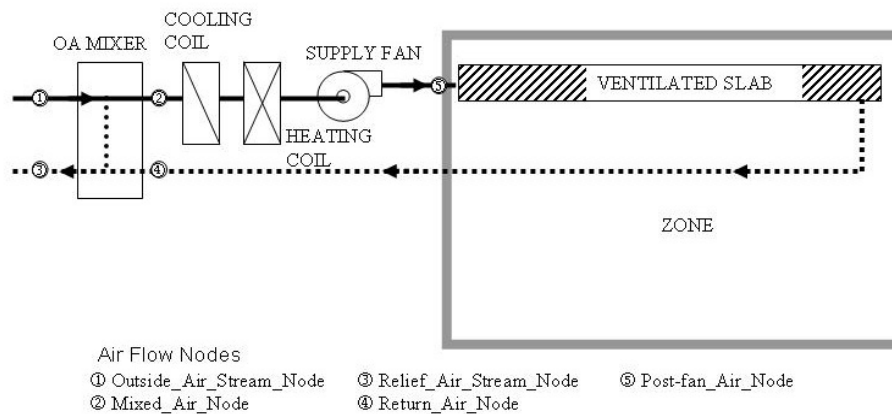


Figure 236. Model with Air Delivered to Slab. (Slab Only Mode)

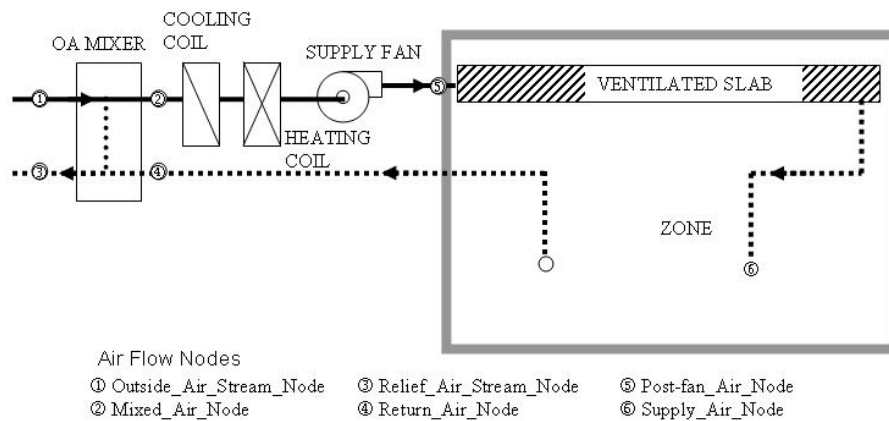


Figure 237. Zone Supply Model using Ventilated Slab (Slab and Zone mode)

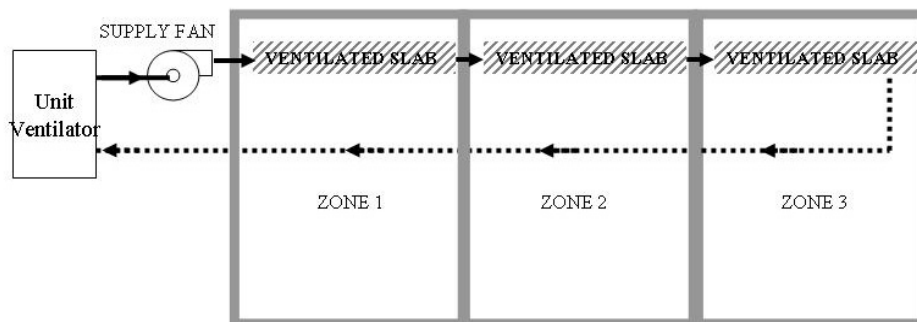


Figure 238. Multiple Slabs model with Several Zones (Series Slabs Mode)

Connections to the Heat Balances

The ventilated slab systems shown in the above diagrams connect or will connect to the various EnergyPlus heat balance equations in a variety of ways. All of the systems send outside or conditioned air through a slab or building element. This portion of the system acts in a fashion that is identical to the low temperature radiant systems. These surfaces that have the air being blown through them will impact the zone and air heat balances through the normal surface heat balances that interact with this surface. The ventilated slab will participate in the surface heat balances by exchanging radiation with other surfaces in the

zone and in the air heat balances via convection to the zone air. So, the ventilated slab is handled identically to the low temperature radiant systems with respect to the zone and air heat balances. This information is valid for all three of the systems shown in the figures above.

Figure 237. Zone Supply Model using Ventilating Slab (Slab and Zone mode) shows that the air, instead of being circulated through the slab and then sent back to the ventilation portion of the system, gets circulated through the space after it gets sent through the slab. When this system is implemented in EnergyPlus, it will have an additional impact on the zone air heat balance because air will be introduced into the space at conditions different than the zone air. Thus, it will have an energy impact on the air directly through the circulation of air from the slab to the zone and then back to the ventilated slab system. It will still have an impact on the slab surface convection and radiation as with the other systems.

For more information on the impact on the zone and air heat balances by the ventilated slab system, please consult the low temperature radiant system documentation and the EnergyPlus code.

Earthtube

The input object ZoneEarthtube provides a simple earth tube model that uses a complex ground heat transfer model to establish the temperature of the soil at the depth of the earth tube. The following information defines the basis for the model including the assumptions and mathematical equations. It supplements the information for the ZoneEarthtube input object given in the Input/Output Reference for EnergyPlus.

■ Input Requirement

Pipe : Pipe radius(m), Pipe thickness(m), Pipe length(m)

Distance between the pipe outer surface and undisturbed soil (m),

Pipe thermal conductivity (W/m-C),

Air velocity inside pipe(m/s), Depth of the radial center of pipe below ground (m)

Soil : Soil density(kg/m³), Soil specific heat(J/kg°C),

Soil thermal Conductivity(W/m°C), Absorption coefficient,

Fraction of evaporation rate

■ Assumption(s)

- Convection flow inside the pipe is hydrodynamically and thermally developed.
- Soil temperature in the pipe vicinity is uniform after the particular distance from the center of the pipe(thickness of the annulus), so that pipe surface temperature is uniform after the distance 'r' from the center of the pipe, where 'r' is the pipe radius.
- The temperature profile in the pipe vicinity is not affected by the presence of the pipe, so that pipe surface temperature is uniform at axial direction.
- The soil surrounding the pipe has homogeneous thermal conductivity.
- Pipe has uniform cross section area at axial direction.

Wind velocity (m/s), u , is the annual average value. This is calculated from EnergyPlus weather data by averaging individual wind velocity values of the whole year. The convective heat transfer coefficient at the soil surface (W/m²°C), h_s , is function of wind speed u . According to McAdams(1954) h_s can be approximated by the following correlation (Krarti, 1995).

$$h_s = 5.7 + 3.8u$$

In case of h_e and h_r , they can be determined by the following equations.

$$h_e = h_s (1 + 0.0168af)$$

$$h_r = h_s (1 + 0.0168 a r_a f)$$

with $a = 103 \text{ Pa/}^\circ\text{C}$.

Average air temperature ($^\circ\text{C}$), T_{ma} , is also calculated from EnergyPlus weather data by averaging individual air temperature values of the whole year.

The appropriate value of hemispherical emittance of the ground surface, ϵ , is 0.93~0.96. Radiation constant (W/m^2), ΔR , depends on soil radiative properties, air relative humidity, and effective sky temperature. An appropriate value of ΔR according to Krarti (1995) is 63 W/m^2 .

The absorption coefficient, β , depends on the soil absorptance and shading condition. The coefficient β is approximately equal to one minus the soil surface albedo. Albedo depends on soil cover and moisture content. Albedo=0.1 corresponds to wet soils, albedo=0.2 to moderate soils, and albedo=0.3 to dry soils.

Average solar radiation (W/m^2), S_m , is determined from EnergyPlus weather data by averaging individual global horizontal solar radiation values of the whole year.

The fraction of evaporation rate, f , also depends mainly on the soil cover and the soil moisture level. Based on the results reported by Penman, it is recommended to estimate the fraction f as follows. For bare soil, f is directly proportional to soil moisture content. For instance, $f=1$ corresponds to saturated soils, $f=0.6\sim 0.8$ to wet soils, $f=0.4\sim 0.5$ to moist soils, $f=0.1\sim 0.2$ to arid soils. For dry soils, $f = 0$, since no evaporation occurs. For covered soils, the fraction f is obtained by multiplying 0.7 by the value of f for bare soil depending on the soil moisture content (Krarti, 1995).

Relative humidity, r_a , is also calculated from EnergyPlus weather data by averaging individual relative humidity values of the whole year.

The soil thermal diffusivity (m^2/s), α_s , and conductivity ($\text{W/m}^\circ\text{C}$), k_s , varies with the density and moisture content. According to the 1991 ASHRAE Handbook of HVAC Applications (Table 4, pp. 11.4), the following values are recommended under different conditions.

Soil condition	k_s (W/m $^\circ\text{C}$)	$\alpha_s \times 10^{-7}$ (m 2 /s)
Heavy soil, saturated	2.42	9.04
Heavy soil, damp solid masonry	1.30	6.45
Heavy soil, dry	0.865	5.16
Light soil, damp	-	-
Light soil, dry	0.346	2.80

Annual angular frequency, w , is equal to $1.992 \times 10^{-7} \text{ rad/s}$, and dampening depth (m), D , is calculated from the following equation:

$$D = \sqrt{\frac{2\alpha_s}{w}}$$

The value of δ is evaluated as follows.

$$\delta = \frac{1+i}{D}$$

Amplitude of the air temperature ($^\circ\text{C}$), T_{va} , can be evaluated from EnergyPlus weather data by dividing the difference between the maximum and minimum air temperature value of the whole year by two. Similarly, amplitude of the solar radiation (W/m^2), S_v , can also be

determined from weather data by dividing the difference between the maximum and minimum solar radiation value of the whole year by two.

Phase angle between the insolation and the air temperature (rad), ϕ_l , is calculated by subtracting insolation phase angle from air temperature phase angle. Phase angle of insolation and air temperature is the point from the beginning of the year at which the insolation and air temperature respectively reaches the minimum value among the whole year.

Phase constant of the air (sec), t_{0a} , is the time elapsed from the beginning of the year at which the air temperature reaches the minimum value in the year.

By using all the input parameters and variables described above, average soil surface temperature ($^{\circ}\text{C}$), T_m , amplitude of the soil surface temperature variation ($^{\circ}\text{C}$), A_s , phase constant of the soil surface (sec), t_0 , and phase angle difference between the air and soil surface temperature (rad), ϕ_s , can be evaluated as follows ¹⁾:

$$T_m = \frac{1}{h_e} [h_r T_{ma} - \varepsilon \Delta R + \beta S_m - 0.0168 h_s f b (1 - r_a)]$$

$$A_s = \left\| \frac{h_r T_{va} - \beta S_v e^{i\phi_l}}{(h_e + \delta k_s)} \right\|$$

$$t_0 = t_{0a} + \frac{\phi_s}{w}$$

$$\phi_s = -\text{Arg} \left[\frac{h_r T_{va} - \beta S_v e^{i\phi_l}}{(h_e + \delta k_s)} \right]$$

(Note: T_m , A_s , and t_0 are calculated by the CalcSoilSurfTemp program and are inputs to EnergyPlus. The remainder of this section describes what has been implemented in EnergyPlus.)

The symbols $\| \cdot \|$ and Arg denote the modulus and the argument of a complex number respectively. In order to calculate A_s and ϕ_s , the complex number under consideration can be rearranged as the following form:

$$\begin{aligned} \frac{h_r T_{va} - \beta S_v e^{i\phi_l}}{(h_e + \delta k_s)} &= \\ \frac{\left(h_e + \frac{k_s}{D} \right) (h_r T_{va} - \beta S_v \cos \phi_l) + \frac{k_s}{D} (-\beta S_v \sin \phi_l)}{\left(h_e + \frac{k_s}{D} \right)^2 + \left(\frac{k_s}{D} \right)^2} + \\ i \frac{\left(h_e + \frac{k_s}{D} \right) (-\beta S_v \sin \phi_l) - \frac{k_s}{D} (h_r T_{va} - \beta S_v \cos \phi_l)}{\left(h_e + \frac{k_s}{D} \right)^2 + \left(\frac{k_s}{D} \right)^2} \end{aligned}$$

Assuming a homogeneous soil of constant thermal diffusivity, the temperature at any depth z and time t can be estimated by the following expression ²⁾.

$$T_{z,t} = T_m - A_s \exp \left[-z \left(\frac{\pi}{365 \alpha_s} \right)^{1/2} \right] \cos \left\{ \frac{2\pi}{365} \left[t - t_0 - \frac{z}{2} \left(\frac{365}{\pi \alpha_s} \right)^{1/2} \right] \right\}$$

In this expression, the unit of time, t , and phase constant of the soil surface, t_0 , should be converted into days. Similarly, the unit of soil thermal diffusivity, α_s , should also be converted into m^2/days .

By integrating the expression with respect to depth, the average temperature of a vertical soil profile ranging between depth z_1 and z_2 ($^\circ\text{C}$) can be determined as follows ²⁾.

$$T_{z_1, z_2, t} = T_m + \frac{A_s}{(z_2 - z_1) \gamma \sqrt{2}} \left\{ e^{-\gamma z_1} \cos \left[\frac{2\pi}{365} (t - t_0 - z_1 L - 45.6) \right] - e^{-\gamma z_2} \cos \left[\frac{2\pi}{365} (t - t_0 - z_2 L - 45.6) \right] \right\}$$

where,

$$\gamma = (\pi / 365 \alpha_s)^{1/2}$$

$$L = \frac{1}{2} (365 / \pi \alpha_s)^{1/2}$$

As the final step with regard to the heat transfer between soil and earth tube system, thermal conductivity of air ($\text{W}/\text{m}^\circ\text{C}$), k_{air} , and kinetic viscosity of air (m^2/s), ν , should be calculated first ³⁾.

$$k_{air} = 0.02442 + (10^{-4} (0.6992 T_a))$$

$$\nu = 10^{-4} (0.1335 + 0.000925 T_a)$$

By using the values of thermal conductivity of air, k_{air} , and kinetic viscosity of air, ν , the convective heat transfer coefficient at the inner pipe surface ($\text{W}/\text{m}^2\text{C}$), h_c , can be evaluated. It is a function of Reynolds number, Re , and Nusselt number, Nu ⁴⁾, where

$$h_c = \frac{\text{Nu} k_{air}}{2r_1}$$

$$\text{Nu} = \frac{(f_a / 2)(\text{Re} - 1000) \text{Pr}}{1 + 12.7 (f_a / 2)^{1/2} (\text{Pr}^{2/3} - 1)}$$

$$f_a = (1.58 \ln \text{Re} - 3.28)^{-2}$$

$$\text{Re} = \frac{2r_1 V_a}{\nu}$$

$$\text{Pr} = \frac{\nu}{\alpha_{air}}$$

where r_1 is inner pipe radius (m), and V_a is average pipe air velocity (m/s).

After determining the convective heat transfer coefficient, R_c , R_p and R_s are respectively calculated as follows.

$$R_c = \frac{1}{2\pi r_1 L h_c}$$

$$R_p = \frac{1}{2\pi L k_p} \ln \frac{r_1 + r_2}{r_1}$$

$$R_s = \frac{1}{2\pi L k_s} \ln \frac{r_1 + r_2 + r_3}{r_1 + r_2}$$

where R_c is thermal resistance due to convection heat transfer between the air in the pipe and the pipe inner surface ($^{\circ}\text{C}/\text{W}$), R_p is thermal resistance due to conduction heat transfer between the pipe inner and outer surface ($^{\circ}\text{C}/\text{W}$), and R_s is thermal resistance due to conduction heat transfer between the pipe outer surface and undisturbed soil ($^{\circ}\text{C}/\text{W}$). In addition r_2 is pipe thickness (m), r_3 is distance between the pipe outer surface and undisturbed soil (m), and L is pipe length (m).

Finally, the heat transfer between the soil and the air inside the pipe is equal to the amount of heat losses as air flows along the pipe (Jacovides and Mihalakakou, 1995).

$$U_t [T_a(y) - T_{z,t}] = -\dot{m}_a C_a [dT_a(y)]$$

with

$$U_t = \frac{1}{R_t}$$

$$R_t = R_c + R_p + R_s$$

where U_t is overall heat transfer coefficient of the whole earth tube system ($\text{W}/^{\circ}\text{C}$), $T_a(y)$ is air temperature of the pipe at the distance y from the pipe inlet ($^{\circ}\text{C}$), and \dot{m}_a is mass flow rate of ambient air through pipe (kg/s). C_a is specific heat of air ($\text{J}/\text{kg}^{\circ}\text{C}$) and R_t is total thermal resistance between pipe air and soil ($^{\circ}\text{C}/\text{W}$).

Initial condition of inlet air temperature is equal to the ambient air temperature. Outlet air temperature is finally evaluated by solving the heat transfer equation above.

Table 59. Nomenclature for Earthtube Model

Variable	Description	Units
A_s	amplitude of the soil surface temperature variation	($^{\circ}\text{C}$)
C_a	specific heat of air	($\text{J}/\text{kg}^{\circ}\text{C}$)
h_c	convective heat transfer coefficient at the inner pipe surface	($\text{W}/\text{m}^2^{\circ}\text{C}$)

h_s	convective heat transfer coefficient at the soil surface	(W/m ² °C)
k_{air}	thermal conductivity of the air	(W/m°C)
k_p	pipe thermal conductivity	(W/m°C)
k_s	soil thermal conductivity	(W/m°C)
L	pipe length	(m)
m_a	mass flow rate of ambient air through pipe	(kg/s)
r_a	relative humidity	
R_c	thermal resistance due to convection heat transfer between the air in the pipe and the pipe inner surface	(°C/W)
R_p	thermal resistance due to conduction heat transfer between the pipe inner and outer surface	(°C/W)
R_s	thermal resistance due to conduction heat transfer between the pipe outer surface and undisturbed soil	(°C/W)
R_t	total thermal resistance between pipe air and soil	(°C/W)
ΔR	radiation constant	(63W/m ²)
r_1	inner pipe radius	(m)
r_2	pipe thickness	(m)
r_3	distance between the pipe outer surface and undisturbed soil	(m)
S_m	average solar radiation	(W/m ²)
S_v	amplitude of the solar radiation	(W/m ²)
t	time elapsed from beginning of calendar year	(days)
$T_a(y)$	air temperature of the pipe at the distance y from the pipe inlet	(°C)
T_m	average soil surface temperature	(°C)
T_{ma}	average air temperature	(°C)
t_0	phase constant of the soil surface	(sec; days)
t_{0a}	phase constant of the air	(sec; days)
T_{va}	amplitude of the air temperature	(°C)
$T_{z,t}$	ground temperature at time t and depth z	(°C)
$T_{z1,z2,t}$	soil profile temperature at time t, averaged over depths between z_1 and z_2	(°C)
u	wind velocity above the ground surface	(m/s)
U_t	overall heat transfer coefficient of the whole earth tube system	(W/°C)
V_a	average pipe air velocity	(m/s)
z	depth of the radial center of pipe below soil surface	(m)
z_1	upper bounds of some vertical profile in soil	(m)
z_2	lower bounds of some vertical profile in soil	(m)
α_s	soil thermal diffusivity	(m ² /s; m ² /days)
β	soil absorption coefficient (= 1 – soil albedo)	
ϵ	hemispherical emittance of the ground surface	
φ_l	phase angle between the insolation and the air temperature	(rad)
Φ_s	phase angle difference between the air and soil surface temperature	(rad)
ν	kinetic viscosity of air	(m ² /s)
w	annual angular frequency (=1.992 x 10 ⁻⁷ rad/s)	

References

Krarti M., Lopez-Alonzo C., Claridge D. E. and Kreider J. F. 1995. Analytical model to predict annual soil surface temperature variation. *Journal of Solar Energy Engineering* 117, 91~99

Labs K. In: Cook J., editor. 1989. Passive cooling. Cambridge Massachusetts, London, England: MIT Press

Al-Ajmi F., Loveday D. L. and Hanby V. I. 2005. The Cooling Potential of Earth-air Heat Exchangers for Domestic Buildings in a Desert Climate, Building and Environment

Necati Ozisik M. 1885. Heat transfer: A basic approach, McGraw-Hill Book Company

Jacovides C. P. and Mihalakakou G. 1995. An Underground Pipe Systems as an Energy Source for Cooling/Heating Purposes. Renewable Energy 6, pp.893~900

Thermal Chimney Model

The ZoneThermalChimney input object is available for modeling a thermal chimney which is a device that uses stack driven air movement to ventilate spaces within a building. These systems have been used successfully in buildings as small as the size of an outhouse up to large commercial buildings. The air within a thermal chimney is heated naturally using solar energy. The air increases in temperature which causes its density to drop. This drop in density results in a natural vertical movement of air and a local drop in pressure. The drop in pressure is relieved by drawing air from the building interior, and the heat gained within the thermal chimney does not enter the occupied portion of the building. These systems are often used in support of natural ventilation systems. The EnergyPlus model will seek to model the air heat balance effects of the thermal chimney, balance air movements caused by the thermal chimney, and report other appropriate system variables. The new model will be linked into the air heat balance in a fashion similar to the current infiltration and simple ventilation models but will not be linked to an HVAC air loop. Any flow through the thermal chimney will be accounted for in the air mass balance in the HVAC calculations. However, other sophisticated ventilation strategies can be handled by other existing EnergyPlus components.

- Thermal Chimney Input Requirements
 - Distance from the top of thermal chimney to each inlet (m),
 - Relative ratio of air flow rates passing through each inlet,
 - Width of the absorber wall (m),
 - Discharge coefficient,
 - Cross sectional area of air channel outlet (m²),
 - Cross sectional areas of each air channel inlet (m²)
- Assumptions
 - Surface temperature of the glass cover is uniformly distributed.
 - Surface temperature of the absorber wall is uniformly distributed.
 - The inlet temperature of the air channel in the thermal chimney is equal to the room air temperature.
 - Resistance to the air flow due to the surface friction is negligible.
 - The discharged amount of interior air induced by the thermal chimney is replaced by the outdoor air infiltration.

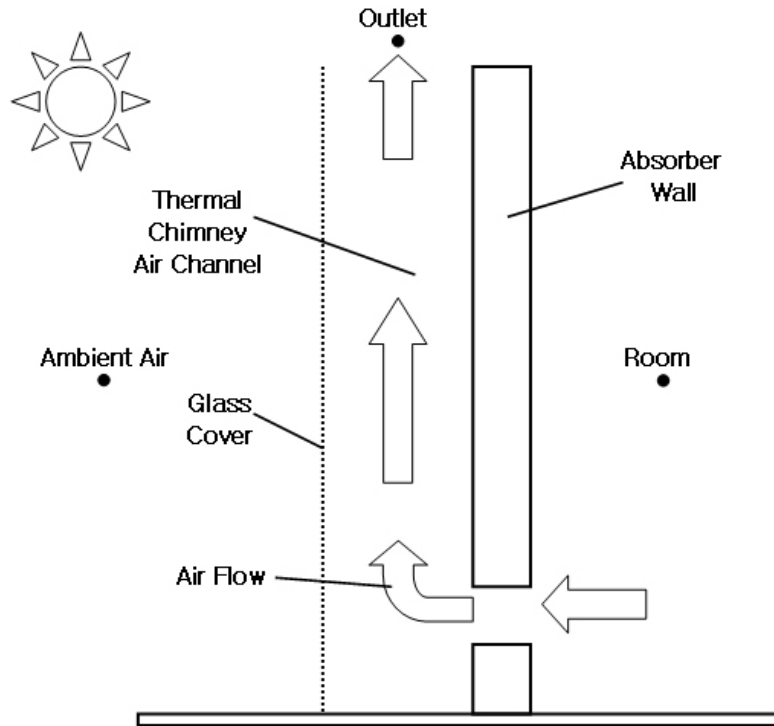


Figure 239. Basic Composition of Thermal Chimney

Mathematical model currently available for thermal chimneys has the capability to handle the thermal chimney having only one inlet. In other words, it is unlikely that thermal chimneys with multiple inlets due to multiple stories utilizing the common thermal chimney can be mathematically modeled without computational fluid dynamics. Therefore, if the thermal chimney to be modeled has multiple inlets, it will be assumed that it will have only one inlet. For this assumption, the user will be required to specify the relative ratio of air flow rates passing through each inlet to compute the overall length of the thermal chimney (m), L , overall room air temperature (K), T_r , and overall cross sectional area of air channel inlet (m^2), A_i , as follows:

$$L = \frac{L_1 r_1 E_1 + L_2 r_2 E_2 + L_3 r_3 E_3 + \dots + L_n r_n E_n}{r_1 E_1 + r_2 E_2 + r_3 E_3 + \dots + r_n E_n}$$

$$T_r = T_{r1} r_1 + T_{r2} r_2 + T_{r3} r_3 + \dots + T_{rn} r_n$$

$$A_i = A_{i1} + A_{i2} + A_{i3} + \dots + A_{in}$$

Where, A_{in} is the cross sectional area of nth air channel inlet (m^2), L_n is the distance from the top of thermal chimney to nth inlet (m), E_n is the room air specific enthalpy corresponding to nth inlet (J/kg), r_n is the relative ratio of air flow rate passing through nth inlet and T_{rn} is the room air temperature corresponding to nth inlet (K). Among them, room air specific enthalpy, E_n , and room air temperature corresponding to each inlet, T_{rn} , are directly calculated inside EnergyPlus. In addition, the relative ratios should meet the following expression:

$$r_1 + r_2 + r_3 + \dots + r_n = 1$$

After merging the multiple inlets into a single inlet condition based on the description above, the following algorithm which is widely used is employed for the modeling of the thermal chimney.

The key output parameter in the thermal chimney model is the enhanced amount of natural ventilation rate caused by the presence of a thermal chimney. In order to determine the enhanced ventilation, the discharge air temperature from a thermal chimney should be calculated, which, in turn, should be computed based on the information on the absorber wall temperature, glass cover temperature and the vertical air temperature distribution within the thermal chimney. Among them, energy balances for the absorber wall and the glass cover are carried out using the existing algorithm currently available in EnergyPlus, which has the similar approach to the Trombe wall. On the other hand, the vertical air temperature distribution and the resultant discharge air temperature of the thermal chimney are computed using the separate thermal chimney algorithm described in the following paragraphs.

Once the glass cover temperature and the absorber wall temperature are computed using the existing modeling algorithm in EnergyPlus, the energy balance for the fluid (air inside the thermal chimney) can be expressed as:

$$h_{wf}(T_w - \overline{T_f}) = h_{gf}(\overline{T_f} - T_g) + \frac{\dot{m}C_p}{w} \frac{dT_f}{dx}$$

Where, m is the total mass flow rate of the air (kg/s), C_p is the specific heat of air (J/kg°C), w is the width of the absorber wall (m) and x is the elemental length of the absorber wall (m).

Since the initial condition of inlet air temperature in this differential equation is equal to the room air temperature (i.e. $x = 0$, $T_{fi} = T_r$), the outlet air temperature, T_{fo} , can be finally evaluated.

Finally, the total air flow rate caused by the thermal chimney (m^3/s), Q , can be evaluated from the following expression [1]:

$$Q = C_d A_o \sqrt{\frac{2\left(\frac{T_{fo} - T_r}{T_r}\right)gL}{(1 + A_r)^2}}$$

$$A_r = A_o / A_i$$

Where, C_d is the discharge coefficient, A_o and A_i is the cross sectional areas of air channel outlet and inlet (m^2), respectively, T_{fo} is the outlet air temperature (K), T_r is the room air temperature (K) and L is the total length of the thermal chimney (m).

Since multiple inlets are merged into a single inlet in the beginning, the air flow rate passing through each inlet due to the existence of the thermal chimney can be finally determined as follows:

$$Q_1 = Qr_1, \quad Q_2 = Qr_2, \quad Q_3 = Qr_3, \quad \dots, \quad Q_n = Qr_n$$

Where, Q_n is the air flow rate passing through nth inlet (m^3/s) and r_n is the relative ratio of air flow rate passing through nth inlet.

The discharged amount of interior air from each zone caused by the presence of the thermal chimney is assumed to be replaced by the outdoor air infiltration.

Table 60. Nomenclature of Thermal Chimney Model

Model Nomenclature	
Variable	Description
A_i	cross sectional area of air channel inlet (m^2)
A_o	cross sectional area of air channel outlet (m^2)
C_d	discharge coefficient
C_p	specific heat of air ($J/kg^\circ C$)
E_n	room air specific enthalpy corresponding to nth inlet (J/kg)
g	acceleration due to gravity ($9.8 m/s^2$)
h_{gam}	convective heat transfer coefficients between the glass and ambient air ($W/m^2^\circ C$)
h_{gf}	convective heat transfer coefficients between the glass and the fluid ($W/m^2^\circ C$)
h_{iw}	convective heat transfer coefficients between absorber wall inner surface and the room air ($W/m^2^\circ C$)
h_{wf}	convective heat transfer coefficients between absorber wall and the fluid ($W/m^2^\circ C$)
h_{wind}	The convective heat transfer coefficient due to the wind ($W/m^2^\circ C$)
H_{sr}	incident solar radiation on vertical surface (W/m^2)
k_{air}	thermal conductivity of air ($W/m^\circ C$)
L	total length of the thermal chimney (m)
m	mass flow rate of the air (kg/s)
r_n	relative ratio of air flow rate passing through nth inlet
S_g	solar radiation absorbed by the glass cover (W/m^2)
S_w	solar radiation absorbed by the absorber wall (W/m^2)
T_{am}	ambient air temperature (K)
T_f	fluid temperature averaged over the entire length of the thermal chimney (K)
T_{fi}	inlet air temperature of the thermal chimney (K)
T_{fo}	outlet air temperature of the thermal chimney (K)
T_g	glass cover temperature (K)
T_r	room air temperature (K)
T_s	surface temperature (K)
T_w	absorber wall temperature (K)
T_∞	fluid temperature (K)
u	wind speed (m/s)
U_w	Overall heat transfer coefficient from the room air and the absorber wall ($W/m^2^\circ C$)
w	width of the absorber wall (m)
x	elemental length of the absorber wall (m)
α_g	absorptance of glass cover
α_w	absorptance of absorber wall
β	air volumetric coefficient of expansion (K^{-1})
ϵ_g	emissivity of the glass cover
σ	Stefan-Boltzmann constant ($5.67 \times 10^{-8} W/m^2 K^4$)
τ	transmittance of the glass cover
ν	kinematic viscosity of air (m^2/s)

References

- N. K. Bansal, R. Mathur and M. S. Bhandari, Solar Chimney for enhanced Stack Ventilation, Building and Environment, 28, pp.373-377, 1993
- K. S. Ong, A Mathematical Model of a Solar Chimney, Renewable Energy, 28, pp.1047-1060, 2003
- N. K. Bansal, R. Mathur and M. S. Bhandari, A Study of Solar Chimney Assisted Wind Tower System for Natural Ventilation in Buildings, Building and Environment, 29, pp.495-500, 1994

J. Marti-Herrero and M. R. Heras-Celemin, Dynamic Physical Model for a Solar Chimney, *Solar Energy*, 81, pp. 614-622, 2007

M. M. Aboulnaga and S. N. Abdrabboh, Improving Night Ventilation into Low-rise Buildings in Hot-arid Climates Exploring a Combined Wall-roof Solar Chimney, *Renewable Energy*, 19, pp. 47-54, 2000

CoolTower

Overview

The input object ZoneCoolTower:Shower is available for modeling a cooltower (which is sometimes referred to as a wind tower or a shower cooling tower) which is a component that is intended to model a passive draught evaporative cooling (PDEC) that is designed to capture the wind at the top of a tower and cool the outdoor air using water evaporation before delivering it to a space. The air flow in these systems is natural as the evaporation process increases the density of the air causing it to fall through the tower and into the space without the aid of a fan. A cooltower typically consists of a water spray or an evaporative pad, a shaft, and a water tank or reservoir. Wind catchers to improve the wind-driven performance at the top of the tower are optional. Water is pumped over an evaporative device by water pump which is the only component consumed power for this system. This water cools and humidifies incoming air and then the cool, dense air naturally falls down through shaft and leaves through large openings at the bottom of cooltowers.

The shower cooling tower shown in figure below is controlled by a schedule and the specification of maximum water flow rate and volume flow rate as well as minimum indoor temperature. The actual flow rate of water and air can be controlled as users specify the fractions of water loss and flow schedule. The required input fields include effective tower height and exit area to obtain the temperature and flow rate of the air exiting the tower. A schedule and rated power for the water pump are also required to determine the power consumed. The component typically has a stand alone water system that is not added to the water consumption from mains. However, users are required to specify the water source through an optional field, the name of water supply storage tank, in case any water comes from a water main.

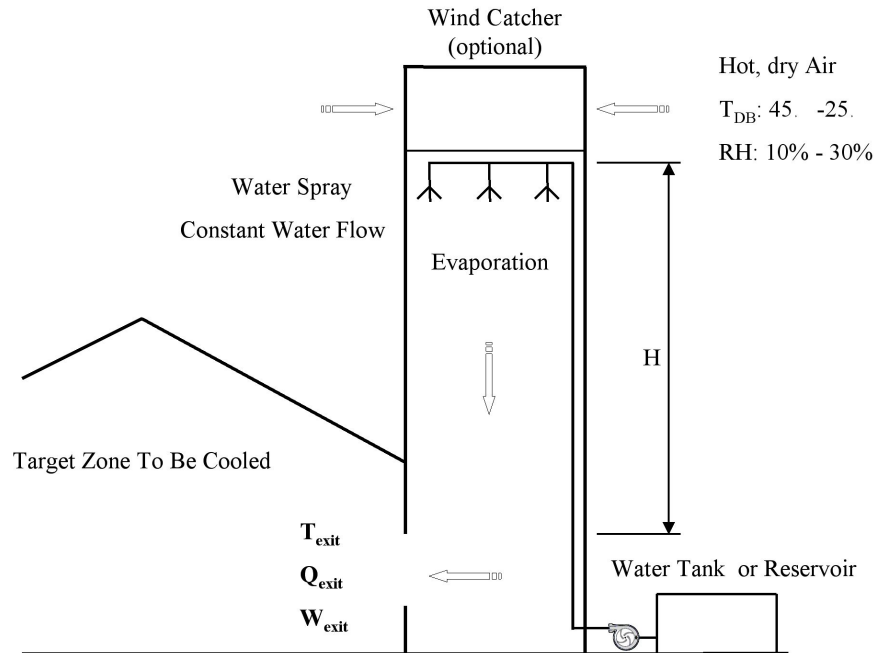


Figure 240. Cooltower Configuration. Temperatures in °C.

The cooltower model employing a model of the inertial shower cooling tower is intended to establish the actual mass flow rate of the air that leaves the cooltower and the evaporation rate consumed during the processes within the cooltower. Like infiltration, ventilation, and earth tubes, the air is assumed to be immediately mixed with the zone air. The determination of simultaneous heat and mass transfer that occurs during natural evaporative cooling in cooltower is complicated. Therefore, some assumptions have been made to obtain the conditions of the air and water. All cooltowers are executed at the start of each time step called by HVAC manager, and the conditions of air temperature and humidity ratio in the zone will be corrected with any other air that enters the zone.

All temperatures in the following descriptions are in degrees C, mass flow rates are in kg/s, and volume flow rates are in m³/s.

Model Description

The user must input the required information according to the Input Output Reference Manual (ref: ZoneCoolTower:Shower). The cooltower model requires a unique identifying name, an availability schedule, and the name of the zone being served. The schedule name must refer to a valid schedule type (range 0-1) and contain values of fractional cooltower operation as well as water pump operation. For the determination of the exit temperature and actual air volume flow rate, four additional inputs are required: effective tower height, exit area, fraction of flow schedule, and fraction of water loss. These define the conditions of the exit air, obtaining the conditions of outdoor air from the weather data and thus allow EnergyPlus to correct both the temperature and humidity level in the zone. The power consumed by the water pump can be directly determined by the schedule and rated power that the user inputs. The component is also controlled by the specification of minimum indoor temperature, maximum volume flow rate, and maximum water flow rate. These allow the model to prevent overcooling the zone or overestimation of the air volume flow rate. In addition, the user must input a flow control type according to the information that the user is able to obtain.

Simulation and Control

The cooltower model first determines the temperature and volume flow rate of the exit air. Both parameters can be directly determined in case of water flow schedule control when the

water flow rate is known. With the outdoor temperatures obtained from weather data, the exit air temperature (T_{out}) can be directly determined as functions of outdoor dry-bulb temperature (DB), outdoor wet-bulb temperature (WB), effective tower height (H) and water flow rate (WF) in l/min by using following equation.

$$T_{out} = DB - (DB - WB)(1 - \exp(-0.8H))(1 - \exp(-0.15WF))$$

The volume flow rate of the exit air (Q) can also be directly obtained as functions of water flow rate and effective tower height from the following equation.

$$Q = 0.0125WF \cdot H^{0.5}$$

In case of that the calculated air volume flow rate is greater than maximum air volume flow rate in this control, which leads to overestimation of actual volume flow rate of the exit air, the calculated air volume flow rate is replaced with the maximum.

For the simulation of wind-driven flow control where the water flow rate is unknown, the model determines velocity of the outlet air (V_{out}) as functions of effective tower height and wind speed of outdoor air (WS) as

$$V_{out} = 0.7H^{0.5} + 0.47(WS - 1)$$

The air volume flow rate (Q) is then calculated by

$$Q_{estimate} = A \cdot V_{out}$$

where A is opening area at the bottom of cooltower.

Substituting the air flow rate to the previous equation for Q, water flow rate is obtained as

$$WF = \frac{Q}{0.0125 \cdot H^{0.5}}$$

Once water flow rate is determined, the model checks the limit of water flow rate that the user inputs, so that the model prevents overestimation of actual volume flow rate of the exit air. If the calculated water flow rate is greater than the maximum water flow rate, the maximum will be chosen. The model also replaces the calculated air volume flow rate with the maximum volume flow rate from the user input when the calculated is greater than the maximum. Then, the model calculates the air volume flow rate and exit temperature using the previous equation for Q and T_{out} .

This cooltower model allows the user to specify water loss due to drift or blow down and the loss of air flow (example: a cooltower which delivers air to both the interior and exterior of the building). If the user inputs the fraction of water loss or flow schedule that means some amount of the air actually goes to outside, the fractional values will be applied to previously calculated ones so that the model calculates both actual water flow rate (WF_{actual}) and air volume flow rate (Q_{actual}) as follows:

$$WF_{actual} = WF(1.0 + Fraction)$$

$$Q_{actual} = Q(1.0 - Fraction)$$

The model then determines the exit humidity ratio (ω_{out}) from the relation of mass balances below.

$$\omega_{in}(\dot{m}_{air} + \dot{m}_{water}) = \omega_{out} \cdot \dot{m}_{air}$$

In this case, actual mass flow rate at the inlet and outlet of cooltower can not be correctly calculated with limited information. Thus, the model estimates initial conditions of the air based on the outdoor temperatures, the calculated exit air temperature, enthalpy of outdoor air (H_{in}) and outdoor barometric pressure (P). Assuming no enthalpy changes and pressure drops between inlet air and the initialized air, the humidity ratio and mass flow of the initialized air can be obtained by using EnergyPlus psychrometric functions and the following equation.

$$\omega_{in} = PsyWRnTdbTwbPb(T_{db_{in}}, T_{wb_{in}}, P)$$

$$\omega_{initial} = PsyWRnTdbH(T_{out}, H_{in}, P)$$

$$\rho_{initial_{air}} = PsyRhoAirFnPbTdbW(T_{out}, \omega_{initial}, P)$$

$$\dot{m}_{air} = \rho_{initial_{air}} \cdot Q_{actual}$$

where $\omega_{initial}$ and $\rho_{initial}$ are the humidity ratio and air density of the initialized air and ω_{in} is the humidity ratio of outdoor air.

Therefore, the humidity ratio of exit air, ω_{out} , is

$$\omega_{out} = \frac{\omega_{in}(\dot{m}_{air} + \dot{m}_{water})}{\dot{m}_{air}}$$

Once the humidity ratio at the exit is determined, the model can obtain the actual density (ρ_{air}), specific heat ($c_{p,air}$), and mass flow rate ($\dot{m}_{cooltower}$) of the air leaving cooltower by using EnergyPlus psychrometric function and following equation.

$$c_{p,air} = PsyCpAirFnWtdb(\omega_{out}, T_{out})$$

$$\rho_{air} = PsyRhoAirFnPbTdbW(T_{out}, \omega_{out}, P)$$

$$\dot{m}_{cooltower} = \rho_{air} \cdot Q_{actual}$$

Assuming that the water temperature equals to outdoor wet-bulb temperature, the model eventually determines density of the water and evaporation rate as follows.

$$\rho_{water} = RhoH2O(T_{wb_{in}})$$

$$Q_{water} = \frac{(\omega_{out} - \omega_{in})}{\rho_{water}} \cdot RhoH2O(T_{wb_{in}})$$

Generation

Electric Load Center Distribution Manager

The electric load center distribution manager (object name: ElectricLoadCenter:Distribution) operates generators specified in the simulation and reports the amount of generated and purchased electrical for the input file. The internal meters used by EnergyPlus for reporting do all of the demand tracking. For more details on the individual inputs required see the Input Output Reference Document.

The electric load center sums all of the building and system electric loads and provides operating schemes for the Generators and Photovoltaic cells to supply. What is not provided by the on-site generation equipment is met by purchasing off-site electric. The electricity produced from photovoltaic arrays is handled somewhat separately and is always “used” and will reduce the demand that the generators will try to meet for that time step.

The available operating schemes are “Baseload”, “DemandLimit”, “TrackElectrical,” “TrackSchedule,” “TrackMeter,” “FollowThermal” and “FollowThermalLimitElectrical”. The Baseload scheme operates the generators at their rated capacity when the generator is scheduled ON. The DemandLimit scheme limits the amount of purchased electrical from the utility to the amount specified in the input. The DemandLimit scheme tries to have the generators meet all of the demand above the purchased electric limit set by the user in the next field. The TrackElectrical scheme tries to have the generators meet all of the electrical demand for the building. The TrackMeter scheme tries to have the generators meet all the electrical demand from a meter chosen by the user rather the usual meter for the entire facility. The meter can be a custom meter so that generators are tied to only certain loads in the building. The TrackSchedule scheme tries to have the generators meet all the electrical demand determined by a user-defined schedule.

The FollowThermal and FollowThermalLimitElectrical schemes run the generators to meet thermal demand. The thermal demand is determined from the plant modeling and depends on the flow requested by other components on the demand side of the plant loop, the loop temperatures, and the loop temperature setpoint. The distribution manager converts the thermal load to an electrical load using a nominal ratio of the thermal to electrical power production for each generator. For these schemes, the generator needs to be connected to the supply side of a plant loop and serve components that use hot water on the demand side of the plant loop. The thermal load request is obtained from the plant data structure (structure location in code is `PlantLoop%LoopSide%Branch%Comp%MyLoad`). The distribution manager converts the thermal load, $q_{thermal}$, to an electrical load using:

$$q_{Elect} = \frac{q_{thermal}}{ThermElectRatio}$$

where,

ThermElectRatio a nominal, constant, user-defined value for the ratio of thermal production to electrical production for a cogenerator. This ratio is used for supervisory control and dispatch, the cogenerator model may determine performance to vary from this nominal value at different times in the simulation when conditions differ from those used for the nominal ratio.

Most of the operating schemes will sequentially load the available electric load centers and generators. Beginning in version 2.2, EnergyPlus can accept multiple “ElectricLoadCenter:Distribution” objects with different operating schemes. With this change there are now two levels of reporting, one for the whole building and a second for each load center. The whole-building results are managed with the internal meters for the entire model.

The load-center results are summed for those generators connected to the load center. The order of the input objects in the input file is used to structure how generators are dispatched with the first load centers and generators managed before the later ones. Therefore, load centers listed earlier in the file effectively have a higher priority. All demand not met by available generator capacity will be met by purchased electrical. If the available generators are not enough to meet demand, then purchased electrical will offset the difference. If a generator is needed in the simulation for a small load and it is less than the minimum part load ratio the generator will operate at the minimum part load ratio and the excess will either reduce demand or the excess energy will be exported back to the power company. The purchased electrical demand limit is the user input for the demand limit above which the generators will try and meet the entire electrical load on the building minus the photovoltaic array if available. It is possible to prescribe a set of ElectricLoadCenter:Distribution objects with inconsistent or conflicting operating schemes, so users need to be careful.

The total electric purchased are both in power and energy units. This value is positive when the amount of energy is purchased from the utility. This value can be negative when the total electric produced is greater than the facility electrical needs. The excess will either be available for storage (but storage is not yet available) or sell back to the power company.

Electric Load Center Generators

The electric load center generators (object name: ElectricLoadCenter:Generators) provide a set of scheduled electric generators for electric power generation. Here is where the user lists what generators and PV's are available at any given time. For more details on the individual inputs required see the Input Output Reference Document.

Generators

Internal Combustion Engine

The engine-driven generator model was originally developed for the BLAST program and was subsequently adapted for use in EnergyPlus. The model uses the following set of equations all of which are quadratic fits to the PLR (Part Load Ratio) of the generator. The coefficients must be derived from manufacturers data.

$$\frac{\text{electric energy output}}{\text{fuel energy input}} = \frac{\text{electric energy output}}{(\dot{m}_{\text{fuel}} \{ \text{kg} / \text{s} \} \cdot \text{LHV} \{ \text{J} / \text{kg} \})} = a_1 + a_2 \text{PLR} + a_3 \text{PLR}^2$$

The electrical load and engine generator nominal load capacity are used to compute the part load ratio.

$$\text{PLR} = \frac{\text{Electric energy output}}{\text{nominal generating capacity}}$$

The exhaust gas temp and flow rate are used if a stack heat exchanger is used to recover waste heat from the exhaust. This temperature is the inlet temperature to the heat exchanger which is modeled in a UA-effectiveness form:

$$\frac{\text{Total Exhaust heat}}{\text{fuel energy input}} = \frac{\text{Total Exhaust heat}}{(\dot{m}_{\text{fuel}} \{ \text{kg} / \text{s} \} \cdot \text{LHV} \{ \text{J} / \text{kg} \})} = d_1 + d_2 \text{PLR} + d_3 \text{PLR}^2$$

$$\frac{\text{Exhaust Gas Temp} \{ \text{K} \}}{\text{fuel energy input}} = \frac{\text{Exhaust Gas Temp} \{ \text{K} \}}{(\dot{m}_{\text{fuel}} \{ \text{kg} / \text{s} \} \cdot \text{LHV} \{ \text{J} / \text{kg} \})} = e_1 + e_2 \text{PLR} + e_3 \text{PLR}^2$$

The exhaust flow rate is then calculated as:

$$\dot{m}_{exhaust} = \frac{\text{Total Exhaust heat}}{Cp_{exhaust} \cdot (T_{exhaust} - T_{reference})}$$

where $T_{reference}$ is the reference temperature for the fuel lower heating value, and is given as 25°C in manufacturer's data, and

$$T_{stack} = T_{DesignMinExhaust} + \frac{(T_{exhaust} - T_{DesignMinExhaust})^2}{\exp\left(\frac{UA}{\dot{m}_{exhaust} Cp_{exhaust}}\right)}$$

Finally heat recovered from the lube oil and the water jacket are accounted for as follows:

$$\frac{\text{Recoverable jacket heat}}{\text{fuel energy input}} = \frac{\text{Recoverable jacket heat}}{(\dot{m}_{fuel} \{kg / s\} \cdot LHV \{J / kg\})} = b_1 + b_2 PLR + b_3 PLR^2$$

$$\frac{\text{Recoverable lube oil heat}}{\text{fuel energy input}} = \frac{\text{Recoverable lube oil heat}}{(\dot{m}_{fuel} \{kg / s\} \cdot LHV \{J / kg\})} = c_1 + c_2 PLR + c_3 PLR^2$$

The manufacturer must supply the recoverable water jacket heat, lube oil heat and exhaust heat and associated fuel consumption for a range of load conditions. This data is then fit to the PLR to obtain the fifteen a,b,c,d, and e coefficients.

Turbine Generator

The combustion turbine generator model was originally developed for the BLAST program and was subsequently adapted for use in EnergyPlus. The model uses the following set of equations all of which are equation fits to the PLR (Part Load Ratio) of the generator and the entering air temperature. The coefficients must be derived from manufacturers data. For electric power generated in Watts, the fuel input rate is calculated in J/s.

$$\frac{\text{fuel energy input rate}}{\text{electric power output}} = [a_1 + a_2 PLR + a_3 PLR^2] * [b_1 + b_2 \Delta T + b_3 \Delta T^2]$$

The electrical load and engine generator nominal load capacity are used to compute the part load ratio.

$$PLR = \frac{\text{Electric energy output}}{\text{nominal generating capacity}}$$

The temperature difference shows the deviation of ambient air temperature from the manufacturers design air temperature.

$$\Delta T = T_{air} - T_{design}$$

A second curve fit calculates the exhaust temperature (C) by multiplying the exhaust temperature (C) for a particular part load by a correction factor based on the deviation from design temperature.

$$T_{exhaust} = [c_1 + c_2 PLR + c_3 PLR^2] * [d_1 + d_2 \Delta T + d_3 \Delta T^2]$$

The exhaust gas temp is used if a stack heat exchanger is used to recover waste heat from the exhaust. This temperature is the inlet temperature to the heat exchanger which is modeled in a UA-effectiveness form:

$$T_{stack} = T_{DesignMinExhaust} + \frac{(T_{exhaust} - T_{DesignMinExhaust})^2}{\exp\left(\frac{UA}{\dot{m}_{exhaust} C_{p_{exhaust}}}\right)}$$

Where the design minimum exhaust temperature is a user input to the model and the exhaust mass flow rate and the UA are fit from manufacturers data as follows:

$$UA = e_3 (\text{Nominal Generating Capacity})^{e4}$$

$$\frac{\text{exhaust gas flow rate}}{\text{Nominal Generating Capacity}} = [f_1 + f_2 \Delta T + f_3 \Delta T^2]$$

Finally, heat recovered from the lube oil is accounted for as follows:

$$\frac{\text{Recoverable lube oil heat}}{\text{electric power generated}} = g_1 + g_2 PLR + g_3 PLR^2$$

Microturbine Generator

Microturbine generators are small combustion turbines that produce electricity on a relatively small scale (e.g., 25kW to 500kW). This model uses nominal performance at reference conditions along with several modifier curves to determine electrical power output and fuel use at non-reference conditions. The modifier curve coefficients must be derived from manufacturers data. Standby and ancillary power can also be taken into account.

Exhaust air energy recovery for heating water can also be modeled. Similar to electrical power output, thermal power (heat recovery to water) output is calculated using nominal performance at reference conditions with modifier curves to account for variations at non-reference conditions. The ElectricLoadCenter:Generators and ElectricLoadCenter:Distribution objects are used to define the availability and control of the electric generators included in the simulation (ref. ElectricLoadCenter:Generators and ElectricLoadCenter:Distribution).

For each simulation time step that the generator is being asked to operate (i.e., produce electrical power as determined by the ElectricLoadCenter), the full load electrical output of the generator is determined using the user-defined reference electrical power output along with a bi-quadratic modifier curve to account for differences in the combustion air inlet temperature and elevation for the current simulation time step compared to the reference temperature and elevation (i.e., the modifier curve should evaluate to 1.0 at the reference combustion air inlet temperature and reference elevation).

$$P_{Elec,FullLoad} = P_{Elec,Ref} (PowerFTempElev)$$

$$PowerFTempElev = a_1 + a_2 (T_{a,i}) + a_3 (T_{a,i})^2 + a_4 (Elev) + a_5 (Elev)^2 + a_6 (T_{a,i})(Elev)$$

where:

$P_{Elec,FullLoad}$ = Full load electrical power output (W)

$P_{Elec,Ref}$ = Reference Electrical Power Output, user input (W)

$PowerFTempElev$ = User-defined Electric Power Modifier Curve (function of temperature and elevation) evaluated at the current combustion air inlet temperature and elevation

$T_{a,i}$ = Combustion air inlet temperature (°C)

$Elev$ = Elevation (m). This value obtained from the Location object or the weather file.

The full load electrical power output of the generator is then checked against the minimum and maximum full load electrical power outputs specified by the user:

$$P_{Elec,FullLoad} = MIN(P_{Elec,FullLoad}, P_{FL_Max})$$

$$P_{Elec,FullLoad} = MAX(P_{Elec,FullLoad}, P_{FL_Min})$$

P_{FL_Max} = Maximum Full Load Electrical Power Output, user input (W)

P_{FL_Min} = Minimum Full Load Electrical Power Output, user input (W)

The actual (operating) electrical power output from the generator is determined next based on the load requested by the Electric Load Center, the generator's minimum and maximum part-load ratios, and the ancillary power.

$$P_{Elec,Operating} = MAX(0.0, (Load + P_{Ancillary}))$$

$$P_{Elec,Operating} = MIN(P_{Elec,Operating}, P_{Elec,FullLoad})$$

$$\begin{aligned}
 & \text{IF } (P_{Elec,Full Load} > 0.0) \text{ THEN} \\
 & \quad PLR = \frac{P_{Elec,Operating}}{P_{Elec,Full Load}} \\
 & \quad PLR = \text{MIN}(PLR, PLR_{\max}) \\
 & \quad PLR = \text{MAX}(PLR, PLR_{\min}) \\
 & \text{ELSE} \\
 & \quad PLR = 0.0 \\
 & \text{END IF} \\
 & P_{Elec,Operating} = P_{Elec,Full Load} (PLR)
 \end{aligned}$$

where:

$P_{Elec,Operating}$ = Actual (operating) electrical power output (W)

$Load$ = Electrical power output being requested by the Electric Load Center (W)

$P_{Ancillary}$ = Ancillary Power, user input (W)

PLR = Part-load ratio of the electric generator

PLR_{\max} = Maximum part-load ratio of the electric generator (i.e., the maximum value for the independent variable [PLR] defined in the Curve:Quadratic or Curve:Cubic object for the Electrical Efficiency Modifier Curve [function of part-load ratio])

PLR_{\min} = Minimum part-load ratio of the electric generator (i.e., the minimum value for the independent variable [PLR] defined in the Curve:Quadratic or Curve:Cubic object for the Electrical Efficiency Modifier Curve [function of part-load ratio])

The generator's electrical efficiency is then calculated based on the user-specified reference electrical efficiency (lower heating value [LHV] basis) and two electrical efficiency modifier curves.

$$ElecEfficiencyFTemp = b_1 + b_2(T_{a,i}) + b_3(T_{a,i})^2 \quad \text{or} \quad b_1 + b_2(T_{a,i}) + b_3(T_{a,i})^2 + b_4(T_{a,i})^3$$

$$ElecEfficiencyFPLR = c_1 + c_2(PLR) + c_3(PLR)^2 \quad \text{or} \quad c_1 + c_2(PLR) + c_3(PLR)^2 + c_4(PLR)^3$$

$$ElecEff_{Operating} = ElecEff_{Ref,LHV} (ElecEfficiencyFTemp)(ElecEfficiencyFPLR)$$

where:

$ElecEfficiencyFTemp$ = User-defined Electrical Efficiency Modifier Curve (function of temperature) evaluated at the current combustion air inlet temperature

$ElecEfficiencyFPLR$ = User-defined Electrical Efficiency Modifier Curve (function of part-load ratio) evaluated at the current operating part-load ratio

$ElecEff_{Operating}$ = Electrical efficiency at the current operating conditions

$ElecEff_{Ref,LHV}$ = Reference Electrical Efficiency (LHV [lower heating value] Basis), user input

The fuel energy consumption rate (LHV Basis) is then calculated as follows:

$$\dot{Q}_{Fuel,LHV} = \frac{P_{Elec,Operating}}{ElecEff_{Operating}}$$

where:

$\dot{Q}_{Fuel,LHV}$ = Fuel energy consumption rate, LHV basis (W)

If $ElecEff_{Operating}$ is equal to zero, then $P_{Operating}$ and $\dot{Q}_{Fuel,LHV}$ are set to zero. The fuel mass flow rate is then calculated.

$$\dot{m}_{fuel} = \frac{\dot{Q}_{Fuel,LHV}}{(LHV * 1000)}$$

where:

\dot{m}_{fuel} = Mass flow rate of fuel being consumed by the generator (kg/s), report variable "Generator <FuelType> Mass Flow Rate [kg/s]"

LHV = Fuel Lower Heating Value, user input (kJ/kg)

The ancillary power is calculated next using the user-specified ancillary power and ancillary power modifier curve. The ancillary power modifier curve is a quadratic function with the generator's fuel mass flow rate as the independent variable. If an ancillary power modifier curve is not specified in the input file, the modifier is assumed to be 1.0 and the ancillary power will be constant throughout the simulation.

$$AnciPowFMdotFuel = d_1 + d_2 (\dot{m}_{fuel}) + d_3 (\dot{m}_{fuel})^2$$

$$P_{Ancillary,Operating} = P_{Ancillary} (AnciPowFMdotFuel)$$

where:

$AnciPowFMdotFuel$ = User-defined Ancillary Power Modifier Curve (function of fuel input) evaluated at the actual fuel mass flow rate. This multiplier is assumed to be 1.0 if an ancillary power modifier curve name is not specified in the input.

$$P_{Ancillary} = \text{Ancillary power, user input (W)}$$

$P_{Ancillary,Operating}$ = Ancillary electric power at the current fuel mass flow rate (W), report variable “Generator Ancillary Electric Power [W]”.

If ancillary power is constant for the simulation (e.g., no modifier curve defined), then the calculations continue as described below. However, if an ancillary power modifier curve has been defined, then the calculations described above for $P_{ElecOperating}$, $ElecEff_{Operating}$, $\dot{Q}_{Fuel,LHV}$ and $P_{Ancillary,Operating}$ are recalculated in sequence until the solution converges.

The generator’s “net” electrical power output is calculated as the difference between the generator’s actual power output and the ancillary electric power as follows.

$$P_{Elec,Produced} = P_{Elec,Operating} - P_{Ancillary,Operating}$$

where:

$P_{Elec,Produced}$ = Generator net electric power output, report variable “Generator Electric Power Produced [W]”

The fuel energy consumption rate (higher heating value basis) for the generator is then calculated as follows:

$$\dot{Q}_{Fuel,HHV} = \dot{m}_{fuel} (HHV)(1000)$$

where:

$\dot{Q}_{Fuel,HHV}$ = fuel energy consumption rate (W), report variables “Generator <FuelType> Consumption Rate HHV Basis [W]” and “Generator Fuel Consumption Rate HHV Basis [W]”

HHV = Fuel Higher Heating Value, user input (kJ/kg)

Standby electrical power may also be modeled to simulate controls or other parasitics used by the generator. The standby power is calculated only when the generator is not operating (i.e., $Load$ from the Electric Load Center is zero). If the generator operates for a given timestep (i.e., $Load > 0.0$), the standby power is set equal to 0.

IF ($Load \leq 0.0$) *THEN*

$$P_{Standby} = P_{Standby,user\ input}$$

ELSE

$$P_{Standby} = 0.0$$

END IF

where:

$$P_{Standby, user\ input} = \text{Standby power, user input (W)}$$

$$P_{Standby} = \text{Report variable "Generator Standby Electric Power" (W)}$$

Report variables for electric energy produced, electric efficiency (LHV basis), fuel consumption (HHV basis), standby electric consumption and ancillary electric consumption are calculated as follows:

$$E_{Elec, Produced} = P_{Elec, Produced} (TimeStepSys)(3600)$$

$$ElecEff_{Operating, LHV} = \frac{P_{Elec, Produced}}{\dot{Q}_{Fuel, LHV}}$$

$$Q_{Fuel, HHV} = \dot{Q}_{Fuel, HHV} (TimeStepSys)(3600)$$

$$E_{Standby} = P_{Standby} (TimeStepSys)(3600)$$

$$E_{Ancillary} = P_{Ancillary, Operating} (TimeStepSys)(3600)$$

where:

$$E_{Elec, Produced} = \text{Report variable "Generator Electric Energy Produced [J]"}$$

$$ElecEff_{Operating, LHV} = \text{Report variable "Generator Electric Efficiency LHV Basis [-]"}$$

$$Q_{Fuel, HHV} = \text{Report variables "Generator <FuelType> Consumption HHV Basis [J]" and "Generator Fuel Consumption HHV Basis [J]"}$$

$$E_{Standby} = \text{Report variable "Generator Standby Electric Consumption [J]"}$$

$$E_{Ancillary} = \text{Report variable "Generator Ancillary Electric Consumption [J]"}$$

$$TimeStepSys = \text{HVAC system simulation time step (hr)}$$

In addition to calculating electric power production and fuel usage, the model is able to determine thermal power (heat recovery) output for heating water. For this case, the water flow rate through the heat recovery heat exchanger is established first. If the Heat Recovery Water Flow Operating Mode (user input) is set to Plant Control, then the Reference Heat Recovery Water Flow Rate (user input) is requested whenever the generator operates (constant value), but the actual flow rate may be restricted by other plant components (e.g., pump). If the Heat Recovery Water Flow Operating Mode is set to Internal Control, then the requested water flow when the generator operates is determined by the Reference Heat Recovery Water Flow Rate and a flow rate modifier curve.

IF (Plant Control) THEN

$$\dot{m}_w = \dot{V}_{w,Ref} (\rho_w)$$

ELSEIF (Internal Control) THEN

$$HeatRecFlowFTempPow = e_1 + e_2 (T_{w,i}) + e_3 (T_{w,i})^2 + e_4 (P_{net}) + e_5 (P_{net})^2 + e_6 (T_{w,i})(P_{net})$$

$$\dot{m}_w = \dot{V}_{w,Ref} (\rho_w) (HeatRecFlowFTempPow)$$

END IF

where:

\dot{m}_w = Report variable "Generator Heat Recovery Water Mass Flow Rate [kg/s]"

$\dot{V}_{w,Ref}$ = Reference Heat Recovery Water Flow Rate (m³/s), user input

ρ_w = Density of water (kg/m³) at 5.05°C

$HeatRecFlowFTempPow$ = User-defined Heat Recovery Water Flow Rate Modifier Curve (function of temperature and power) evaluated at the current inlet water temperature and net electrical power output. This multiplier is assumed to be 1.0 if a water flow rate modifier curve name is not specified in the input.

$T_{w,i}$ = Heat recovery inlet water temperature (°C), report variable "Generator Heat Recovery Inlet Temperature [C]"

P_{net} = Net electrical power output from the generator (W)

The methodology for determining thermal power (heat recovery to water) is similar to that used for calculating electric power production. The generator's steady-state thermal efficiency is calculated based on the user-specified reference thermal efficiency (LHV basis) and a thermal efficiency modifier curve.

$$ThermalEff_{SS} = ThermalEff_{Ref,LHV} (ThermalEffFTempElev)$$

$$ThermalEffFTempElev = f_1 + f_2 (T_{a,i}) + f_3 (T_{a,i})^2 + f_4 (Elev) + f_5 (Elev)^2 + f_6 (T_{a,i})(Elev)$$

where:

$ThermalEff_{SS}$ = Steady-state thermal efficiency at current conditions

$ThermalEff_{Ref,LHV}$ = Reference Thermal Efficiency (LHV Basis), user input

$ThermalEffFTempElev$ = User-defined Thermal Efficiency Modifier Curve (function of temperature and elevation) evaluated at the current combustion air inlet temperature and

elevation. This multiplier is assumed to be 1.0 if a thermal efficiency modifier curve name is not specified in the input.

The steady-state thermal power produced (heat recovery rate) is then calculated:

$$P_{Thermal,SS} = ThermalEff_{SS} \left(\dot{Q}_{Fuel,LHV} \right)$$

The actual (operating) thermal power is then calculated using the steady-state thermal power and three modifier curves:

$$P_{Thermal,Operating} = P_{Thermal,SS} (HeatRecRateFPLR)(HeatRecRateFTemp)(HeatRecRateFFlow)$$

$$HeatRecRateFPLR = g_1 + g_2 (PLR) + g_3 (PLR)^2 \quad -or- \\ g_1 + g_2 (PLR) + g_3 (PLR)^2 + g_4 (PLR)^3$$

$$HeatRecRateFTemp = h_1 + h_2 (T_{w,i}) + h_3 (T_{w,i})^2$$

$$HeatRecRateFFlow = i_1 + i_2 (\dot{m}_w) + i_3 (\dot{m}_w)^2$$

where:

$$P_{Thermal,Operating} = \text{Report variable "Generator Thermal Power Produced [W]"}$$

$HeatRecRateFPLR$ = User-defined Heat Recovery Rate Modifier Curve (function of part-load ratio) evaluated at the current operating part-load ratio. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

$HeatRecRateFTemp$ = User-defined Heat Recovery Rate Modifier Curve (function of inlet water temperature) evaluated at the current inlet water temperature. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

$HeatRecRateFFlow$ = User-defined Heat Recovery Rate Modifier Curve (function of water flow rate) evaluated at the current heat recovery water flow rate. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

The heat recovery output water temperature is then calculated.

$$T_{w,o} = T_{w,i} + \frac{P_{Thermal,Operating}}{(\dot{m}_w * Cp_w)}$$

where:

$T_{w,o}$ = Heat recovery outlet water temperature (°C), report variable "Generator Heat Recovery Outlet Temperature [C]"

Cp_w = Heat capacity of water (J/kg-K)

If the calculated heat recovery outlet water temperature exceeds to Maximum Heat Recovery Water Temperature (user input), then the outlet water temperature is reset to the maximum temperature (user input) and the thermal power is recalculated.

If combustion air inlet and outlet node names are specified in the input, along with exhaust air flow rate and exhaust air temperature information, then the model calculates the exhaust air conditions for each simulation time step. The exhaust air mass flow rate is first calculated based on the Reference Exhaust Air Mass Flow Rate, two modifier curves and an air density adjustment. Since fans are volumetric flow devices, the ratio of the air density at actual inlet air conditions to air density at reference inlet air conditions is used as an adjustment factor.

$$\dot{m}_{ExhAir} = \dot{m}_{ExhAir,Ref} (ExhFlowFTemp)(ExhFlowFPLR) \left(\frac{\rho_{a,i}}{\rho_{a,Ref}} \right)$$

$$ExhFlowFTemp = j_1 + j_2 (T_{a,i}) + j_3 (T_{a,i})^2 \quad -or- \\ j_1 + j_2 (T_{a,i}) + j_3 (T_{a,i})^2 + j_4 (T_{a,i})^3$$

$$ExhFlowFPLR = k_1 + k_2 (PLR) + k_3 (PLR)^2 \quad -or- \\ k_1 + k_2 (PLR) + k_3 (PLR)^2 + k_4 (PLR)^3$$

where:

$$\dot{m}_{ExhAir} = \text{Exhaust air mass flow rate (kg/s)}$$

$$\dot{m}_{ExhAir,Ref} = \text{Reference Exhaust Air Mass Flow Rate (kg/s), user input}$$

ExhFlowFTemp = User-defined Exhaust Air Flow Rate Modifier Curve (function of temperature) evaluated at the current combustion air inlet temperature. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

ExhFlowFPLR = User-defined Exhaust Air Flow Rate Rate Modifier Curve (function of part-load ratio) evaluated at the current operating part-load ratio. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

$$\rho_{a,i} = \text{Density of the combustion inlet air (kg/m}^3\text{)}$$

$$\rho_{a,Ref} = \text{Density of combustion inlet air at reference conditions (kg/m}^3\text{)}$$

In an analogous fashion, the exhaust air temperature is calculated using the Nominal (reference) Exhaust Air Outlet Temperature and two modifier curves.

$$T_{a,o} = T_{a,o,Nom} (ExhAirTempFTemp)(ExhAirTempFPLR)$$

$$ExhAirTempFTemp = l_1 + l_2 (T_{a,i}) + l_3 (T_{a,i})^2 \quad -or- \\ l_1 + l_2 (T_{a,i}) + l_3 (T_{a,i})^2 + l_4 (T_{a,i})^3$$

$$ExhAirTempFPLR = m_1 + m_2 (PLR) + m_3 (PLR)^2 \quad -or- \\ m_1 + m_2 (PLR) + m_3 (PLR)^2 + m_4 (PLR)^3$$

where:

$T_{a,o}$ = Exhaust air outlet temperature (°C)

$T_{a,o,Nom}$ = Nominal Exhaust Air Outlet Temperature (°C), user input

$ExhAirTempFTemp$ = User-defined Exhaust Air Temperature Modifier Curve (function of temperature) evaluated at the current combustion air inlet temperature. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

$ExhAirTempFPLR$ = User-defined Exhaust Air Flow Rate Rate Modifier Curve (function of part-load ratio) evaluated at the current operating part-load ratio. This multiplier is assumed to be 1.0 if a modifier curve name is not specified in the input.

The above calculations for exhaust air outlet temperature assume no heat recovery to water is being done. If thermal power (water heating) is being produced, then the exhaust air outlet temperature is recalculated as follows:

$$T_{a,o} = T_{a,o} - \frac{P_{Thermal,Operating}}{(\dot{m}_{ExhAir} * Cp_{air})}$$

where:

Cp_{air} = Heat capacity of air at the actual combustion air inlet conditions (J/kg-K)

The exhaust air outlet humidity ratio is also calculated.

$$w_{a,o} = w_{a,i} + \frac{[\dot{m}_{fuel} (HHV - LHV)(1000)/(h_{fg,16})]}{(\dot{m}_{ExhAir})}$$

where:

$w_{a,o}$ = Exhaust air outlet humidity ratio (kg/kg)

$w_{a,i}$ = Exhaust air inlet humidity ratio (kg/kg)

$h_{fg,16}$ = Enthalpy of vaporization of moisture at 16°C (J/kg)

The remaining report variables are calculated as follows.

$$E_{Thermal,Produced} = P_{Thermal,Operating} (TimeStepSys)(3600)$$

$$ThermalEff_{Operating,LHV} = \frac{P_{Thermal,Operating}}{\dot{Q}_{Fuel,LHV}}$$

where:

$$E_{Thermal,Produced} = \text{Report variable "Generator Thermal Energy Produced [J]"}$$

$$ThermalEff_{Operating,LHV} = \text{Report variable "Generator Thermal Efficiency LHV Basis [-]"}$$

Micro-Cogenerator

The input object Generator:MicroCHP provides a model that is a direct implementation of a model developed by IEA Annex 42 – The Simulation of Building-Integrated Fuel Cell and Other Cogeneration Systems (FC+COGEN-SIM). Annex 42 was formed as a working group within the International Energy Agency (IEA) program on Energy Conservation in Buildings and Community Systems (ECBCS). A full description of the model specification can be found in the report by Subtask B of FC+COGEN-SIM with the title “Specifications for Modelling Fuel Cell and Combustion-Based Residential Cogeneration Device within Whole-Building Simulation Programs.” The “Micro CHP” model in EnergyPlus is the one referred to as “A Generic Model for Combustion-based Residential Cogeneration Devices.”

The Micro CHP model is a straightforward empirical model with the exception that it is dynamic with respect to thermal heat recovery where performance is cast as a function of engine temperature. It is also dynamic with respect to possible warm up and cool down periods that may affect the ability of the generator to deliver the requested power. The relevant model equations are:

$$\eta_e = f(\dot{m}_{cw}, T_{cw,i}, P_{net,ss})$$

$$\eta_q = f(\dot{m}_{cw}, T_{cw,i}, P_{net,ss})$$

$$q_{gross} = P_{net,ss} / \eta_e$$

$$q_{gen,ss} = \eta_q q_{gross}$$

$$\dot{N}_{fuel} = q_{gross} / LHV_{fuel}$$

$$\dot{m}_{fuel}^{t+\Delta t} = \begin{cases} \dot{m}_{fuel,demand}^{t+\Delta t} & \text{if } d\dot{m}_{fuel} / dt \leq (d\dot{m}_{fuel} / dt)_{max} \\ \dot{m}_{fuel}^t \pm (d\dot{m}_{fuel} / dt)_{max} & \text{if } d\dot{m}_{fuel} / dt > (d\dot{m}_{fuel} / dt)_{max} \end{cases}$$

$$\dot{m}_{air} = f(P_{net,ss})$$

$$P_{net}^{t+\Delta t} = \begin{cases} P_{net,ss}^{t+\Delta t} & \text{if } dP_{net} / dt \leq (dP_{net} / dt)_{max} \\ P_{net}^t \pm (dP_{net} / dt)_{max} & \text{if } dP_{net} / dt > (dP_{net} / dt)_{max} \end{cases}$$

$$[MC]_{eng} \frac{dT_{eng}}{dt} = UA_{HX} (T_{cw,p} - T_{eng}) + UA_{loss} (T_{room} - T_{eng}) + q_{gen,ss}$$

$$[MC]_{cw} \frac{dT_{cw,o}}{dt} = [\dot{m}c_p]_{cw} (T_{cw,i} - T_{cw,o}) + UA_{HX} (T_{eng} - T_{cw,o})$$

where,

η_e is the steady-state, part load, electrical conversion efficiency of the engine (-)

η_q is the steady-state part load, thermal conversion efficiency of the engine (-)

\dot{m}_{cw} is the mass flow rate of plant fluid through the heat recovery section [kg/s]

$T_{cw,i}$ is the bulk temperature of the plant fluid entering the heat recovery section (°C)

$T_{cw,o}$ is the bulk temperature of the plant fluid leaving the heat recovery section (°C)

$P_{net,ss}$ is the steady-state electrical output of the system (W),

q_{gross} is the gross heat input into the engine (W),

$q_{gen,ss}$ is the steady-state rate of heat generation within the engine (W)

LHV_{fuel} is the lower heating value of the fuel used by the system (J/kg or J/kmol),

\dot{N}_{fuel} is the molar fuel flow rate (kmol/s)

\dot{m}_{fuel} is the mass fuel flow rate (kg/s)

\dot{m}_{air} is the mass flow rate of air thru the engine (kg/s)

$[MC]_{eng}$ is the thermal capacitance of the engine control volume (W/K)

T_{eng} is the temperature of the engine control volume (C)

UA_{HX} is the effective thermal conductance between the engine control volume and the cooling water control volume (W/K).

UA_{loss} is the effective thermal conductance between the engine control volume and the surrounding environment (W/K)

T_{room} is the air temperature of the surrounding environment (C)

$[MC]_{cw}$ is the thermal capacitance of the encapsulated cooling water and heat exchanger shell in immediate thermal contact (J/K)

$[\dot{m}c_p]_{cw}$ is the thermal capacity flow rate associated with the cooling water (W/K)

The functional forms for η_e and η_q are 2nd order trivariate polynomials with all of the cross terms.

EnergyPlus solves these for state values for the engine mass temperature, T_{eng} , and the outlet plant node, $T_{cw,o}$, in the following manner. The last two equations are interrelated but otherwise ordinary differential equations with the general form

$$\frac{dT}{dt} = a + bT$$

and have analytical solution

$$T = \left(T_o + \frac{a}{b}\right)e^{bt} - \frac{a}{b}$$

The engine temperature at the current timestep is calculated using

$$a = \frac{UA_{HX}}{[MC]_{eng}} * T_{cw,o} + \frac{UA_{loss}}{[MC]_{eng}} * T_{room} + \frac{q_{gen,ss}}{[MC]_{eng}}$$

$$b = -\left(\frac{UA_{HX}}{[MC]_{eng}} + \frac{UA_{loss}}{[MC]_{eng}}\right)$$

The plant node outlet fluid temperature (heat recovered) is solved using

$$a = \frac{[\dot{m}c_p]_{cw}}{[MC]_{cw}} * T_{cw,i} + \frac{UA_{HX}}{[MC]_{cw}} * T_{eng}$$

$$b = -\left(\frac{[\dot{m}c_p]_{cw}}{[MC]_{cw}} + \frac{UA_{HX}}{[MC]_{cw}}\right)$$

The interrelation of these two is handled by sequential substitution using an iteration scheme that alternates between calculations of T_{eng} and $T_{cw,o}$. The iteration loop exits once the energy is determined to be balanced using the following criteria:

Number of iterations > 3\

$$\frac{(q_{gen,ss})_{max}}{10000000} > UA_{HX} (T_{cw,o} - T_{eng}) + UA_{loss} (T_{room} - T_{eng}) + q_{gen,ss} - [MC]_{eng} \frac{dT_{eng}}{dt}$$

$$\frac{(q_{gen,ss})_{max}}{10000000} > [\dot{m}c_p]_{cw} (T_{cw,i} - T_{cw,o}) + UA_{HX} (T_{eng} - T_{cw,o}) - [MC]_{cw} \frac{dT_{cw,o}}{dt}$$

The Micro CHP model has a number of different operating modes. The operating mode for a given system timestep is determined from the mode during the previous timestep, user inputs, and high-level controls from elsewhere in EnergyPlus. The operating mode is reported for the state at the end of each timestep. The following table summarizes the various operating modes and the criteria for switching to a new mode for any given timestep. The EnergyPlus implementation adds the "Off" mode to the modes specified by Annex 42 which corresponds to the unit being scheduled to be unavailable. The difference between OFF and Standby modes determines whether or not standby power is consumed.

Operating mode	Main Criteria	Notes
----------------	---------------	-------

Off	Availability schedule value = 0	No consumption of power or fuel.
Stand By	Availability schedule value \neq 0	Consumes stand by power but no fuel
Warm Up	Load (thermal or electric) > 0.0 Availability schedule value \neq 0 Time Delay < elapsed time since entering warm up mode Engine temp < nominal engine temp	Two alternate sub-modes: Stirling Engines use warm up by “nominal engine temperature” while Internal Combustion Engines use “time delay” Fuel is consumed but no power is produced
Normal Operation	Load (thermal or electric) > 0.0 Availability schedule value \neq 0 Time Delay > elapsed time since entering warm up mode Engine temp \geq nominal temp	Fuel is consumed and power is produced
Cool Down	Load (thermal or electric) = 0.0 Availability schedule value \neq 0	Two alternate sub-modes where engine can be forced to go thru a complete cool down cycle before allowed to go back into warm up or normal mode. No fuel is consumed and no power is produced.

For timesteps where the generator switches from warm up mode to normal mode in the middle of the timestep, part load ration values are calculated for the portion of the time step that the generator is in normal operation.

The engine and heat recovery thermal conditions are modeled for all modes so, for example, an engine that is off but still warm could provide some hot water recovery.

The engine model can use an arbitray fuel mixture that is defined by the user – see the entry for Generator:FuelSupply.

References

Kelly, N. and A. Ferguson. 2007. A Generic Model Specification for Combustion-based Residential Cogeneration Devices. In *Specifications for Modelling Fuel Cell and Combustion-Based Residential Cogeneration Device within Whole-Building Simulation Programs*. I. Beausoleil-Morrison and N. Kelly editors. Draft report of Annex 42 of the International Energy Agency ECBCS.

Fuel Cell Cogenerator

The Generator:FuelCell input objects provides a model which is a direct implementation of a model developed by IEA Annex 42 – The Simulation of Building-Integrated Fuel Cell and Other Cogeneration Systems (FC+COGEN-SIM). Annex 42 was formed as a working group within the International Energy Agency (IEA) program on Energy Conservation in Buildings and Community Systems (ECBCS). A full description of the model specification can be found in the report by Subtask B of FC+COGEN-SIM with the title “Specifications for Modelling Fuel Cell and Combustion-Based Residential Cogeneration Device within Whole-Building Simulation Programs.” The “Specifications for Modelling Fuel Cell Cogeneration Devices within Whole-Building Simulation Programs.”

The Annex 42 Fuel Cell model is characterized as a “grey box” empirical model where a mixture of thermodynamic principles and empirical performance maps are used to model the cogeneration performance of a fairly complex device with many individual subsystems. In EnergyPlus, the individual subsystems are separate into individual input objects such as

Generator:FuelCell:PowerModule or
 Generator:FuelCell:ExhaustGasToWaterHeatExchanger. The resulting model is relatively complex requiring on the order of one hundred inputs. The model is not for the faint of heart; this model is far more involved than most component models in building simulation. This stems from the fact that fuel cell cogenerators are complicated devices that interact with the built environment in a number of ways. Fuel cells could drawn in gas/fuel, air, and water with as many as six separate streams. In addition to electricity and heated water, they also give off heat in the form of convection and radiation and exhaust air out of the zone. The devices may take a long time to start up and include storage to follow loads rather than attempt to vary the power the fuel cell. The fuel cell model allows examining system level interactions over annual timeframes that include all the important interactions with a building's energy and comfort systems.

The Annex 42 fuel cell model is described more thoroughly in the references (see below). Here we provide a summary of the relevant model equations which are taken from the Annex 42 model specification. The first equation is the main energy balance for the fuel cell power module (includes the fuel reformation and fuel cell stacks). This energy balance is used to model the enthalpy of the product gases that leave the fuel cell power module.

$$\begin{aligned} & \sum_i \left(\dot{N}_i \cdot \left[\hat{h}_i - \Delta_f \hat{h}_i^o \right] \right)_{fuel} + \sum_i \left(\dot{N}_i \cdot \left[\hat{h}_i - \Delta_f \hat{h}_i^o \right] \right)_{air} + \dot{N}_{liq-water} \cdot \left(\left[\hat{h} - \Delta_f \hat{h}^o \right]_{H_2O,liq} - \Delta_f \hat{h}_{H_2O,fg}^o \right) \\ & + \dot{H}_{dilution-air-in} + \dot{N}_{fuel} \cdot LHV_{fuel} + P_{el,ancillaries-AC} \\ = & P_{el} + \sum_i \left(\dot{N}_i \cdot \left[\hat{h}_i - \Delta_f \hat{h}_i^o \right] \right)_{FCPM-cg} + q_{s-cool} + q_{skin-loss} + \dot{H}_{dilution-air-out} \end{aligned}$$

The remaining equations describe various terms and the balance of systems. The electrical efficiency is modeled using:

$$\varepsilon_{el} = \left[\varepsilon_0 + \varepsilon_1 \cdot P_{el} + \varepsilon_2 \cdot P_{el}^2 \right] \cdot \left[1 - N_{stops} \cdot D \right] \cdot \left[1 - (MAX(\int dt - t_{threshold}, 0.0)) \cdot L \right]$$

$$P_{blower-el} = b_0 + b_1 \cdot \dot{N}_{air} + b_2 \cdot \dot{N}_{air}^2 + b_3 \cdot \dot{N}_{air}^3$$

In several places the model is formulated to offer different options. For example, the flow rate of process air can be described either as a function of electrical power produced or the fuel flow rate.

$$\dot{N}_{air} = \left[a_0 + a_1 \cdot P_{el} + a_2 \cdot P_{el}^2 \right] \cdot \left[1 + a_3 \cdot T_{air} \right]$$

or

$$\dot{N}_{air} = \left[a_0 + a_1 \cdot \dot{N}_{fuel} + a_2 \cdot \dot{N}_{fuel}^2 \right] \cdot \left[1 + a_3 \cdot T_{air} \right]$$

$$\dot{N}_{liq-water} = w_0 + w_1 \cdot \dot{N}_{fuel} + w_2 \cdot \dot{N}_{fuel}^2$$

$$P_{pump-el} = p_0 + p_1 \cdot \dot{N}_{water} + p_2 \cdot \dot{N}_{water}^2 + p_3 \cdot \dot{N}_{water}^3$$

$$P_{comp-el} = c_0 + c_1 \cdot \dot{N}_{fuel} + c_2 \cdot \dot{N}_{fuel}^2 + c_3 \cdot \dot{N}_{fuel}^3$$

$$P_{el,ancillaries-AC} = anc_0 + anc_1 \cdot \dot{N}_{fuel}$$

$$P_{el,aux-ancillaries} = x_0 + x_1 \cdot \dot{N}_{aux-fuel}$$

$$q_{HX} = \varepsilon_{HX} \cdot (\dot{N} \hat{c}_p)_{\min} \cdot (T_{aux-mix} - T_{water,in})$$

$$q_{HX} = (UA)_{eff} \cdot \frac{(T_{aux-mix} - T_{water,out}) - (T_{HX-exh} - T_{water,in})}{\ln \left(\frac{T_{aux-mix} - T_{water,out}}{T_{HX-exh} - T_{water,in}} \right)}$$

$$(UA)_{eff} = hx_{s,0} + hx_{s,1} \cdot \dot{N}_{water} + hx_{s,2} \cdot \dot{N}_{water}^2 + hx_{s,3} \cdot \dot{N}_{aux-mix} + hx_{s,4} \cdot \dot{N}_{aux-mix}^2$$

or

$$(UA)_{eff} = \left[\frac{1}{(hA)_{gas}} + \frac{1}{(hA)_{water}} + F_{HX} \right]^{-1}$$

where,

F_{HX} is an adjustment factor,

$$h_{gas} = h_{gas}^0 \cdot \left(\frac{\dot{N}_{gas}}{\dot{N}_{gas}^0} \right)^n,$$

$$h_{water} = h_{water}^0 \cdot \left(\frac{\dot{N}_{water}}{\dot{N}_{water}^0} \right)^m$$

$$q_{HX} = (UA)_{eff} \cdot \frac{(T_{aux-mix} - T_{water,out}) - (T_{HX-exh} - T_{water,in})}{\ln \left(\frac{T_{aux-mix} - T_{water,out}}{T_{HX-exh} - T_{water,in}} \right)} + \dot{N}_{H_2O-cond} \cdot \hat{h}_{fg}$$

$$\dot{N}_{H_2O-cond} = (T_{cond-threshold} - T_{water,in}) \cdot \left[hx_{l,1} \cdot \left(\frac{\dot{N}_{H_2O}}{\dot{N}_{aux-mix}} \right) + hx_{l,2} \cdot \left(\frac{\dot{N}_{H_2O}}{\dot{N}_{aux-mix}} \right)^2 \right]$$

$$q_{aux-skin-losses} = (UA)_{aux} \cdot (T_{aux-mix} - T_{room})$$

$$q_{skin-loss} = s_0 + s_1 \cdot \dot{N}_{fuel} + s_2 \cdot \dot{N}_{fuel}^2$$

$$Q_{battery}^{t+\Delta t} = Q_{battery}^t + P_{battery-charge} \cdot \varepsilon_{charge} \cdot \Delta t$$

$$\eta_{PCU} = u_0 + u_1 \cdot P_{PCU-in} + u_2 \cdot P_{PCU-in}^2$$

$$q_{s-cool} = \left[r_0 + r_1 (T_{stack} - T_{stack}^o) \right] \cdot \left[1 + r_2 P_{el} + r_3 P_{el}^2 \right]$$

$$(UA)_{s-cogen} = \left[\frac{1}{(hA)_{s-cogen}} + F_{s-cogen} \right]^{-1}$$

$$h_{s-cogen} = h_{s-cogen}^0 \cdot \left(\frac{\dot{N}_{s-cogen}}{\dot{N}_{s-cogen}^0} \right)^{n_s}$$

$$P_{s-air-el} = f_0 + f_1 \cdot q_{s-air} + f_2 \cdot q_{s-air}^2$$

The Annex 42 fuel cell was implemented directly in EnergyPlus. A sequential substitution method is used to handle all the interactions between the different subsystems. The main energy balance drawn for the fuel cell power module is rearranged to put all the terms on the right hand side. The enthalpy of the product gas stream is determined from this energy balance. The Shomate equation is used to evaluate the enthalpy and specific heat of the various streams. The EnergyPlus implementation evaluates fluid properties using the average temperature of inlet and outlet streams whereas the Annex 42 specification often uses just the inlet temperature. The Shomate equation is inverted using the regula falsi numerical method available within EnergyPlus to calculate the temperature of the product gases from their enthalpy.

References

Beausoleil-Morrison, I., A. Schatz, and F. Marechal. 2006. A model for simulating the thermal and electrical production of small-scale solid-oxide fuel cell cogeneration systems within building simulation programs. *HVAC & R Research*. Amer. Soc. Heating, Ref. Air-Conditioning Eng. Inc. Atlanta, GA.

Beausoleil-Morrison, I., A. Weber, F. Marechal, and B. Griffith. 2007. Specifications for Modelling Fuel Cell Cogeneration Devices within Whole-Building Simulation Programs. In *Specifications for Modelling Fuel Cell and Combustion-Based Residential Cogeneration Device within Whole-Building Simulation Programs*. I. Beausoleil-Morrison and N. Kelly editors. **Draft** report of Annex 42 of the International Energy Agency ECBCS.

Custom Fuel Supply for Generators

The Generator:FuelSupply input object in EnergyPlus implements a fairly comprehensive capability to calculate properties of fuel mixtures from a description of the molar composition of all the constituents. The fuel supply modeling is based on the specifications prepared by IEA Annex 42 for their generator models. This modeling capability allows taking into consideration the exact gas composition of local natural gas service. Or the user can explore the implications of an various alternative fuels such as alcohols or biogas. An unlimited number of possible mixtures can be analyzed.

Gas phase thermochemistry calculations and data are programmed into EnergyPlus to handle the set of constituents listed in the table below. The relevant properties of each fuel constituent, i , are calculated as a function of temperature using the Shomate equation:

$$\hat{h}_i - \Delta_f \hat{h}_i^o = A \cdot \left(\frac{T}{1000} \right) + \frac{B}{2} \cdot \left(\frac{T}{1000} \right)^2 + \frac{C}{3} \cdot \left(\frac{T}{1000} \right)^3 + \frac{D}{4} \cdot \left(\frac{T}{1000} \right)^4 - \frac{E}{\left(\frac{T}{1000} \right)} + F - H$$

where,

\hat{h}_i is the enthalpy (J/kmol)

$\Delta_f \hat{h}_i^o$ is the molar enthalpy at the standard state (J/kmol)

T is the temperature of the gas (K)

A, B, C, D, E, F, H are the coefficients for the Shomate equation.

The lower heating value (LHV) of a fuel mixture is calculated from the molar fractions using:

$$LHV_{fuel} = \sum_i (\chi_i \cdot LHV_i)$$

Where,

$$LHV_i = \left[\Delta_f \hat{h}_{C_xH_y}^o - x \cdot \Delta_f \hat{h}_{CO_2}^o - \frac{y}{2} \cdot \Delta_f \hat{h}_{H_2O}^o \right]$$

x is the number of carbon atoms

y is the number of hydrogen atoms

Similarly, the higher heating value (HHV) of the fuel mixture is calculated using:

$$HHV_{fuel} = \sum_i (\chi_i \cdot HHV_i)$$

Where,

$$HHV_i = \left[\Delta_f \hat{h}_{C_xH_y}^o - x \cdot \Delta_f \hat{h}_{CO_2}^o - \frac{y}{2} \cdot \Delta_f \hat{h}_{H_2O}^o + \frac{y}{2} \cdot (\Delta_f \hat{h}_{H_2O}^o - H_{liq}) \right]$$

The Shomate coefficients used in EnergyPlus are listed in the table below. Data source "NIST" indicates the data were directly from Chemistry WebBook. Data source "CHEMKIN" indicates the data were developed by curve fitting library data for the CHEMKIN commercial program (which uses the Gorden-McBride polynomial rather than the Shomate formulation).

Constituent	A	B	C	D	E	F	H	Source
N ₂	26.092	8.218801	-1.976141	0.159274	0.044434	-7.98923	0.0	NIST
O ₂	29.659	6.137261	-1.186521	0.09578	-0.219663	-9.861391	0.0	NIST
Ar	20.786	2.8259E-7	-1.4642E-7	1.0921E-8	-3.6614E-8	-6.19735	0.0	NIST
CO ₂	24.99735	55.18696	-33.69137	7.948387	-0.136638	-403.6075	-393.5224	NIST
H ₂ O(gas)	29.0373	10.2573	2.81048	-0.95914	0.11725	-250.569	-241.8264	CHEMKIN
H ₂ O(liq)	-203.606	1523.29	-3196.413	2474.455	3.85533	-256.5478	-285.8304	NIST
H ₂	33.066178	-11.363417	11.432816	-2.772874	-0.158558	-9.9808	0.0	NIST

CH ₄	-0.703029	108.4773	-42.52157	5.862788	0.678565	-76.84376	-74.8731	NIST
C ₂ H ₆	-3.03849	199.202	-84.9812	11.0348	0.30348	-90.0633	-83.8605	CHEMKIN
C ₃ H ₈	-23.1747	363.742	-222.981	56.253	0.61164	-109.206	-103.855	CHEMKIN
C ₄ H ₁₀	-5.24343	426.442	-257.955	66.535	-0.26994	-149.365	-133.218	CHEMKIN
C ₅ H ₁₂	-34.9431	576.777	-338.353	76.8232	1.00948	-155.348	-146.348	CHEMKIN
C ₆ H ₁₄	-46.7786	711.187	-438.39	103.784	1.23887	-176.813	-166.966	CHEMKIN
CH ₃ OH	14.1952	97.7218	-9.73279	-12.8461	0.15819	-209.037	-201.102	CHEMKIN
C ₂ H ₅ OH	-8.87256	282.389	-178.85	46.3528	0.48364	-241.239	-234.441	CHEMKIN

References

Beausoleil-Morrison, I., A. Weber, F. Marechal, and B. Griffith. 2007. Specifications for Modelling Fuel Cell Cogeneration Devices within Whole-Building Simulation Programs. In *Specifications for Modelling Fuel Cell and Combustion-Based Residential Cogeneration Device within Whole-Building Simulation Programs*. I. Beausoleil-Morrison and N. Kelly editors. Report of Annex 42 of the International Energy Agency ECBCS.

NIST. 2003. Chemistry WebBook, National Institute of Standards and Technology Standard Reference Database Number 69, March 2003 Release, <http://webbook.nist.gov/chemistry/>.

Gordon S. and B.J. McBride. 1971. Computer program for calculation of complex chemical equilibrium composition, rocket performance, incident and reflected shocks and Chapman-Jouguet detonations. NASA SP-273.

Photovoltaic Arrays

The Generator:Photovoltaic input object provides three different models referred to as "Simple", "Equivalent One-Diode" and "Sandia" and the choice will determine the mathematical models (and input data) used to determine the energy produced by solar/electric conversion panels. The EnergyPlus photovoltaic array models are called one at a time at the HVAC system timestep along with other Electrical Generation components such as gas turbines and diesel engines.

All of the photovoltaic models share the same models for predicting incident solar radiation that are also used for the solar thermal calculations and are described in the section Climate, Sky and Solar/Shading Calculations.

Note that some of the terminology used to discussed photovoltaics overlaps with terminology used to discuss Fortran programs. The word *module* may refer to a PV panel or to a fortran90 programming entity. *Model* may refer to a manufacturers production model for a specific type of PV module or to a mathematical model used for engineering analysis. *Array* may refer to a collection of PV modules wired together or to a mathematical variable with multiple elements.

Simple Model

The PhotovoltaicPerformance:Simple object describes about the simplest model for predicting photovoltaic energy production. In this model the user specifies the efficiency with which surfaces convert incident solar radiation to electricity. (In the other models this efficiency is determined as part of the model.) The full geometric model for solar radiation is used, including sky models, shading, and reflections, to determine the incident solar resource. The model accepts arbitrary conversion efficiencies and does not require actual production units be tested to obtain empirical performance coefficients. (The Energy+.idd sets the range of conversion efficiencies to be on [0..1], but the user could alter the Energy+.idd to extend this range if desired.)

Mathematical Description

Table 61. Nomenclature for Simple Photovoltaic model

Mathematical variable	Description
P	Electrical power produced by photovoltaics [W]
A_{surf}	Net area of surface [m^2]
f_{activ}	Fraction of surface area with active solar cells []
G_T	Total solar radiation incident on PV array [W/m^2]
η_{cell}	Module conversion efficiency []
η_{invert}	DC to AC conversion efficiency []

The usable electrical power produced by a PV surface are calculated using:

$$P = A_{surf} \cdot f_{activ} \cdot G_T \cdot \eta_{cell} \cdot \eta_{invert} \quad (714)$$

On the right hand side of this equation, only G_T is calculated by EnergyPlus and the rest are user inputs. Power levels are assumed constant over the timestep to arrive at energy production.

There are two modes that can be selected by the user that govern how the PV system is coupled to the building surfaces. If the integration mode is selected as 'DECOUPLED' then no adjustments are made to account for energy extracted in the form of electricity. If the integration mode is selected as 'INTEGRATED' then the energy extracted in the form of electricity is removed from surface heat transfer calculations using a sink term. This sink term is lagged from the previous timestep.

Equivalent One-Diode Model

The PhotovoltaicPerformance:EquivalentOne-Diode input objects provides another model for predicting the electrical performance of a photovoltaic (PV) array. This model is also known as the "TRNSYS PV" or "Five-Parameter" model.

Mathematically speaking, the EnergyPlus PV module employs equations for an empirical equivalent circuit model to predict the current-voltage characteristics of a single module. This circuit consists of a DC current source, diode, and either one or two resistors. The strength of the current source is dependent on solar radiation and the IV characteristics of the diode are temperature-dependent. The results for a single module equivalent circuit are extrapolated to predict the performance of a multi-module array.

The module employs a "five-parameter" equivalent circuit to model crystalline (both mono and poly) PV modules developed at the University of Wisconsin – Madison [2]. The values of these parameters cannot normally be obtained directly from manufacturers' catalogs. However, the PV module will automatically calculate them from commonly available data (except for shunt resistance). The PV module also includes an optional incidence angle modifier correlation to calculate how the reflectance of the PV module surface varies with the angle of incidence of solar radiation.

The module determines PV current as a function of load voltage. Other OUTPUTS include current and voltage at the maximum power point along the IV curve, open-circuit voltage, short circuit current as well as electrical load met and unmet.

Table 62. General Nomenclature for the PV model

Mathematical variable	Description
β	Slope of PV array [degrees]
γ	Empirical PV curve-fitting parameter
ε_{γ}	Semiconductor bandgap [eV]
η_c	Module conversion efficiency
$\mu_{I_{sc}}$	Temperature coefficient of short-circuit current [A/K]
$\mu_{V_{oc}}$	Temperature coefficient of open-circuit voltage [V/K]
θ	Angle of incidence for solar radiation [degrees]
$\tau\alpha$	Module transmittance-absorptance product
$\tau\alpha_{normal}$	Module transmittance-absorptance product at normal incidence
G_T	Total radiation incident on PV array
$G_{T,beam}$	Beam component of incident radiation
$G_{T,diff}$	Diffuse component of incident radiation
$G_{T,gnd}$	Ground-reflected component of incident radiation
$G_{T,NOCT}$	Incident radiation at NOCT conditions
$G_{T,ref}$	Incident radiation at reference conditions
I	Current
I_L	Module photocurrent
$I_{L,ref}$	Module photocurrent at reference conditions
I_o	Diode reverse saturation current
$I_{o,ref}$	Diode reverse saturation current at reference conditions
I_{sc}	Short-circuit current
$I_{sc,ref}$	Short-circuit current at reference conditions
I_{mp}	Current at maximum power point along IV curve
$I_{mp,ref}$	Current at maximum power point along IV curve, reference conditions
IAM	Dimensionless incidence angle modifier
K	Boltzmann constant [J/K]
NP	Number of modules in parallel in array
NS	Number of modules in series in array
N_s	Number of individual cells in module
P	PV output power
P_{max}	PV output power at maximum power point along IV curve
Q	Electron charge constant
R_s	Module series resistance [Ω]
R_{sh}	Module shunt resistance [Ω]
T_c	Module temperature [K]

$T_{c,NOCT}$	Module temperature at NOCT conditions [K]
$T_{c,ref}$	Module temperature at reference conditions [K]
U_L	Array thermal loss coefficient
V	Voltage
V_{mp}	Voltage at maximum power point along IV curve
$V_{mp,ref}$	Voltage at maximum power point along IV curve, reference conditions
V_{oc}	Open-circuit voltage
$V_{oc,ref}$	Open-circuit voltage at reference conditions [V]

Mathematical Description

PV Section 1: Four-Parameter Model

The model description begins with the four-parameter equivalent circuit model that was developed largely by Townsend [1989] and is detailed by Duffie and Beckman [1991]. The model was first incorporated into a component for the TRNSYS simulation package by Eckstein [1990]. The EnergyPlus module employs the Eckstein model for crystalline PV modules, using it whenever the short-circuit IV slope is set to zero or a positive value as modified by Ulleberg [2000]. The four parameter model assumes that the slope of the IV curve is zero at the short-circuit condition:

$$\left(\frac{dI}{dV} \right)_{v=0} = 0 \quad (715)$$

This is a reasonable approximation for crystalline modules. The “four parameters” in the model are $I_{L,ref}$, $I_{o,ref}$, γ , and R_s . These are empirical values that cannot be determined directly through physical measurement. The EnergyPlus model calculates these values from manufactures’ catalog data as discussed in the following section on calculating these parameters

The four-parameter equivalent circuit is shown in the following figure:

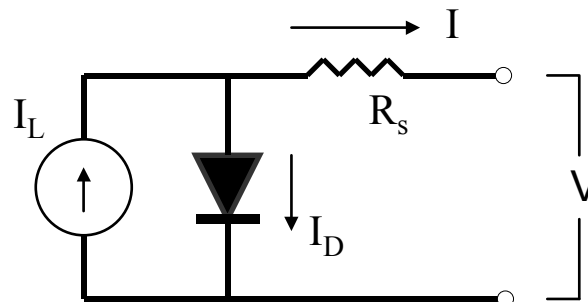


Figure 241. Equivalent Circuit in the Four Parameter PV Model

V is the load voltage and I is the current flowing through the load and PV.

Determining Performance under Operating Conditions

The IV characteristics of a PV change with both insolation and temperature. The PV model employs these environmental conditions along with the four module constants $I_{L,ref}$, $I_{o,ref}$, γ , and R_s to generate an IV curve at each timestep.

The current-voltage equation of circuit shown in the previous figure is as follows:

$$I = I_L - I_o \left[\exp \left(\frac{q}{\gamma k T_c} (V + I R_s) \right) - 1 \right] \quad (716)$$

R_s and γ are constants. The photocurrent I_L depends linearly on incident radiation:

$$I_L = I_{L,ref} \frac{G_T}{G_{T,ref}} \quad (717)$$

The reference insolation G_{ref} is nearly always defined as 1000 W/m². The diode reverse saturation current I_o is a temperature dependent quantity:

$$\frac{I_o}{I_{o,ref}} = \left(\frac{T_c}{T_{c,ref}} \right)^3 \quad (718)$$

Equation (716) gives the current implicitly as a function of voltage. Once I_o and I_L are found from Eqs. (717)3 and (718)4, Newton's method is employed to calculate the PV current. In addition, an iterative search routine finds the current (I_{mp}) and voltage (V_{mp}) at the point of maximum power along the IV curve.

Calculating $I_{L,ref}$, $I_{o,ref}$, γ , and R_s

The input specification for the PV model include several values which must be read from manufacturers' PV module catalogs. The manufactures' values are used to determine the equivalent circuit characteristics $I_{L,ref}$, $I_{o,ref}$, γ , and R_s . These characteristics define an equivalent circuit that is employed to find the PV performance at each timestep, as described previously. This section describes the algebra and calculation algorithms used to solve for the four equivalent circuit characteristics.

Three of these values, $I_{L,ref}$, $I_{o,ref}$, γ , may be isolated algebraically. The first step is to substitute the current and voltage into Eq. (716) at the open-circuit, short circuit, and maximum power conditions:

$$0 = I_{L,ref} - I_{o,ref} \left[\exp \left(\frac{q}{\gamma k T_{c,ref}} V_{oc,ref} \right) - 1 \right] - \frac{V_{oc,ref}}{R_{sh}} \quad (719)$$

$$I_{sc,ref} = I_{L,ref} - I_{o,ref} \left[\exp \left(\frac{q I_{sc,ref} R_s}{\gamma k T_{c,ref}} \right) - 1 \right] - \frac{I_{sc,ref} R_s}{R_{sh}} \quad (720)$$

$$I_{mp,ref} = I_{L,ref} - I_{o,ref} \left[\exp \left(\frac{q}{\gamma k T_{c,ref}} (V_{mp,ref} + I_{mp,ref} R_s) \right) - 1 \right] - \frac{V_{mp,ref} + I_{mp,ref} R_s}{R_{sh}} \quad (721)$$

In each case the "-1" term is may be dropped to simplify the algebra. This approximation has little influence on the right side of the equations since because the magnitude of I_o is very small, generally on the order of 10⁻⁶ A. Some rearrangement then yields the following three expressions which isolate $I_{L,ref}$, $I_{o,ref}$, γ :

$$I_{L,ref} \approx I_{sc,ref} \quad (722)$$

$$\gamma = \frac{q(V_{mp,ref} - V_{oc,ref} + I_{mp,ref}R_s)}{kT_{c,ref} \ln\left(1 - \frac{I_{mp,ref}}{I_{sc,ref}}\right)} \quad (723)$$

$$I_{o,ref} = \frac{I_{sc,ref}}{\exp\left(\frac{qV_{oc,ref}}{\gamma kT_{c,ref}}\right)} \quad (724)$$

At this point an additional equation is needed in order to determine the last unknown parameter. Taking the analytical derivative of voltage with respect to temperature at the reference open-circuit condition derives the fourth equation. This analytical value is matched to the open-circuit temperature coefficient, a catalog specification:

$$\frac{\partial V_{oc}}{\partial T_c} = \mu_{voc} = \frac{\gamma k}{q} \left[\ln\left(\frac{I_{sc,ref}}{I_{o,ref}}\right) + \frac{T_c \mu_{isc}}{I_{sc,ref}} - \left(3 + \frac{q\mathcal{E}}{AkT_{c,ref}}\right) \right] \quad (725)$$

where

$$A = \frac{\gamma}{N_s}$$

The “TRNSYS PV model” uses an iterative search routine in these four equations to calculate the equivalent circuit characteristics. The first step is to set upper and lower bounds for the series resistance parameter R_s : physical constraints require the R_s value to lie between 0 and the value such that $\gamma = N_s$. The initial guess for R_s is midway between these bounds. γ and $I_{o,ref}$ are found from Eq. (723) and Eq. (724), while Eq. (722) gives a trivial solution for $I_{L,ref}$. The model then employs Eq. (725) to compare the analytical and catalog values for μ_{voc} . When all other variables are held constant, the analytical value for μ_{voc} increases monotonically with series resistance (Townsend 1989). If the analytical voltage coefficient is less than the catalog value, the lower bound for R_s is reset to the present guess value. Likewise, the upper bound is set to the current value if the calculated μ_{voc} is too large. After resetting the upper or lower bound for R_s , a new guess value is found by averaging the bounds. This procedure repeats until R_s and γ converge. Note that for $I_{L,ref}$, $I_{o,ref}$, γ , and R_s are assumed to be constant and are calculated only on the first call in the simulation. Alternatively, the user may enter a known series resistance by entering a **positive** value in the IDF. In this case the iterative routine described above is skipped and Eqs. (722), (723), and (724) find $I_{L,ref}$, $I_{o,ref}$, and γ directly from the given value of R_s .

The four-parameter model described up to this point does not adequately describe the current-voltage characteristics of amorphous silicon or thin-film PV modules. There is one major qualitative difference between the IV curves of crystalline and amorphous PVs. The short-circuit slope of the IV curve for crystalline modules is very close to zero, while the slope for amorphous modules is generally finite and negative. The five parameter model is used to model this by adding a shunt resistance to the equivalent circuit used in the four-parameter model. The four-parameter model may be considered a special case of the five-parameter model in which the shunt resistance approach infinity. Fry (1999) has shown that the negative reciprocal of the short-circuit IV slope closely approximates the shunt resistance:

$$R_{sh} \approx \frac{-1}{\left(\frac{dI}{dV}\right)_{V=0}} \quad (726)$$

PV Section 2 : Module Operating Temperature

The PV model uses one of five methods for determining cell temperature data. The cell temperature of a PV module is important because the hotter the temperature of the panel, the lower its electrical output. The cell temperature calculation method is chosen by the user in the EnergyPlus IDF file through a parameter choice in the IDD entry called Integration and Cell Temperature Mode.

If the value of this parameter is “**Decoupled**” then the cell temperature of the PV is modeled using the method from the Duffie and Beckman (1991) for estimating cell temperature. This is based upon the standard NOCT (Nominal Operating Cell Temperature) measurements to compute the module temperature T_c at each timestep. The NOCT temperature ($T_{c,NOCT}$) is the operating temperature of the module with a wind speed of 1 m/s, no electrical load, and a certain specified insolation and ambient temperature [Beckman and Duffie, 1991]. The values for insolation $G_T,NOCT$ and ambient temperature $T_{a,NOCT}$ are usually 800 W/m² and 20° C. η_c is the conversion efficiency of the module, which varies with ambient conditions. $\tau\alpha$ is a user-defined constant.

The equation is:

$$T_c = T_a + \frac{\left(1 - \frac{\eta_c}{\tau\alpha}\right)}{\left(\frac{G_T \tau\alpha}{U_L}\right)} \quad (727)$$

If the user specifies the “**DecoupledUllebergDynamic**” mode for calculating cell temperature, then a method developed by Ulleberg is used:

$$T_{cell}|_t = T_{ambient} + \left(T_{cell}|_{t-1} - T_{ambient}\right) * e^{\frac{-UL}{Cap} \Delta t} \quad (728)$$

In other words, the cell temperature is a function of the previous cell temperature and the thermal capacity of the PV module material.

If the user specifies “**IntegratedSurfaceOutsideFace**” for this parameter, then the temperature result from EnergyPlus’s modeling of surfaces is used for the cell temperature. Also the energy exported from the surface as electricity becomes a sink in the internal source modeling for the heat transfer surface.

If the user specifies “**IntegratedTranspiredCollector**” for this parameter, then the temperature result for the unglazed transpired collector surfaces is used for the cell temperature. Also the energy exported from the collector surface as electricity is deprecated using a source term in the collector’s temperature modeling.

If the user specifies “**IntegratedExteriorVentedCavity**” for this parameter, then the temperature result for the exterior cavity is used for the cell temperature. Also the energy exported from the baffle surface as electricity is deprecated using a source term in the baffle’s temperature modeling.

PV Section 3 : Multi-Array Modules

The electrical calculations discussed in the sections above deal only with a single module. The EnergyPlus PV component may be used to simulate arrays with any number of modules.

The IDF defines the number of modules in series (NS) and modules in parallel (NP) for the entire array. The total number of modules in the array is the product of NS and NP. When simulating a single module only, both NS and NP are set to 1. The single-module values for all currents and voltages discussed in PV Section 1 are multiplied by NP or NS to find values for the entire array. This approach neglects module mismatch losses.

With the above equations, and the assumption that the panels operate at the maximum power point, it is a direct calculation to determine DC power production. The performance of an array of identical modules is assumed to be linear with the number of modules in series and parallel. The inverter efficiency is applied linearly to derate the energy production. The inverter capacity forms a limit for power production from a PV generator. A 'load' is passed the PV array acting as a generator and various trivial calculations compare PV production to this load. If the PV array is associated with a surface that is associated with a zone, then if the zone has any multipliers associated with it, electricity production will be multiplied accordingly.

References

Duffie, John A. and William A. Beckman. 1991. *Solar Engineering of Thermal Processes*. New York: John Wiley & Sons, Inc..

Eckstein, Jürgen Helmut. 1990. *Detailed Modeling of Photovoltaic Components*. M. S. Thesis – Solar Energy Laboratory, University of Wisconsin, Madison: 1990.

Ulleberg, Øystein. *HYDROGEMS Component Library for TRNSYS 15 User Manual*, Institute for Energy Technology, Kjeller, Norway

TRNSYS Type 94 documentation. TRNSYS Reference Manual. University of Wisconsin, Madison, WI.

Sandia Photovoltaic Performance Model

The input object PhotovoltaicPerformance:Sandia provides a third model for predicting the electricity generated by photovoltaics and is referred to as the Sandia model. This model is based on work done at Sandia National Lab, Albuquerque, NM by David King -- with the help of many others. The model consists of a series of empirical relationships with coefficients that are derived from actual testing. Once the coefficients for a particular module are available, it is straightforward matter to use the model equations to calculate five select points on the current-voltage curve.

The implementation in EnergyPlus is also based on work done by Greg Barker (2003) for the National Renewable Energy Lab who implemented the Sandia model in FORTRAN77 as a custom type (Type101) for the TRNSYS computer program.

There are several climate and solar orientation inputs to the model that are managed elsewhere in EnergyPlus including: incident beam solar, incident diffuse solar, incidence angle of beam solar, solar zenith Angle, outdoor drybulb, wind speed, and elevation.

Mathematical Description

This section presents the mathematical description of the Sandia model from a draft report by King et, al. (2003). The core of the model predicts the performance of a single PV module. The following nomenclature and equations summarize the Sandia model.

Table 63. Nomenclature for Sandia PV model

Mathematical variable	Description
I_{sc}	Short-circuit current (A)
I_{mp}	Current at the maximum-power point (A)
I_x	Current at module $V = 0.5 V_{oc}$, defines 4th point on I-V curve

I_{xx}	Current at module $V = 0.5 (V_{oc} + V_{mp})$, defines a 5th point on the I-V curve
V_{oc}	Open-circuit voltage (V)
V_{mp}	Voltage at maximum-power point (V)
P_{mp}	Power at maximum-power point (W)
f_d	Fraction of diffuse irradiance used by module
N_s	Number of cells in series in a module's cell-string
N_p	Number of cell-strings in parallel in module
k	Boltzmann's constant, 1.38066E-23 (J/k)
q	Elementary charge, 1.60218E-19 (coulomb)
T_c	Cell temperature inside module ($^{\circ}\text{C}$)
$\delta(T_c)$	'Thermal voltage' per cell at temperature T_c , approximately 1 volt for a typical 26-cell crystalline silicon module
E_e	'Effective' solar irradiance
E_b	Beam solar irradiance
E_{diff}	Diffuse solar irradiance
C_0, C_1	Empirical coefficients relating I_{mp} to E_e , $C_0 + C_1 = 1$ (both dimensionless)
C_2, C_3	Empirical coefficients relating V_{mp} to E_e (C_2 dimensionless, C_3 is $1/V$)
C_4, C_5	Empirical coefficients relating I_x to E_e , $C_4 + C_5 = 1$ (both dimensionless)
C_6, C_7	Empirical coefficients relating I_{xx} to E_e , $C_6 + C_7 = 1$ (both dimensionless)
n	Empirically determined 'diode factor' for individual cells
AMa	Absolute Air Mas
AOI	Solar angle-of-incidence (degrees) from normal
$f_1(AM_a)$	Empirical polynomial function used to relate short-circuit current to the solar spectrum via air mass
$f_2(AOI)$	Empirical polynomial function used to relate short-circuit current to the solar angle-of-incidence
a_0, a_1, a_2, a_3, a_4	Empirical coefficients for $f_1(AM_a)$ polynomial
$b_0, b_1, b_2, b_3, b_4, b_5, b_6$	Empirical coefficients for $f_1(AOI)$ polynomial
T_o	Reference cell temperature for rating, typically fixed at 25°C
I_{sco}	Short circuit current at reference conditions
I_{mpo}	Max power point current at reference conditions
V_{mpo}	Voltage at max power at reference conditions
V_{oco}	Open circuit voltage at reference conditions
I_{xo}	Current at $V = 0.5 V_{oc}$ and at reference conditions
I_{xxo}	Current at $V = 0.5 (V_{mp} + V_{oc})$ and at reference conditions
α_{Isc}	Normalized temperature coefficient for I_{sc} ($1/^{\circ}\text{C}$)

$\alpha_{I_{mp}}$	Normalized temperature coefficient for I_{mp} ($1/^\circ\text{C}$)
$\beta_{V_{oc}}(E_e)$	Temperature coefficient for module open-circuit-voltage as function of E_e
$\beta_{V_{oco}}$	Temperature coefficient for module open-circuit-voltage at reference conditions
$m\beta_{V_{oco}}$	Coefficient for irradiance dependence of open-circuit-voltage-temperature coefficient, often zero ($V/^\circ\text{C}$)
$\beta_{V_{mp}}(E_e)$	Temperature coefficient for module maximum-power-voltage as a function of E_e
$\beta_{V_{mpo}}$	Temperature coefficient for module maximum-power-voltage at reference conditions
$m\beta_{V_{mpo}}$	Coefficient for irradiance dependence of maximum-power-voltage-temperature coefficient, often zero ($V/^\circ\text{C}$)
T_m	PV module temperature at back surface ($^\circ\text{C}$)
T_a	Ambient outdoor drybulb temperature ($^\circ\text{C}$)
E	Solar irradiance incident on module surface (W/m^2)
WS	Wind speed at standard 10-m height (m/s)
a	Empirical coefficient relating module temperature at low wind and high solar irradiance
b	Empirical coefficient relating module temperature decrease with increasing wind speed
T_c	Temperature of solar cell inside module ($^\circ\text{C}$)
E_o	Reference solar irradiance ($1000 \text{ W}/\text{m}^2$)
ΔT	Temperature difference between T_c and T_m at E_o ($^\circ\text{C}$), (This is $d(T_c)$ in Sandia database)

The current implementation in EnergyPlus focuses on determining performance at the maximum power-point but also calculates, and reports, four other points on the I-V curve so that the data are available for analyses outside of EnergyPlus. The equations below use the module performance parameters that are available in a database provided by Sandia National Laboratory (see www.sandia.gov/pv). The following equations form the basis of the Sandia model implemented in EnergyPlus:

$$I_{sc} = I_{sco} \cdot f_1(AM_a) \cdot \left\{ \left(E_b \cdot f_2(AOI) + f_d \cdot E_{diff} \right) / E_o \right\} \cdot \left\{ 1 + \alpha_{isc} \cdot (T_c - T_o) \right\}$$

$$I_{mp} = I_{mpo} \cdot \left\{ C_o \cdot E_e + C_1 \cdot E_e^2 \right\} \cdot \left\{ 1 + \alpha_{imp} \cdot (T_c - T_o) \right\}$$

$$V_{oc} = V_{oco} + N_s \cdot \delta(T_c) \cdot \ln(E_e) + \beta_{V_{oc}}(E_e) \cdot (T_c - T_o)$$

$$V_{mp} = V_{mpo} + C_2 \cdot N_s \cdot \delta(T_c) \cdot \ln(E_e) + C_3 \cdot N_s \cdot \left\{ \delta(T_c) \cdot \ln(E_e) \right\}^2 + \beta_{V_{mp}}(E_e) \cdot (T_c - T_o)$$

$$P_{mp} = I_{mp} \cdot V_{mp}$$

$$I_x = I_{xo} \cdot \{C_4 \cdot E_e + C_5 \cdot E_e^2\} \cdot \{1 + (\alpha_{isc}) \cdot (T_c - T_o)\}$$

$$I_{xx} = I_{xvo} \cdot \{C_6 \cdot E_e + C_7 \cdot E_e^2\} \cdot \{1 + (\alpha_{imp}) \cdot (T_c - T_o)\}$$

where,

$$E_e = I_{sc} / [I_{sco} \cdot \{1 + \alpha_{isc} \cdot (T_c - T_o)\}]$$

$$\delta(T_c) = n \cdot k \cdot (T_c + 273.15) / q$$

$$f_1(AM_a) = a_0 + a_1 AM_a + a_2 (AM_a)^2 + a_3 (AM_a)^3 + a_4 (AM_a)^4$$

$$f_2(AOI) = b_0 + b_1 \cdot AOI + b_2 (AOI)^2 + b_3 (AOI)^3 + b_4 (AOI)^4 + b_5 (AOI)^5$$

$$\beta_{voc}(E_e) = \beta_{voco} + m_{\beta voc} \cdot (1 - E_e)$$

$$\beta_{vmp}(E_e) = \beta_{vmpo} + m_{\beta vmp} \cdot (1 - E_e)$$

$$T_m = E \cdot \{e^{a+b \cdot WS}\} + T_a$$

$$T_c = T_m + \frac{E}{E_o} \cdot \Delta T$$

With the above equations, and the assumption that the panels operate at the maximum power point, it is a direct calculation to determine DC power production. The performance of an array of identical modules is assumed to be linear with the number of modules in series and parallel. The inverter efficiency is applied linearly to derate the energy production. The inverter capacity forms a limit for power production from a PV generator. A 'load' is passed the PV array acting as a generator and various trivial calculations compare PV production to this load. If the PV array is associated with a surface that is associated with a zone, then if the zone has any multipliers associated with it, electricity production will be multiplied accordingly.

The equation above for T_m is used to predict back-of-module temperature when the mode 'Decoupled' is selected. This would be appropriate for most rack mounted PV installations. If the user selects 'IntegratedSurfaceOutsideFace' then the back-of-module temperature is obtained from the outside face surface temperature calculated by the full complement of Heat Balance models using Conduction Transfer Functions that is native to EnergyPlus. And energy exported from the surface is accounted for using a source/sink term that is lagged from the previous timestep (pingpong).

References

- King, D.L., Boyson, W.E., Kratochvil J.A. 2003. Photovoltaic Array Performance Model. Sandia National Laboratories, Albuquerque, NM 87185, November 2003 currently in DRAFT
- Barker, G. 2003 (unknown). Predicting Long-Term Performance of Photovoltaic Arrays. Submitted as deliverable for NREL subcontract LAX-1-30480-02. Currently in DRAFT

Davis, M.W., Fanney, A.H., and Dougherty B.P. 2002. Measured Versus Predicted Performance of Building Integrated Photovoltaics. from the conference Solar 2002, Sunrise on the Reliable Energy Economy, June 15-19, 2002, Reno, NV. Available from NIST website.

King, D.L. 1996. Photovoltaic Module and Array Performance Characterization Methods for All System Operating Conditions. Sandia National Laboratory. Albuquerque, NM 87185

Economics Calculations

Component Costs

EnergyPlus provides simple cost estimating capabilities as an aid to design analysis and for life cycle costs. There are three broad steps involved. The first involves determining *construction* costs by summing individual “line items.” The second involves determining *project* costs by adjusting construction costs to account for things like profit and design fees. The third involves comparing the current simulation to a reference case so that marginal increases can be calculated. The reference documentation contained in this section pertains to the following input object names.

- **ComponentCost:LineItem**
- **ComponentCost:Adjustments**
- **ComponentCost:Reference**

Line Item Costs

Line item cost calculations are generally trivial involving simple multiplication and summation. This section documents details of how line item costs are calculated. The program code consists mainly of a **Case** construct where the **Line Item Type** is used to control the details of how calculations are performed.

The overall philosophy is to provide methods of calculating items using either direct entry of needed data (**General** using object type), or using component descriptive data entered elsewhere in the input file (e.g. **Lights**), or by using quantities that are calculated by the program during the simulation (e.g. **Coil:DX** and **Chiller:Electric**).

The rest of this section provides details by organized by the type of line item.

General

The line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

Construction

This line item type is called “Construction” but is used to estimate the costs of surfaces in the model. The Construction attribute of Surface objects is useful for categorizing surfaces. The number of units, N , is determined by summing the Area, A , of Surface objects that have the type of construction specified in the **Item Name** field. Surfaces are screened to eliminate any duplicates that may exist for interior partitions creating a list of 1 to m unique surfaces. If a surface is associated with a Zone, then zone multiplier, M_z , and list multipliers, M_G , are applied (these are usually defaulted to 1).

$$N = \sum_1^m (A * M_z * M_G)$$

The line item subtotal, L , is calculated by multiplying the number of units (actually units of area here), N (m^2) by the Cost-per-Area, P_a , ($$/m^2$):

$$L = N * P_a$$

Coil:DX and Coil:Cooling:DX:SingleSpeed

DX coil costs can be estimated in one of three ways: per-each, per-kW, per-kW-COP. The program determines which method to use based on there being non-zero values in appropriate input fields.

If **cost per each** is greater than 0.0 then the number of units, N , is the number of cooling coils. This will be 1 if the **Item Name** is the name of a specific coil described elsewhere. If the name is set to the wildcard (*) then this will equal the total number of DX:Coils in the model. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

If **cost per kilowatt** is greater than 0.0 then the number of units, N , is the number of kilowatts of total, rated, cooling capacity. This will be based on all the DX:Coils in the model if **Item Name** is set to the wildcard (*) and will be that of the named coil if set to a valid coil name. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt, P_{kW} :

$$L = N * P_{kW}$$

If **Cost per kilowatt per coefficient-of-performance** is greater than 0.0 then the number of units, N , is the number of kilowatts of total, rated, cooling capacity multiplied by the nominal coefficient of performance (COP) of the DX:Coils. This will be based on all the DX:Coils in the model if **Item Name** is set to the wildcard (*) and will be that of the named coil if set to a valid coil name. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt-per-COP, P_{kW-COP} :

$$L = N * P_{kW-COP}$$

Coil:Heating:Gas

Gas-fired heating coil costs can be estimated in one of three ways: per-each, per-kW, per-kW-COP. The program determines which method to use based on there being non-zero values in appropriate input fields.

If **cost per each** is greater than 0.0 then the number of units, N , is the number of heaters. This will be 1 if the **Item Name** is the name of a specific heater described elsewhere. If the name is set to the wildcard (*) then this will equal the total number of Coil:Heating:Gas objects in the model. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

If **Cost per kilowatt** is greater than 0.0 then the number of units, N , is the number of kilowatts of total, rated, heating capacity. This will be based on all the Coil:Heating:Gas objects in the model if **Item Name** is set to the wildcard (*) and will be that of the named coil if set to a valid coil name. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt, P_{kW} :

$$L = N * P_{kW}$$

If **Cost per kilowatt per coefficient-of-performance** is greater than 0.0 then the number of units, N , is the number of kilowatts of total, rated, heating capacity multiplied by the theoretical efficiency of the heater(s). (Here we are treating gas heating efficiency as a Coefficient of Performance (COP)). This will be based on all the Coil:Heating:Gas objects in the model if **Item Name** is set to the wildcard (*) and will be that of the named coil if set to a

valid coil name. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt-per-COP, P_{kW-COP} :

$$L = N * P_{kW-COP}$$

Chiller:Electric

Electric chiller costs can be estimated in one of three ways: per-each, per-kW, per-kW-COP. The program determines which method to use based on there being non-zero values in appropriate input fields.

If **cost per each** is greater than 0.0 then the number of units, N , is the number of chillers. This will be 1 if the **Item Name** is the name of a specific coil described elsewhere. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

If **Cost per kilowatt** is greater than 0.0 then the number of units, N , is the number of kilowatts of total, rated, cooling capacity for the specified chiller. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt, P_{kW} :

$$L = N * P_{kW}$$

If **Cost per kilowatt per coefficient-of-performance** is greater than 0.0, then the number of units, N , is the number of kilowatts of total, rated, cooling capacity multiplied by the nominal coefficient of performance (COP) of the chiller. This will be based on the named chiller (if set to a valid coil name). Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt-per-COP, P_{kW-COP} :

$$L = N * P_{kW-COP}$$

Daylighting:Controls

The costs of controllers for harvesting daylight are determined by the number of reference points. The cost for each controller, P_e , are input. The of units, N , is determined from the number of daylight reference points in all the zones if the **Item Name** is the wildcard (*). If **Item Name** is set to a valid Zone name then N is the number of daylight reference points in just that zone (zones can have more than one daylight controllers). Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

Shading:Zone:Detailed

Shading surfaces need to be handled separately because they do not have the **Construction** attribute. The **Item Name** must be set to a valid name for a **Shading:Zone:Detailed** object defined elsewhere in the file. The number of units, N , is determined from the area of the named surface multiplied by zone multiplier, M_z , and list multipliers, M_g :

$$N = A * M_z * M_g$$

The line item subtotal, L , is calculated by multiplying the number of units, N (m^2) by the Cost-per-Area, P_a , ($$/m²):$

$$L = N * P_a$$

Lights

The cost of electric lighting systems can be modeled in two ways: per-each and per-kW. The program determines which method to use based on there being non-zero values in appropriate input fields. The **Item Name** must be the name of a valid Zone defined elsewhere in the input.

If **cost per each** is greater than 0.0, then the number of units, N , is the number lighting systems in the zone and is assumed to be 1. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-Each, P_e :

$$L = N * P_e$$

If **Cost per kilowatt** is greater than 0.0 then the number of units, N , is the number of kilowatts in the design level for electric lighting systems defined in Lights objects associated with the zone. The **Item Name** must be the name of a valid Zone defined elsewhere in the input. N is then the sum of all the Lights associated with the named Zone. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt, P_{kW} :

$$L = N * P_{kW}$$

Generator:Photovoltaic

The costs of simple photovoltaic power systems can be modeled using cost per kilowatt. The number of units, N , is the nominal rated peak power of the photovoltaic panels. The photovoltaic generator must be modeled using the PhotovoltaicPerformance:Simple input object which is a very simplistic model and does not include input for the nominal rated peak power of the system. Therefore a peak power is calculated using the usual 1000 W/m² of incident solar radiation, G_p , multiplied by the active solar cell area, A and the efficiency, E , and converted to units of kilowatts.

$$N = \frac{G_p * A * E}{1000.0}$$

Where, the cell area A is calculated by multiplying the area of the surface associated with the simple photovoltaic system and the active area fraction defined in the PhotovoltaicPerformance:Simple. Then the line item subtotal, L , is calculated by multiplying the number of units, N , by the Cost-per-kilowatt, P_{kW} :

$$L = N * P_{kW}$$

Adjustments

Various adjustments are available to model total project costs from the component costs modeled using individual line items. These adjustments are provided to allow propagating how changes in component costs are amplified by things like design fees, profit, bonding, and taxes.

The subtotal of individual line items, S_L , is obtained by summing all line item subtotals, L :

$$S_L = \sum L$$

For the reference building, S_L is user input (typically from a prior run of baseline building model).

The subtotal for miscellaneous construction costs (typically those costs not explicitly included as line items), S_m , are calculated by multiplying the **Miscellaneous Cost Model** (per Square Meter), C_m , by the total conditioned floor area, A_c :

$$S_m = A_c * C_m$$

The subtotal for the amount that the construction costs should be altered because of regional differences in material and labor costs (e.g. when using national average data), S_r , is determined by multiplying the Regional Adjustment Factor, R_f , by the sum of S_L and S_m :

$$S_r = (S_L + S_m)R_f$$

Remaining adjustments are applied to the subtotal for construction costs, S_c , which is the sum of S_L , S_m , and S_r :

$$S_c = S_L + S_m + S_r$$

The **Design and Engineering Fee Fraction**, F_d , is the fraction of construction costs, S_c , attributable to costs associated with architectural and engineering services needed for the project. The subtotal for costs associated with these fees, S_D , are determined by multiplying S_c by F_d :

$$S_D = S_c * F_D$$

The **Contractor Fee Fraction**, F_c , is the fraction of construction costs, S_c , attributable to costs associated with the contractor's profit that should be included in the project. The subtotal for costs associated with contracting, S_p , are determined by multiplying S_c by F_c :

$$S_p = S_c * F_C$$

The **Contingency Fraction**, F_s , is the fraction of construction costs, S_c , which should be included in a cost estimate to provide for contingencies (perhaps related to errors and uncertainty in the estimate and project). The subtotal for contingency costs, S_S , are determined by multiplying S_c by F_S :

$$S_S = S_c * F_S$$

The **Permits, Bonding, Insurance Fraction**, F_B , is the fraction of construction costs, S_c , which should be included in a cost estimate to provide for things like taxes, bonding, insurance, and permits. The subtotal for these extra costs, S_B , are determined by multiplying S_c by F_B :

$$S_B = S_c * F_B$$

The **Commissioning Fee Fraction**, F_{CX} , is the fraction of construction costs, S_c , which should be included in a cost estimate to provide for commissioning the building systems. The subtotal for these extra costs, S_{CX} , are determined by multiplying S_c by F_{CX} :

$$S_{CX} = S_c * F_{CX}$$

Finally, the cost estimate total, T , is obtained by summing the various subtotals:

$$T = S_L + S_m + S_r + S_D + S_p + S_S + S_B + S_{CX}$$

The normalized total cost estimate, C , is calculated by dividing T by the total conditioned floor area, A_C :

$$C = \frac{T}{A_C}$$

Comparisons

The capability of comparing the current cost estimate to that of a reference building is provided because it is common to consider the marginal increases in costs associated with applying different energy design measures. EnergyPlus calculates and reports the difference between the current model and a reference case for all the subtotals and totals discussed above. The reported differences are the reference values subtracted from the current value.

Tariff Computation

The EnergyPlus economic (Utility Costs) objects related to computing utility bills include:

- UtilityCost:Tariff
- UtilityCost:Qualify
- UtilityCost:Charge:Simple
- UtilityCost:Charge:Block
- UtilityCost:Ratchet
- UtilityCost:Variable
- UtilityCost:Computation

This section builds upon the discussion that appears in the Input Output Reference under the heading “EnergyPlus Economics.” The actual computation of monthly utility bills is not difficult since it mostly consists of multiplying energy consumptions or demands by the price on a per unit basis and adding different bill components. The implementation in EnergyPlus becomes more complex since the objects were crafted to allow a great deal of flexibility in specifying a utility tariff while, at the same time, being as simple as possible.

The following discussion on variables and hierarchies is based on the text that appears in the Input Output Reference.

Conceptual Framework – Variables and Hierarchy

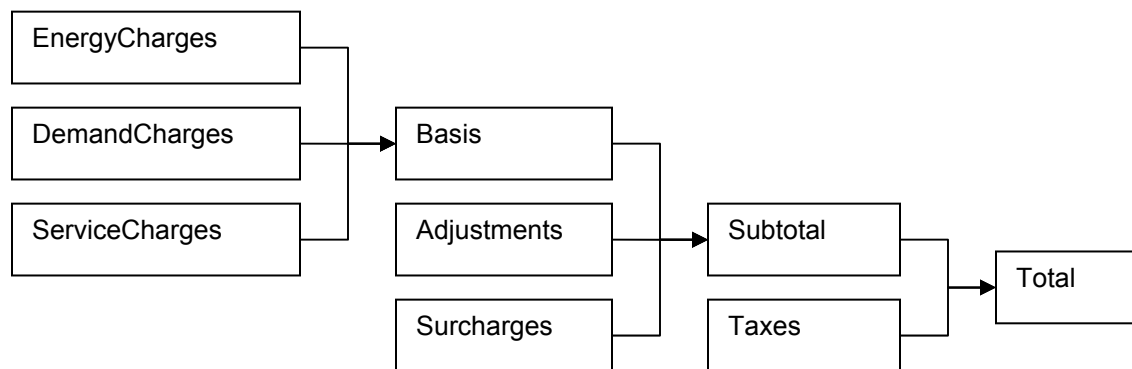
To understand how to use the utility bill calculation portion of EnergyPlus you first need to understand some important concepts of variables and hierarchy. A variable, for the purposes of this section, is simply a named holder of a series of numbers. In most cases, the variable will be a named holder of 12 numbers, one number for each monthly utility bill. Here is a visualization of a variable called Electric Energy Use:

Month	Electric Energy Use
January	12143
February	13454
March	14178
April	14876
May	15343
June	16172
July	16105
August	15762
September	14543
October	13987
November	13287
December	12403

If you have ever done any computer programming, you can think of a variable as an array. Many of the names used in the utility bill calculation portion of EnergyPlus are names of variables. In the case of the UtilityCost:Charge objects, the name of the object is also used as a name of a variable.

In many of today's utility rates, the charges for energy or demand are broken into distribution and supply charges. To allow for this, more than one charge may be defined for a particular category. The variables assigned to the same category are added together.

The categories are combined in following hierarchy:



Any charges included in the EnergyCharges category are added together. The EnergyCharges, DemandCharges and ServiceCharges are added together to form the Basis. The Basis, Adjustments and Surcharges are added together to form the Subtotal. The Subtotal and Taxes are added together to be the Total. The total represents the total monthly charges on that tariff for the energy source used. The combining of categories together is performed automatically unless the user specifies the UtilityCost:Computation. In addition, each category, which is also a variable, may be used as a source. For example, a tax that is 5% of the subtotal would be shown as:

```

UtilityCost:Charge:Simple,
  TaxOfFivePercent,      ! Charge Variable Name
  TariffExample1,        ! Tariff Name
  Subtotal,              ! Source Variable
  Annual,                ! Season
  Taxes,                 ! Category Variable Name
  0.05;                  ! Cost Per Unit Value (or Variable)
  
```

As you can see, the UtilityCost:Charge:Simple and UtilityCost:Charge:Block objects do most of the "work" of computing the annual energy cost. The benefit of using this categorization is

that totals of each category are shown in the output reports and it organizes the charges in the monthly calculations in a logical way that fits almost all tariffs. If no categorization is desired, theoretically, all charges could be assigned to the Total category. The categories themselves are simply variable names. Charges may also be assigned to the “NotIncluded” category if the result of the charge is used as an intermediate calculation and should not be included in the Total.

The objects that create variables are:

- UtilityCost:Qualify
- UtilityCost:Charge:Simple
- UtilityCost:Charge:Block
- UtilityCost:Ratchet
- UtilityCost:Variable

Default Order of Computation

The user has the option of two different ways to determine the order of computation. If an UtilityCost:Computation object is specified for the tariff, the sequence specified in that object is used for computing the various variables. If no UtilityCost:Computation object is specified, a sequence of computational steps is automatically derived and shown as part of the report. The routine that creates this automatic sequence of computation steps is called CreateDefaultComputation as part of the EconomicTariff module.

The order in which the computation should be made is complicated by the fact that the objects can each have variables that are inputs and others that are outputs. Since any of the variables can be used as inputs, we must ensure that they are computed prior to being used. In other words, because the objects allow a great deal of flexibility, there is no simple default order that the computations should be made.

Luckily there are known algorithms for sorting through these types of interdependencies. In fact, the method that spreadsheets use for sorting through the dependencies of cell formulas referencing other cells with formula is very similar. In addition, linkers (used as part of the computer language compiling process) face similar issues of sorting through dependences. Figuring out the optimal path in a complex project represented by a PERT Chart also uses a similar algorithm.

Generically, dependency problems are usually characterized as Directed Acycle Graphs (DAGs). A DAG shows the individual formulas as circles and uses arrows between the circles to show which formula is dependent on which other formulas. One of the simplest explanations can be seen at the following website:

<http://www.cee.hw.ac.uk/~alison/alg/lectures.html> (Click on lecture 14 “Critical Path”).

The lecture and site was created by Alison Cawsey, Department of Computing and Electrical Engineering, Heriot-Watt University Edinburgh EH14 4AS, UK. The specific algorithm that was used in EnergyPlus is described at this site and is quoted below:

Calculate, for each node, the in-degree of that node (ie, how many edges end up there). Store these in array D.

Repeat:

– Remove (output) node such that $D[n]=0$.

– Decrement $D[x]$ for all nodes x that are neighbors of n (edge from n to x).

Of course in this case “node” has nothing to do with EnergyPlus nodes but is just describing one of the formulas in a DAG. This is just one of several different methods to solve a DAG.

The general method for solving a DAG is called a topological sort. The algorithm used in EnergyPlus is one of the simplest methods available and is appropriate given the number of dependencies. More efficient algorithms are known but are probably only appropriate for much larger number of dependencies.

One important note, if after the algorithm is exercised, and some of the formulas still have a count on the number of dependencies, it must be the result of a circular dependency and an error condition is flagged in the ERR file.

The objects have specific variables that are used as inputs and outputs, and thus the outputs are dependent on the inputs, are shown in the following table:

Object Variables Inputs and Outputs

<i>Object</i>	<i>Outputs</i>	<i>Inputs</i>
Qualify	Name	Source Threshold
Charge:Simple	Name Category	Source Cost Per Unit
Charge:Block	Name Category Remaining	Source Block Size Multiplier Block Size Block Cost
Ratchet	Name	Baseline Adjustment Multiplier Offset

In addition, the hierarchy shown in the first diagram in this section also represents dependencies that are included when determining the order of computation.

The resulting order of computation is shown at the bottom of the economics report.

Computation Steps

Once the order that the formulas should be computed is known, the actual evaluation of the formulas is based on a simple Last In First Out (LIFO) stack. This is a common method to compute expressions where values are stored on the stack and operands work off of the top of the stack.

Special Modules/Reporting

Environmental Impacts

Typically when a new building technology is evaluated the energy performance of a baseline building is compared to the energy and life-cycle costs of alternatives to determine cost-effectiveness. But what if the lowest energy or life-cycle cost alternative is not the cleanest or lowest environmental impact? By calculating environmental impact, designers can compare alternatives not only in terms of their energy performance but also their environmental performance—working towards a more sustainable design (Liesen 1997; Stroot, Nemeth, and Fournier 1996). Environmental impacts are quantified, in part, by modeling the amount of emissions and in EnergyPlus this is done using the input objects “EnvironmentalImpactFactors,” “FuelFactors,” and “Output:EnvironmentalImpactFactors.”

Based on emissions factors entered by the user, EnergyPlus calculates the mass or volume of thirteen different pollutants: CO₂ (carbon dioxide), CO (carbon monoxide), CH₄ (methane), NO_x (nitrogen oxides), N₂O (nitrous oxide), SO₂ (sulphur dioxide), PM (particulate matter), PM₁₀ (particulate matter 10>PM₁₀>2.5 microns), PM_{2.5} (particulate matter<2.5 microns), NH₃ (ammonia), NMVOC (non-methane volatile organic compounds), Hg (mercury), and Pb (lead) as well as water consumed through evaporation in thermo- and hydro-electric generation and high- and low-level nuclear waste from nuclear electricity generation for on- and off-site energy production. Note that while these comprise the largest proportion of pollutants, more than one hundred other pollutants are emitted from fossil fuel combustion or electricity generation. Much of the information compiled here for fossil fuel combustion comes from *AP-42 Compilation of Air Pollutant Emission Factors* (EPA 1998a, 1998b, 1996). For more information on pollutants, see the U.S. Environmental Protection Agency (EPA) Clearinghouse for Inventories & Emission Factors (www.epa.gov/ttn/chief/efinformation.html).

EnergyPlus models energy performance of on-site fossil fuels and purchased electricity (generated from a variety of fuels including natural gas, oil, gasoline, diesel, coal, hydroelectric, nuclear, wind, solar power, and biomass). The energy performance calculated by EnergyPlus is converted into a mass or volume of pollutants emitted. From a baseline building, alternative energy and pollution saving technologies can be explored, and the energy savings and pollution reduction can be calculated. Figure 242 and Figure 243 illustrate a comparison of two buildings simulated using Chicago weather data in EnergyPlus and the calculated pollutant levels (based on U.S. national average pollutants) (Crawley 2003).

To calculate the mass or volume of each pollutant, consumption is multiplied by an emissions factor for each fuel (natural gas, electricity, fuel oil, diesel, or coal). In future versions, users will be able to schedule how the emissions factors by time of day, month, season and year. For electricity, the mix of generation fuel sources—whether utility, state or regional—is used to adjust the emission factors. If a user has emissions factors specific to the building site and equipment, these can be entered directly. **There are no default emissions factors.**

Types of Pollutants

EPA categorizes pollutants as either Criteria Pollutants or Hazardous Pollutants. Criteria pollutants are the six substances for which EPA has set health-based standards, including carbon monoxide (CO), nitrogen oxides (NO_x), sulfur dioxide (SO₂), and particulate matter (PM₁₀ and PM_{2.5}), ozone (O₃), and lead (Pb). Because ozone is created in atmospheric photochemical reactions of volatile organic compounds, ammonia, and other substances rather than direct building-related energy emissions, we do not calculate ozone emissions in EnergyPlus. But we do include ozone precursors: methane (CH₄), non-methane volatile organic compounds (NMVOC), and ammonia (NH₃). Hazardous pollutants are substances that are known or suspected to cause serious health problems such as cancer. We include typical hazardous substances associated with energy production and use including lead (Pb)

and mercury (Hg). We also include CO₂ (carbon dioxide) since it is largest greenhouse gas in terms of impact.

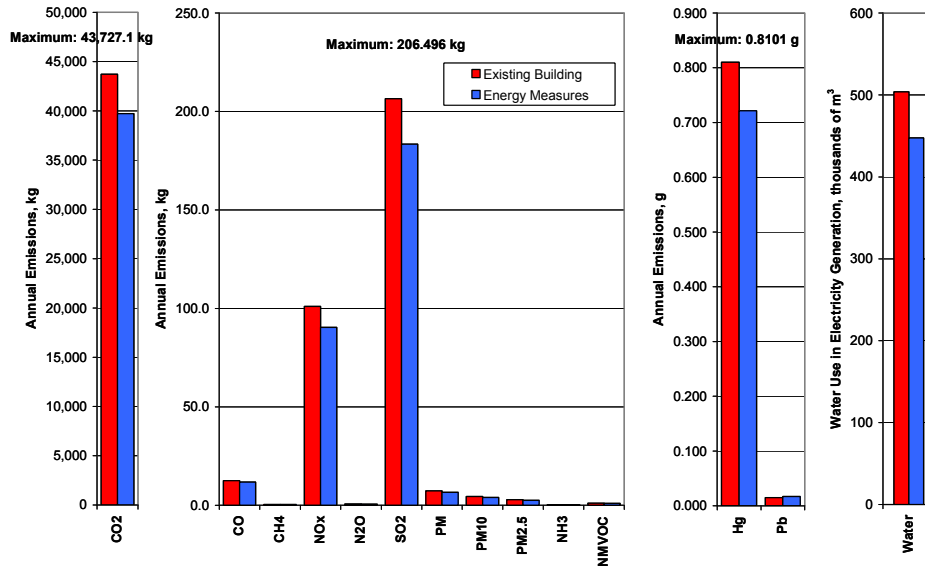


Figure 242. Example Annual Atmospheric Pollutants and Water Consumption

Chicago, Illinois

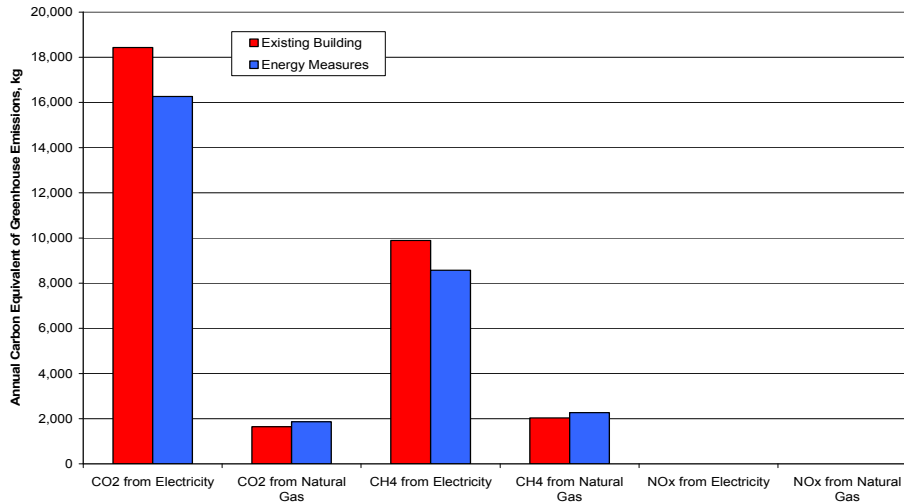


Figure 243. Example Annual Total Carbon Equivalent for Major Greenhouse Gases

Carbon Equivalent

The Intergovernmental Panel on Climate Change has studied the effects on the relative radiative forcing effects of various greenhouse gases. This effect, called Global Warming Potential (GWP), is described in terms of the Carbon Equivalent of a particular greenhouse gas. This equivalent is based on a factor of 1.0 for carbon. This group of gases includes carbon dioxide (CO₂), carbon monoxide, nitrous oxide, methane, halocarbon emission, hydrofluorocarbons (HFC), perfluorocarbons (PFC), and chlorofluorocarbons (CFC). For building energy use, the main gases of concern are carbon dioxide, carbon monoxide,

methane, and nitrous oxide. Although carbon monoxide has a relatively short life, CO emissions into the atmosphere may have a significant impact on climate forcing due to chemical impact on CH₄ lifetime, and tropospheric O₃ and CO₂ photochemical production normally reacts to produce carbon dioxide, but it can't be ignored since it is produced in incomplete combustion and the carbon remains to interact as CO₂. Yet there is no agreement on its carbon equivalent (IPCC 2001). The carbon equivalent of carbon dioxide, methane, and nitrous oxide are calculated and then multiplied by their GWP on a 100 year time frame. The Carbon Equivalents of the following gases have been determined and used in the program are shown in the following table.

Table 64. Carbon Equivalents (IPCC 2001)

Gas	Carbon Equivalent
NO _x	80.7272
CH ₄	6.2727
CO ₂	0.2727

The resulting carbon equivalents by fuel type are shown in the output of the program along with the individual gas pollutants.

Fossil Fuel Emissions Factors

Emission factors for on-site fossil fuel consumption are based on Section 1.4 Natural Gas Combustion in EPA (1998a) Table 65 shows the greenhouse gas and precursors and criteria pollutant emissions factors for natural gas. Similar emissions factors are shown for residual fuel oil (No. 4 and No. 6 fuel oil) [Table 66], distillates (No. 1 and No. 2 fuel oil) [Table 67], residential oil furnace [Table 68], LPG (butane and propane) [Table 69], gasoline and diesel [Table 70], and coal [Table 71] in the indicated tables. Note that a zero for a pollutant in the table may mean that no data were available, not that there are no emissions of that pollutant.

Table 65. Emission Factors for Natural Gas

Pollutant	Emission Factor^a (g/MJ)
Carbon Dioxide (CO ₂)	50.23439
Carbon Monoxide (CO)	3.51641E-02
Methane (CH ₄)	9.62826E-04
Nitrogen Oxides (NO _x)	4.18620E-02
Nitrous Oxide (N ₂ O) ^b	9.20964E-04
Sulphur Dioxide (SO ₂) ^c	2.51172E-04
Particulate Matter (PM) ^d	3.18151E-03
Particulate Matter (PM10) ^d	2.38613E-03
Particulate Matter (PM2.5) ^d	7.95378E-04
Ammonia (NH ₃)	0 ^e
Volatile Organic Compounds (NMVOC)	2.30241E-03
Mercury (Hg)	1.08841E-07
Lead (Pb)	2.09310E-07

a Based on data from Tables 1.4-1, 1.4-2 and 1.4.4 in EPA (1998a), Natural gas heat value of 1027 Btu/ft³ based on data for 2003 in Table A-4 in DOE (2004)

b Values shown are for uncontrolled burner. For controlled-low NO_x burner, use 0.64 lb/106 ft³, 0.000627 lb/MMBtu, 0.0002679 g/MJ

c Based on 100% conversion of fuel sulfur to SO₂. Assumes sulfur content is natural gas of 2,000 grains/106 ft³. The SO₂ emission factor can be converted to other natural gas sulfur contents by multiplying the SO₂ emission factor by the ratio of the site-specific sulfur content (grains/106 ft³) to 2,000 grains/106 ft³.

d PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

e No data.

Table 66. Emission Factors for Residual Fuel Oil (No. 4 and No. 6 Fuel Oil)

Pollutant	No. 6 Fuel Oil	No. 4 Fuel Oil
	Emission Factor ^a (g/MJ)	Emission Factor ^a (g/MJ)
Carbon Dioxide (CO ₂)	76.77128	76.77128
Carbon Monoxide (CO)	1.53543E-02	1.53543E-02
Methane (CH ₄)	1.45865E-03	6.63304E-04
Nitrogen Oxides (NO _x)	1.68897E-01	6.14170E-02
Nitrous Oxide (N ₂ O)	3.37794E-04	3.37794E-04
Sulphur Dioxide (SO ₂) ^b	4.82124E-01	4.60628E-01
Particulate Matter (PM) ^c	2.56109E-02	2.14960E-02
Particulate Matter (PM10) ^c	1.58763E-02	1.58763E-02
Particulate Matter (PM2.5) ^c	5.89603E-03	5.89603E-03
Ammonia (NH ₃)	0 ^d	0 ^d
Volatile Organic Compounds (NMVOC)	3.47006E-03	1.04409E-03
Mercury (Hg)	3.47006E-06	3.47006E-06
Lead (Pb)	4.63699E-06	4.63699E-06

a Based on data from Tables 1.3-1, 1.3-3, 1.3-8, 1.3-10, and 1.3-12 in EPA (1998b).

b Based on 100% conversion of fuel sulfur to SO₂. Assumes 1% sulfur content. The SO₂ emission factor in this table can be converted to other natural gas sulfur contents by multiplying the SO₂ emission factor by percentage sulfur content.

c PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

d No Data.

Table 67. Emission Factors for Distillates (No. 1 and No. 2 Fuel Oil)

Pollutant	No. 1 Fuel Oil	No. 2 Fuel Oil
	Emission Factor ^a (g/MJ)	Emission Factor ^a (g/MJ)
Carbon Dioxide (CO ₂)	66.02330	68.47998
Carbon Monoxide (CO)	1.53543E-02	1.53543E-02
Methane (CH ₄)	6.63304E-04	6.63304E-04
Nitrogen Oxides (NO _x)	6.14170E-02	7.37004E-02
Nitrous Oxide (N ₂ O)	3.37794E-04	3.37794E-04
Sulphur Dioxide (SO ₂) ^b	4.36061E-01	4.82124E-01
Particulate Matter (PM) ^c	6.14170E-03	6.14170E-03
Particulate Matter (PM10) ^c	3.31652E-03	3.31652E-03
Particulate Matter (PM2.5) ^c	2.54881E-03	2.54881E-03
Ammonia (NH ₃)	0 ^d	0 ^d
Volatile Organic Compounds (NMVOC)	1.04409E-03	1.04409E-03
Mercury (Hg)	3.47006E-06	3.47006E-06
Lead (Pb)	4.63699E-06	4.63699E-06

a Based on data from Tables 1.3-1, 1.3-3, 1.3-8, 1.3-10, and 1.3-12 in EPA (1998b).

b Based on 100% conversion of fuel sulfur to SO₂. Assumes 1% sulfur content. The SO₂ emission factor in this table can be converted to other natural gas sulfur contents by multiplying the SO₂ emission factor by percentage sulfur content.

c PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

d No data.

Table 68. Emission Factors for Residential Oil Furnace

Pollutant	Emission Factor^a (g/MJ)
Carbon Dioxide (CO ₂)	68.48237
Carbon Monoxide (CO)	1.53543E-02
Methane (CH ₄)	5.46612E-02
Nitrogen Oxides (NO _x)	5.52753E-02
Nitrous Oxide (N ₂ O)	1.53543E-04
Sulphur Dioxide (SO ₂) ^b	4.36061E-01
Particulate Matter (PM) ^c	2.14960E-02
Particulate Matter (PM10) ^c	1.58763E-02
Particulate Matter (PM2.5) ^c	5.89603E-03
Ammonia (NH ₃)	0 ^d
Volatile Organic Compounds (NMVOC)	2.18952E-03
Mercury (Hg)	3.47006E-06
Lead (Pb)	4.63699E-06

a Based on data from Tables 1.3-1, 1.3-3, 1.3-8, 1.3-10, and 1.3-12 in EPA (1998b).

b Based on 100% conversion of fuel sulfur to SO₂. Assumes 1% sulfur content. The SO₂ emission factor in this table can be converted to other natural gas sulfur contents by multiplying the SO₂ emission factor by percentage sulfur content.

c PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

d No data.

Table 69. Emission Factors for LPG (butane and propane)

Pollutant	LPG (butane)	Propane
	Emission Factor^a (g/MJ)	Emission Factor^a (g/MJ)
Carbon Dioxide (CO ₂)	66.02330	68.47998
Carbon Monoxide (CO)	1.53543E-02	1.53543E-02
Methane (CH ₄)	6.63304E-04	6.63304E-04
Nitrogen Oxides (NO _x)	6.14170E-02	7.37004E-02
Nitrous Oxide (N ₂ O)	3.37794E-04	3.37794E-04
Sulphur Dioxide (SO ₂) ^b	4.36061E-01	4.82124E-01
Particulate Matter (PM) ^c	6.14170E-03	6.14170E-03
Particulate Matter (PM10) ^c	3.31652E-03	3.31652E-03
Particulate Matter (PM2.5) ^c	2.54881E-03	2.54881E-03

Ammonia (NH ₃)	0 ^d	0 ^d
Volatile Organic Compounds (NMVOC)	1.04409E-03	1.04409E-03
Mercury (Hg)	3.47006E-06	3.47006E-06
Lead (Pb)	4.63699E-06	4.63699E-06

a Based on data from Table 1.5-1 in EPA (1996), Higher Heating value of 1.02 MMBtu/gal for butane and 0.915 MMBtu/gal for propane based on data in EPA (1996).

b Based on 100% conversion of fuel sulfur to SO₂. Assumes sulphur content is 0.18 gr/100 ft³. The SO₂ emission factor can be converted to other LPG sulphur contents by multiplying the SO₂ emission factor by the ratio of the site-specific sulphur content gr/100 ft³ to 0.18 gr/100 ft³.

c PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

d No data.

Table 70. Emission Factors for Gasoline and Diesel

Pollutant	Gasoline Emission Factor^a (g/MJ)	Diesel Emission Factor^a (g/MJ)
Carbon Dioxide (CO ₂)	66.20808	70.50731
Carbon Monoxide (CO)	2.69561E+01	4.08426E-01
Methane (CH ₄)	0 ^c	0 ^c
Nitrogen Oxides (NO _x)	7.00774E-01	1.89596E+00
Nitrous Oxide (N ₂ O)	0 ^c	0 ^c
Sulphur Dioxide (SO ₂)	3.61135E-02	1.24678E-01
Particulate Matter (PM) ^b	0 ^c	0 ^c
Particulate Matter (PM10) ^b	4.29923E-02	1.33276E-01
Particulate Matter (PM2.5) ^b	0 ^c	0 ^c
Ammonia (NH ₃)	0 ^c	0 ^c
Volatile Organic Compounds (NMVOC)	9.02837E-01	1.50473E-01
Mercury (Hg)	0 ^c	0 ^c
Lead (Pb)	0 ^c	0 ^c

a Based on data from Table 3.3-1 in EPA (1996), Diesel higher heating value of 19,300 Btu/lb, and gasoline higher heating value of 20,300 Btu/lb based on data in EPA (1996).

b PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

c No data.

Table 71. Emission Factors for Coal

Pollutant	Bituminous Emission Factor^a (g/MJ)	Anthracite Emission Factor^b (g/MJ)	Lignite Emission Factor^c (g/MJ)
Carbon Dioxide (CO ₂)	91.11052	99.26669	152.12646
Carbon Monoxide (CO)	8.26774E-03	1.04859E-02	8.26774E-03
Methane (CH ₄)	6.61419E-04	0 ^f	0 ^f
Nitrogen Oxides (NO _x)	1.98426E-01	3.14578E-01	2.34804E-01
Nitrous Oxide (N ₂ O)	4.96065E-04	0 ^f	0 ^f
Sulphur Dioxide (SO ₂) ^d	6.28348E-01	6.81585E-01	9.92129E-01

Particulate Matter (PM) ^e	1.65355E-01	1.74765E-01	2.18268E-01
Particulate Matter (PM10) ^e	3.80316E-02	4.01960E-02	7.60632E-02
Particulate Matter (PM2.5) ^e	9.92129E-03	1.04859E-02	2.18268E-02
Ammonia (NH ₃)	0 ^f	0 ^f	0 ^f
Volatile Organic Compounds (NMVOC)	9.92129E-04	2.14961E-02	1.32284E-03
Mercury (Hg)	6.94490E-06	2.27195E-06	2.74489E-06
Lead (Pb)	1.37245E-06	1.55541E-04	1.38898E-05

a Based on data on pulverized coal from Tables 1.1-3, 1.1-6, 1.1-18, 1.1-19 in EPA (1998a), Coal average higher heating value of 26.0 MMBtu/ton based on EPA (1998a).

b Based on data on pulverized coal from Tables 1.2-1, 1.2-2, 1.2-3, 1.2-4, 1.2-7 in EPA (1996), Coal average higher heating value of 24.6 MMBtu/ton based on EPA (1996).

c Based on data on pulverized coal from Tables 1.7-1, 1.7-3, 1.7-7, 1.7-14 in EPA (1998b), Coal average higher heating value of 13.0 MMBtu/ton based on EPA (1998b).

d Based on 100% conversion of fuel sulfur to SO₂. Assumes 1% sulfur content. The SO₂ emission factor in this table can be converted to other natural gas sulfur contents by multiplying the SO₂ emission factor by percentage sulfur content.

e PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 are particles smaller than 10 and 2.5 microns, respectively. Expressed in terms of coal ash content, assumes 1% ash content. Multiply weight % ash content of coal (as fired) by the value.

f No data.

Off-Site Electricity Generation Emissions

While estimating emissions from on-site fossil fuel combustion can be fairly straight-forward, emissions from off-site electricity is more challenging. How the electricity is generated, i.e., from gas, oil, coal, nuclear, renewable sources (wind, PV) or hydroelectric, and the mix of generation determines the resulting level of emissions. While data are available at utility and even power plant level (from the sources cited), data are shown here for United States national- and state-level average emissions from electricity generation. Table 72 provides average greenhouse gas and precursors and criteria pollutant emissions factors for the entire United States from electricity generation. Table 73 provides average electricity emissions factors by state, for greenhouse gas and precursors, and Table 74 for criteria pollutant emission factors. These two tables also include a ratio of heat input to electric output (efficiency of electricity generation) including distribution and transmission losses to allow calculation of source energy.

As mentioned in the introduction to this section, EnergyPlus also calculates water consumed through evaporation in thermo-electric and hydro-electric generation. Torcellini, Long, and Judkoff (2004) provide data on average water consumption by generator type by state. These data are summarized in units suitable for EnergyPlus in Table 72, for national and state average water consumption for thermal-electric, hydro-electric, and Table 73 for weighted total electricity generation.

Table 72. United States National Average Emission Factors for Electricity Generation

	Efficiency Ratio (J/J)
Ratio of Heat Input to Electricity Output ^a	2.253
Pollutant	Emission Factor (g/MJ)
Carbon Dioxide (CO ₂) ^b	168.333168
Carbon Monoxide (CO) ^c	4.20616E-02
Methane (CH ₄) ^b	1.39858E-03
Nitrogen Oxides (NO _x) ^a	4.10753E-01
Nitrous Oxide (N ₂ O) ^b	2.41916E-03
Sulphur Dioxide (SO ₂) ^a	8.65731E-01

Particulate Matter (PM) ^{c,d}	2.95827E-02
Particulate Matter (PM10) ^{c,d}	1.80450E-02
Particulate Matter (PM2.5) ^{c,d}	1.15377E-02
Ammonia (NH ₃) ^c	1.10837E-03
Volatile Organic Compounds (NMVOC) ^a	3.72332E-03
Mercury (Hg) ^c	3.36414E-06
Lead (Pb)	0 ^e

^aData based on 1999 data from eGRID version 2.01 (EPA 2003a).

^bData based on 1998-2000 average data in DOE (2002).

^cData based on tier emissions report for criteria air pollutants in EPA (2003b).

^dPM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

^eNo data.

Table 73. U. S. State Average Greenhouse Gas Emission Factors for Electricity Generation, in g/MJ

	Ratio of Heat Input to Electric Output	Carbon Dioxide (CO ₂) ^b	Carbon Monoxide (CO) ^c	Methane (CH ₄) ^b	Nitrogen Oxides (NO _x) ^a	Nitrous Oxide (N ₂ O) ^b	Sulphur Dioxide (SO ₂) ^a
Alabama	2.230	165.30922	1.45087E+03	1.72617E-03	4.02311E-01	2.80975E-03	1.13940E+00
Alaska	2.734	173.87708	3.71694E+02	8.56786E-04	7.29024E-01	1.12138E-03	2.38136E-01
Arizona	1.694	132.29777	8.26924E+02	8.56786E-04	2.74423E-01	1.94037E-03	2.27552E-01
Arkansas	2.207	162.03327	6.41833E+02	1.57497E-03	2.87149E-01	2.55776E-03	4.24991E-01
California	1.422	76.35472	2.91370E+03	8.44186E-04	6.56449E-02	4.66192E-04	3.04915E-02
Colorado	3.101	242.67192	1.51197E+03	1.60017E-03	4.74004E-01	3.64134E-03	5.83874E-01
Connecticut	1.720	118.69000	3.21421E+02	2.19236E-03	1.81563E-01	1.51197E-03	3.78750E-01
Delaware	2.736	230.57612	1.31290E+02	1.54977E-03	4.12517E-01	2.86015E-03	1.10866E+00
District of Columbia	4.844	172.11310	8.94585E+00	1.48677E-03	7.29528E-01	2.59556E-03	1.62487E+00
Florida	2.694	175.64105	6.12954E+03	1.88997E-03	4.73122E-01	2.26796E-03	1.00584E+00
Georgia	2.119	172.11310	1.05889E+03	1.62537E-03	4.00043E-01	2.84755E-03	1.11974E+00
Hawaii	2.950	209.40848	1.17682E+02	2.69635E-03	7.27512E-01	2.30576E-03	5.44437E-01
Idaho	0.213	3.52794	0	1.00798E-03	1.07098E-02	4.15793E-04	1.05838E-02
Illinois	1.694	146.66153	1.85292E+03	1.03318E-03	4.41749E-01	2.26796E-03	1.11811E+00
Indiana	3.281	261.57160	2.14058E+03	1.80177E-03	7.30284E-01	4.06973E-03	1.88846E+00
Iowa	3.033	237.12801	7.57877E+02	1.73877E-03	5.61447E-01	3.75474E-03	1.04566E+00
Kansas	2.826	212.18043	8.65983E+02	1.41118E-03	5.58801E-01	3.20035E-03	7.06596E-01
Kentucky	3.234	253.00374	1.50857E+03	1.76397E-03	8.41036E-01	4.04453E-03	1.79257E+00
Louisiana	2.624	148.42550	1.68116E+04	1.18438E-03	3.41958E-01	1.41118E-03	5.05755E-01
Maine	2.191	107.35019	4.92778E+02	7.11888E-03	1.79547E-01	3.40194E-03	4.04327E-01
Maryland	2.277	172.11310	4.89502E+02	1.48677E-03	5.37759E-01	2.59556E-03	1.38837E+00
Massachusetts	2.729	161.02529	7.89125E+02	2.19236E-03	2.89165E-01	2.00337E-03	8.01347E-01
Michigan	2.616	199.07665	1.69354E+03	1.83957E-03	4.91770E-01	3.14995E-03	9.76988E-01
Minnesota	2.331	163.04126	6.96264E+02	1.66317E-03	5.02354E-01	2.07897E-03	5.06889E-01
Mississippi	2.404	231.83610	2.17976E+03	1.58757E-03	4.66570E-01	3.62874E-03	8.98239E-01
Missouri	2.857	192.02077	1.29753E+03	1.97817E-03	6.41707E-01	3.11215E-03	9.07059E-01
Montana	1.936	180.68096	4.13147E+02	1.36078E-03	3.58212E-01	2.86015E-03	2.01219E-01
Nebraska	2.195	176.39703	4.68208E+02	1.19698E-03	3.94121E-01	2.75935E-03	5.29065E-01
Nevada	2.615	191.26478	3.82278E+02	1.13398E-03	4.01807E-01	2.45696E-03	4.02815E-01
New Hampshire	1.394	85.93055	2.63588E+02	2.16716E-03	2.03109E-01	1.77657E-03	8.71275E-01

	Ratio of Heat Input to Electric Output	Carbon Dioxide (CO ₂) ^b	Carbon Monoxide (CO) ^c	Methane (CH ₄) ^b	Nitrogen Oxides (NO _x) ^a	Nitrous Oxide (N ₂ O) ^b	Sulphur Dioxide (SO ₂) ^a
New Jersey	1.451	88.95450	2.27250E+03	9.70184E-04	1.76649E-01	9.95383E-04	2.31206E-01
New Mexico	3.307	254.26372	8.56408E+02	1.65057E-03	6.57583E-01	3.72954E-03	5.70140E-01
New York	1.808	108.10618	1.93835E+03	1.02058E-03	1.69089E-01	1.12138E-03	4.68082E-01
North Carolina	1.969	156.48937	1.10286E+03	1.32298E-03	4.68712E-01	2.55776E-03	1.00131E+00
North Dakota	3.244	282.48725	9.01389E+02	1.85217E-03	6.44731E-01	4.27133E-03	1.52697E+00
Ohio	2.736	226.79619	1.58757E+03	1.63797E-03	7.67579E-01	3.62874E-03	2.33562E+00
Oklahoma	3.024	216.96835	1.67262E+03	1.38598E-03	5.11425E-01	2.80975E-03	5.11047E-01
Oregon	0.526	35.53140	1.86855E+02	4.15793E-04	5.26671E-02	4.28393E-04	7.50947E-02
Pennsylvania	1.827	159.26132	1.85885E+03	1.34818E-03	3.29232E-01	2.55776E-03	1.25834E+00
Rhode Island	2.561	132.54977	1.67955E+02	8.56786E-04	6.21170E-02	5.92190E-04	4.53592E-03
South Carolina	1.300	105.08223	8.38642E+02	1.14658E-03	2.54264E-01	1.82697E-03	6.04790E-01
South Dakota	1.192	100.54631	9.79004E+01	6.67789E-04	5.44941E-01	1.52457E-03	5.81354E-01
Tennessee	1.902	163.29325	9.09579E+02	1.32298E-03	5.10165E-01	2.67116E-03	1.18123E+00
Texas	2.749	184.46090	9.63405E+03	9.70184E-04	3.27720E-01	1.83957E-03	4.90888E-01
Utah	3.095	243.67990	5.13063E+02	1.68837E-03	5.26545E-01	3.88073E-03	2.13314E-01
Vermont	0.306	3.52794	1.38472E+02	1.20958E-03	1.94037E-02	4.91392E-04	2.14196E-03
Virginia	1.924	146.66153	9.12729E+02	1.72617E-03	3.65016E-01	2.41916E-03	7.86857E-01
Washington	0.414	30.99548	4.29653E+02	4.66192E-04	5.30451E-02	5.03992E-04	1.90383E-01
West Virginia	2.917	248.97181	1.27938E+03	1.72617E-03	7.77659E-01	3.98153E-03	1.86918E+00
Wisconsin	2.680	206.88852	1.00471E+03	1.73877E-03	4.97440E-01	3.27594E-03	9.25076E-01
Wyoming	3.534	270.39145	9.01389E+02	1.85217E-03	5.59431E-01	4.25873E-03	5.78708E-01

a Data based on 1999 data from eGRID version 2.01 (EPA 2003a).

b Data based on 1998-2000 average data in DOE (2002).

c Data based on tier emissions report for criteria air pollutants in EPA (2003b).

d PM is the sum of all particulate matter including PM10 and PM2.5. PM10 and PM2.5 stand for particles smaller than 10 and 2.5 microns, respectively.

e No data.

Table 74. U. S. State Average Criteria Pollutant Emission Factors for Electricity Generation, in g/MJ

	Particulate Matter (PM) ^{cd}	Particulate Matter (PM10) ^{cd}	Particulate Matter (PM2.5) ^{cd}	Ammonia (NH ₃) ^c	Volatile Organic Compounds (NMVOC) ^a	Mercury (Hg) ^c	Lead (Pb) ^e
Alabama	7.91048E-03	7.86328E-03	4.72023E-05	3.55049E-05	4.66784E-03	5.14071E-06	0 ^e
Alaska	8.96502E-03	8.85977E-03	1.05247E-04	6.51454E-06	2.82297E-03	3.27594E-07	0 ^e
Arizona	1.70555E-02	1.69322E-02	1.23202E-04	1.80226E-04	2.27385E-03	1.88997E-06	0 ^e
Arkansas	9.27803E-03	9.19307E-03	8.49561E-05	4.59383E-04	3.46429E-03	2.73415E-06	0 ^e
California	7.16813E-03	7.07819E-03	8.99402E-05	4.02651E-03	2.62453E-03	1.38598E-07	0 ^e
Colorado	7.29822E-03	7.23699E-03	6.12291E-05	9.56430E-05	4.36770E-03	1.62537E-06	0 ^e
Connecticut	1.22734E-02	1.21694E-02	1.04033E-04	2.22944E-03	3.93896E-03	1.18438E-06	0 ^e
Delaware	1.39283E-02	1.38287E-02	9.96131E-05	1.54469E-03	4.74441E-03	3.62874E-06	0 ^e
District of Columbia	2.88269E-02	2.84861E-02	3.40760E-04	8.76496E-03	2.30080E-02	0	0 ^e
Florida	4.33040E-02	4.29055E-02	3.98460E-04	1.58386E-03	3.39110E-03	1.71357E-06	0 ^e
Georgia	2.05237E-02	2.03865E-02	1.37175E-04	7.51637E-05	2.16686E-03	3.20035E-06	0 ^e
Hawaii	5.97339E-03	5.90409E-03	6.92999E-05	3.55697E-03	7.01715E-03	1.77657E-06	0 ^e
Idaho	0	0	0	0	0	0	0
Illinois	1.53276E-01	1.52156E-01	1.12047E-03	5.93583E-03	2.77818E-02	4.62412E-06	0 ^e
Indiana	2.18862E-02	2.17008E-02	1.85453E-04	5.75861E-04	4.87283E-03	4.86352E-06	0 ^e
Iowa	2.34564E-02	2.32698E-02	1.86674E-04	9.09228E-05	4.78644E-03	6.19910E-06	0 ^e
Kansas	2.41783E-02	2.39535E-02	2.24799E-04	4.19292E-04	5.39089E-03	4.93912E-06	0 ^e
Kentucky	1.69397E-02	1.68139E-02	1.25825E-04	4.35029E-05	3.80922E-03	4.71232E-06	0 ^e
Louisiana	1.79917E-02	1.77843E-02	2.07304E-04	1.66720E-03	1.02357E-02	1.41118E-06	0 ^e

Maine	3.36399E-03	3.34131E-03	2.26813E-05	1.55132E-03	9.62614E-03	4.53592E-07	0 ^e
Maryland	2.13382E-02	2.12038E-02	1.34391E-04	7.21581E-04	2.78881E-03	4.88872E-06	0 ^e
Massachusetts	8.85244E-03	8.76617E-03	8.62772E-05	1.70282E-03	3.49171E-03	1.88997E-06	0 ^e
Michigan	9.08755E-03	9.00520E-03	8.23449E-05	2.22140E-04	2.79017E-03	3.67914E-06	0 ^e
Minnesota	4.04781E-02	4.01455E-02	3.32617E-04	6.74889E-05	4.13240E-03	3.30114E-06	0 ^e
Mississippi	5.44446E-02	5.37910E-02	6.53601E-04	4.06289E-02	1.54329E-02	2.45696E-06	0 ^e
Missouri	1.25537E-02	1.24368E-02	1.16929E-04	6.48638E-05	4.98085E-03	4.67452E-06	0 ^e
Montana	3.67504E-03	3.64126E-03	3.37798E-05	4.01020E-05	3.14400E-03	3.77994E-06	0 ^e
Nebraska	1.33751E-02	1.32829E-02	9.21636E-05	6.90470E-05	5.23031E-03	3.59094E-06	0 ^e
Nevada	2.09146E-02	2.07657E-02	1.48886E-04	3.79659E-04	3.33440E-03	1.36078E-06	0 ^e
New Hampshire	3.09503E-02	3.06487E-02	3.01568E-04	7.16019E-04	2.45937E-03	3.40194E-07	0 ^e
New Jersey	3.45712E-02	3.41460E-02	4.25200E-04	1.24216E-04	1.52830E-02	1.28518E-06	0 ^e
New Mexico	5.20754E-02	5.16748E-02	4.00653E-04	4.41665E-04	5.12951E-03	8.41666E-06	0 ^e
New York	5.35802E-03	5.31301E-03	4.50068E-05	2.22325E-03	4.26008E-03	1.33558E-06	0 ^e
North Carolina	3.40955E-02	3.38325E-02	2.62976E-04	3.00504E-05	1.73434E-03	3.33894E-06	0 ^e
North Dakota	5.17162E-02	5.13979E-02	3.18277E-04	6.41672E-05	6.88995E-03	8.71905E-06	0 ^e
Ohio	1.38722E-02	1.37957E-02	7.64501E-05	1.11541E-04	2.59201E-03	6.24949E-06	0 ^e
Oklahoma	1.83971E-02	1.82479E-02	1.49160E-04	9.73713E-04	4.68484E-03	3.88073E-06	0 ^e
Oregon	3.47911E-03	3.46199E-03	1.71186E-05	4.43277E-06	5.31933E-04	3.77994E-07	0 ^e
Pennsylvania	2.21604E-02	2.20060E-02	1.54356E-04	1.47657E-04	1.51542E-03	6.56449E-06	0 ^e
Rhode Island	1.03973E-03	1.02744E-03	1.22906E-05	0	2.25247E-03	0	0 ^e
South Carolina	2.45530E-02	2.43803E-02	1.72632E-04	2.51344E-05	1.16735E-03	1.54977E-06	0 ^e
South Dakota	4.67825E-03	4.63562E-03	4.26239E-05	8.82976E-05	4.63562E-03	1.22218E-06	0 ^e
Tennessee	2.51650E-02	2.48944E-02	2.70575E-04	2.70034E-05	2.88396E-03	2.98615E-06	0 ^e
Texas	1.73147E-02	1.71765E-02	1.38283E-04	1.26310E-03	4.32150E-03	3.52794E-06	0 ^e
Utah	1.47314E-02	1.46364E-02	9.50155E-05	9.59315E-05	2.48737E-03	9.70184E-07	0 ^e
Vermont	1.16247E-03	1.14873E-03	1.37415E-05	1.80704E-05	2.12073E-03	0	0 ^e
Virginia	1.22315E-02	1.21362E-02	9.53635E-05	2.93259E-04	2.50975E-03	2.21756E-06	0 ^e
Washington	5.37627E-04	5.32210E-04	5.41708E-06	6.46409E-06	6.87348E-04	5.92190E-07	0 ^e
West Virginia	2.39677E-03	2.38177E-03	1.50018E-05	4.25792E-05	3.09497E-03	6.55189E-06	0 ^e
Wisconsin	7.34187E-03	7.28252E-03	5.93472E-05	6.45613E-05	4.61829E-03	4.83832E-06	0 ^e
Wyoming	5.08215E-02	5.06042E-02	2.17349E-04	5.19787E-05	4.78782E-03	5.27931E-06	0 ^e

Table 75. United States National Average Water Consumption Factors^a

	Thermoelectric Generation		Hydroelectric Generation		Weighted Total Water Consumption
	L/MJ	Percent of Total Generation	L/MJ	Percent of Total Generation	L/MJ
United States	0.4960	89.4%	19.2095	8.6%	2.1007

^a Based on data from Torcellini, Long, and Judkoff (2004).

Table 76. U.S. State Average Water Consumption Factors for Electricity Generation^a

State	Thermoelectric Generation		Hydroelectric Generation		Weighted Total Water Consumption
	L/MJ	Percent of Total Generation	L/MJ	Percent of Total Generation	L/MJ
Alabama	0.1503	89.8%	38.9053	6.4%	2.6274
Alaska	0.3295	86.2%	--	13.8%	0.2839
Arizona	0.3313	88.3%	68.1928	11.7%	8.2533
Arkansas	0.3000	89.5%	--	5.7%	0.2684
California	0.0511	74.1%	21.9430	22.0%	4.8739
Colorado	0.5368	96.0%	18.8333	4.0%	1.2600
Connecticut	0.0860	90.8%	--	1.5%	0.0781
Delaware	0.0132	99.9%	--	0.0%	0.0132
District of Columbia	1.6959	100.0%	--	0.0%	1.6959
Florida	0.1506	95.7%	--	0.1%	0.1441
Georgia	0.6267	93.6%	49.8599	2.3%	1.7339
Hawaii	0.0440	92.4%	--	1.1%	0.0407
Idaho	0.0000	2.7%	8.9528	92.2%	8.2501
Illinois	1.1093	99.4%	--	0.1%	1.1032
Indiana	0.4350	99.6%	--	0.3%	0.4331
Iowa	0.1229	97.3%	--	2.5%	0.1196
Kansas	0.6099	100.0%	--	0.0%	0.6098
Kentucky	1.1521	97.2%	162.2884	2.8%	5.5990
Louisiana	1.6411	94.2%	--	0.9%	1.5461
Maine	0.3049	40.4%	--	28.7%	0.1231
Maryland	0.0343	95.3%	7.0617	2.7%	0.2259
Massachusetts	0.0000	92.4%	--	2.4%	0.0000
Michigan	0.5221	95.8%	--	1.4%	0.4999
Minnesota	0.4657	93.4%	--	2.4%	0.4351
Mississippi	0.4145	94.4%	--	0.0%	0.3912
Missouri	0.3213	97.4%	--	2.5%	0.3130
Montana	1.0051	55.8%	38.6619	44.1%	17.5997
Nebraska	0.2020	94.5%	2.2888	5.5%	0.3165
Nevada	0.5936	90.6%	77.1023	9.2%	7.6260
New Hampshire	0.1231	83.9%	--	8.6%	0.1033
New Jersey	0.0747	97.6%	--	0.0%	0.0729
New Mexico	0.6609	99.3%	71.5070	0.7%	1.1886
New York	0.8951	81.3%	5.8535	16.7%	1.7040
North Carolina	0.2445	95.5%	10.9089	3.1%	0.5751
North Dakota	0.3809	91.7%	60.7730	8.3%	5.3968
Ohio	0.9972	99.1%	--	0.3%	0.9884
Oklahoma	0.5378	93.7%	144.0133	5.8%	8.8254
Oregon	0.8633	18.4%	4.6351	80.7%	3.8990
Pennsylvania	0.5700	97.6%	--	1.0%	0.5563
Rhode Island	0.0000	98.2%	--	0.1%	0.0000
South Carolina	0.2754	97.2%	--	1.9%	0.2677
South Dakota	0.0143	36.7%	120.7558	63.2%	76.3811
Tennessee	0.0026	90.8%	45.5853	8.3%	3.7833
Texas	0.4595	99.0%	--	0.3%	0.4550
Utah	0.5959	96.6%	77.1150	3.4%	3.2090
Vermont	0.3642	71.5%	--	20.9%	0.2605
Virginia	0.0693	94.9%	--	0.9%	0.0657
Washington	0.3013	15.7%	3.3506	83.2%	2.8344

State	Thermoelectric Generation		Hydroelectric Generation		Weighted Total Water Consumption
	L/MJ	Percent of Total Generation	L/MJ	Percent of Total Generation	L/MJ
West Virginia	0.6180	99.0%	--	1.0%	0.6119
Wisconsin	0.5199	93.6%	--	3.3%	0.4867
Wyoming	0.5190	97.1%	144.0177	2.7%	4.3654

^a Based on data from Torcellini, Long, and Judkoff (2004).

Other Energy-Related Pollutants and Sources of Other Information

EnergyPlus (with user entered-data) will also calculate high- and low-level nuclear waste from electricity generation. Few utilities now provide data on nuclear waste resulting from electricity generation and no US national or state-level data are yet available (this will be added as data become available). Two Illinois utilities regularly report nuclear waste in terms of pounds per kWh or MWh for high-level waste and cubic feet per kWh or MWh for low-level waste. For these two utilities, high level nuclear waste values range from 0.0042 to 0.01 lb/MWh (7000 to 16000 g/MJ); low-level nuclear waste ranges from 0.0001 to 0.0002 ft³/MWh (0.01 to 0.02 m³/MJ) depending on relative proportion of nuclear as compared with other electricity generation sources.

IEA (2003) contains carbon dioxide (CO₂) emissions factors for electricity generation by country and region. Carbon dioxide (CO₂) is responsible for over 60% of the anthropogenic greenhouse effect (UNEP 2002). Because only limited greenhouse gas emissions factors and data (other than CO₂) is available for other countries, an interim method for estimating emission factors would be to compare the CO₂ emission factor for the particular country from IEA (2003) and match it to the state with the closest CO₂ emission factor in Table 73—using the other emissions factors for that state. Since the Kyoto Protocol (UNFCCC 1997) requires each country to report emissions of the major greenhouse gases [carbon dioxide (CO₂), methane (CH₄), and nitrous oxide (N₂O)] as well as ozone-depleting substances [hydrofluorocarbons (HFC), perfluorocarbons (PFC), and sulphur hexafluoride (SF₆)] and all energy consumption in their annual 'national communication', more complete emission factors for a larger number of countries should become available over the next few years. More information and other resources for calculating emissions factors are available in IPCC (2000, 1997).

References

- Crawley, Drury B. 2003. "Impact of Climate Change on Buildings," in *Proceedings of the CIBSE/ASHRAE International Conference 2003*, September 2003, Edinburgh, Scotland. London, England: CIBSE.
- Intergovernmental Panel on Climate Change. 2001. *Climate Change 2001: The Scientific Basis*. Cambridge: Cambridge University Press.
- Intergovernmental Panel on Climate Change. 2000. *Good Practice Guidance and Uncertainty Management in National Greenhouse Gas Inventories*, Paris, France: IPCC/OECD/IEA.
- Intergovernmental Panel on Climate Change. 1997. *Revised 1996 IPCC Guidelines for National Greenhouse Gas Inventories*, J.T. Houghton, L.G. Meira Filho, B. Lim, K. Treanton, I. Mamaty, Y. Bonduki, D.J. Griggs and B.A. Callender (editors). Paris, France: IPCC/OECD/IEA.
- International Energy Agency. 2003. *CO₂ Emissions from Fuel Combustion 1971-2001 (2003) -- 2003 Edition*. Paris, France: IEA.
- Liesen, Richard J. 1997. *Atmospheric Pollution Prediction in a Building Energy Simulation Program*, April 1997, BLAST Support Office, Department of Mechanical Engineering. Champaign, Illinois: University of Illinois.

- Stroot, Peter J., Robert J. Nemeth, & Donald F. Fournier. 1996. *Pollution Reduction Through Energy Conservation, REEP Model*. Champaign, Illinois: U S Army Construction Engineering Research Laboratory.
- Torcellini Paul A, Nicholas Long, and Ronald D. Judkoff. 2004. "Consumptive Water Use for U.S. Power Production," in *ASHRAE Transactions*, Volume 110, Part 1. Atlanta, Georgia: ASHRAE.
- United Nations Environment Programme. 2002. *Climate Change Information Kit*. Châtelaine, Switzerland: UNEP.
- United Nations Framework Convention on Climate Change. 1997. *Kyoto Protocol to the United Nations Framework Convention on Climate Change*. Bonn, Germany: UNFCCC.
- U.S. Department of Energy. 2004. *Monthly Energy Review*. Washington, DC: Energy Information Administration, U S Department of Energy.
- U.S. Department of Energy. 2002. *Updated State-level Greenhouse Gas Emission Coefficients for Electricity Generation 1998-2000*. April 2002, Energy Information Administration, Office of Integrated Analysis and Forecasting, Energy Information Administration. Washington, DC: U.S. Department of Energy.
- U.S. Environmental Protection Agency. 2003a. *eGRID Emissions and Generation Resource Integrated Database*, May 2003, Washington, DC: U.S. Environmental Protection Agency.
- U.S. Environmental Protection Agency. 2003b. *AirData*, Tier Emissions Report - Criteria Air Pollutants, 1999 data, May 2003. Washington, DC: U.S. Environmental Protection Agency.
- U.S. Environmental Protection Agency. 1998a. *Compilation of Air Pollutant Emission Factors, AP-42, Fifth Edition, Volume I: Stationary Point and Area Sources*, Chapter 1 External Combustion Sources, Supplement D, July 1998. Research Triangle Park, North Carolina: U. S. Environmental Protection Agency.
- U.S. Environmental Protection Agency. 1998b. *Compilation of Air Pollutant Emission Factors, AP-42, Fifth Edition, Volume I: Stationary Point and Area Sources*, Chapter 1 External Combustion Sources, Supplement E, September 1998. Research Triangle Park, North Carolina: U. S. Environmental Protection Agency.
- U.S. Environmental Protection Agency. 1996. *Compilation of Air Pollutant Emission Factors, AP-42, Fifth Edition, Volume I: Stationary Point and Area Sources*, Chapter 1 External Combustion Sources, Supplement B, October 1996. Research Triangle Park, North Carolina: U. S. Environmental Protection Agency.