JCSS Technical Reference Manual

7.0 Final (OPNET 2.6.2)

Contract DASW01 03 D 0008

Disclaimer: As of October 2007, NETWARS was redesignated by the Program Manager Office as the Joint Communications Simulation System (JCSS). JCSS was selected as the new industry name to better reflect the inherent joint communication capabilities of the software. Users should be aware that no software updates were conducted as part of the software name change.

November 30, 2007

Prepared for:
Defense Contracting Command - Washington
Washington, DC 20310-5200

Prepared by:
OPNET Technologies, Inc.
7255 Woodmont Avenue
Bethesda, MD 20814-7904

OPNET®
Making Networks and Applications Perform™

Identification

Document Identification

Document Title: JCSS Technical Reference Manual
Version: 7.0 Final

Software Identification

Product Name: JCSS
Product Release: 7.0

Documentation Conventions

This documentation uses specific formatting and typographic conventions to present the following types of information:

- Objects, examples, and system I/O
- Object hierarchies
- Computer commands
- Lists and procedures

Objects, Examples, and System I/O

- Directory paths and file names are in standard Courier typeface:

```
C:\Netwars\User_Data\Projects
```

- Function names in body text are in italics:

op_dist_outcome()

- The names of functions of interest in example code are in bolded Courier typeface:

```
/* determine the object ID of packet's creation module */
src_mod_objid = op_pk_creation_mod_get (pkptr);
```

- Variables are enclosed in angle brackets (< >):

```
<NETWARS path>\Scenario_Builder\op_admin\err_log
```

Object Hierarchies

Menu hierarchies are indicated by right angle brackets (>); for example:

Edit > Preferences > Advanced

Computer Commands

These conventions apply to Windows systems and navigation methods that use the standard graphical-user-interface (GUI) terminology such as click, drag, and dialog box.

• Key combinations appear in the form “press **<button>+x**”; this means press the **<button>** and **x** keys *at the same time* to do the operation.

• The mouse operations *left-click* (or *click*) and *right-click* indicate that you should press the left mouse button or right mouse button, respectively.

Lists and Procedures

Information is often itemized in bulleted (unordered) or numbered (ordered) lists:

• In bulleted lists, the sequence of items is not important.

• In numbered lists, the sequence of items is important.

Procedures are contained within procedure headings and footings that indicate the start and end of the procedure. Each step of a procedure is numbered to indicate the sequence in which you should do the steps.

Document Revision History

Release Date	Product Version	Chapter	Description of Change
November 30, 2007	7.0 Final	All	- Began introduction of product name change to JCSS from NETWARS.
		3	- Updated figure 3-33 Logical View Selection dialog box, and removed refs to Frame Relay, VTC, etc., in this section. - Added list of filters to Show Network Browser section, and added note that Modeler will not launch from JCSS without a Modeler license. - Updated LOS Settings section to include info about altitude and altitude unit conversion. - Modified Deploy Circuits section to clarify that sample procedure is for generic circuit deployment and works for all circuits. - Added figures to Deploy Broadcast Network and Deploy Relationship sections. - Updated Results Browser figure to reflect addition of Capacity Planning tables/graphs for Link-16 (and UHF DAMA).
October 26, 2007	2007 Final	3	- Added sentence to “Show Logical Views” that Network Layer Filters shown are only those which apply to the deployed network. - Removed “Animate” section, and updated “Show Time Controller” section. - Added section on opening a Node Model using Modeler, and setting related preferences (pgs 3-32 and 3-33.) - Added “Save As Link Type” section (pg 3-100.) Added and/or updated figures: - 3-78 Free-Text Questions Report - 3-120 through 3-122 UHF DAMA SATCOM Wizard - 3-127 through 3-130 Multiplexer Circuit Deployment - 3-131 View Multiplexer Circuits dialog box - 3-132 Enable Cut-Through on Selected Devices - 3-133 Disable Cut-Through on Selected Devices - Removed “Scenario Treeview” references. - Added Topology > Deploy Broadcast Network, and Topology > Deploy Relationship.
		Appendix C	- Added “Undo/Redo Template Units” to “Editing the Contents of the Library” section.
October 10, 2007	2007 Draft	2	- Updated figures to reflect change to NETWARS v2007.
		3	Updated figures: - 3-8 updated models list - 3-21 updated dialog box options layout - 3-32 updated groups list

Release Date	Product Version	Chapter	Description of Change
			- 3-34 added UHF DAMA and Link 16 options and updated dialog box layout - 3-44 and 3-47 updated file path - 3-56 updated dialog box layout - 3-61 added Preview button - 3-63 and 3-64 changed 'phases' to 'scenarios' - 3-88 and 3-89 updated list of tables - 3-90 added Load button - 3-93 updated dialog box layout - 3-94 updated list order - 3-99 updated list of pre-configured OPFACs and organizations - 3-179 added figure of Probe Model Window - 3-181 updated list of attributes
			Changes to View menu: - Added View > Background > Add CADRG/CIB Raster Catalog Directories - Added View > Background > Add MrSID Image - Added View > Circuits - Added View > Wireless Domains - Added View > Visualize Protocol Configuration > IP Address Types
			Change to Scenarios menu: - Removed Scenarios > Scenario Components submenu
		4 Appendix B Appendix C	Changes to Topology menu: - Changed Topology > Import submenu to Topology > Import Topology - Added Topology > Import Topology > Convert DCI to NETWARS Scenario - Added Topology > Import Topology > From Configlets. - Changed Topology > Model Assistant > View Error Log to Topology > Model Assistant > Open Error Log. - Added Topology > Link Operations > Import Links from File. - Added Topology > Deploy Circuit. - Removed Topology > Configuration Utilities > Promina. - Changed Topology > Terrain > Define Trajectory to Topology > Define Trajectory.
			Change to Protocols menu: - Added Protocols > IPv6 submenu.
			Change to NetDoctor menu: - Added NetDoctor > Configure/Run Trending - Added NetDoctor > Open NetDoctor Log.
			Change to DES menu: - Changed DES > Panel Options submenu to DES > Panel Operations.
			- Updated Figure 4-1 to reflect new toolbar icons in Icon Database. - Added File > Manage Model Files submenu, and Edit > Preferences.
			- Updated figures to reflect change to NETWARS v2007.
			- Updated Figure C-1to reflect current list of pre-configured OPFACs and organizations in Library Treeview.

Release Date	Product Version	Chapter	Description of Change
December 13, 2006	6.2 Final	2	- Removed refs to User's Guides 1-4, since they are no longer built into the NETWARS software.
		3	- Clarified use of Rules folders (pg TR-3-22). - Removed unnecessary step (old step 2) from View > Set Area of Interest section. - Removed reference to Map menu (changed to View > Background > Map Edit Mode). - Added Topology > Define Trajectory. - Updated Figure 3-151, Traffic Flows Import dialog box (now Figure 3-155), and accompanying text description. - Removed ref to traffic_import_suppress_unrecognized_ sources_dialog preference from Unrecognized Traffic Assistant section. - Removed Traffic > Specify Core/Edge Devices. - Updated list of Capacity Planning web reports (pg TR-3-145).
		4	- Updated instructions for accessing Icon Database and locating files.
December 8, 2006	6.2 Final	1	- Removed descriptions of editors that are not provided with standard NETWARS.
		2	- Updated "Open Project" section to direct users to use the Project directory. - Updated Figure 2-13 to reflect version 12.0. - Updated Figure 2-19 and associated text to reflect new Preferences Editor.
		3	- Removed duplicate step from "Edit > Preferences > Device Model Map > New" section, and added ref to specific directory. - Updated figures 3-38, 3-79, 3-87, 3-88, 3-92, 3-170, 3-171, 3-179, 3-180, and 3-181. - Added "DES > IER Results" section.
		Appendix C	- Updated refs from 11.5 to 12.0.

Release Date	Product Version	Chapter	Description of Change
November 3, 2006	6.2 Final	3	- Provided updated figures: 3-1 (Scenario Builder with a project open); 3-3, 3-5, 3-7, and 3-33 (removed references to old UserGuide_SB_Scenario); 3-6 (removed Add Logical View button); 3-34 (removed Type of View drop-down list, and Re-use/Re-create options); 3-35 (added new Settings drop-down); 3-79 (names in CCSD Summary Report); 3-87 and 3-88 (clearer); 3-101 (Library treeview changes); 3-113 (added Classification drop-down); 3-111, 3-117, 3-187, and 3-188 (reflect new toolbar buttons). - Added new figures 3-36 and 3-37 for network showcase feature. - Removed Map Menu (moved Map > Set Area of Interest to View > Set Area of Interest, moved Map > Background > <options> to View > Background > <options>, removed Map > Background > Add CADRG/CIB Raster catalog directories, moved and renamed Map > Background > Toggle Map Edit Mode to View > Background > Map Edit Mode, moved Map > LOS Settings to View > Show LOS > LOS Settings, moved Map > Clear LOS Ranges to View > Show LOS > Clear LOS Ranges, moved Map > Display LOS Legend to View > Show LOS > Display LOS Legend, moved Map > View OPFAC Distances to View > OPFAC Distances > View OPFAC Distances, moved Map > Remove OPFAC Distances to View > OPFAC Distances > Remove OPFAC Distances, removed Map > Define Trajectory, moved Map > Animate to Scenarios > Animate.) - Added Topology > Configuration Utilities > UHF DAMA SATCOM and Topology > Configuration Utilities > Cut-through > <options>. - Moved Terrain menu options to Topology menu. - Added Ctrl+W shortcut keys to Traffic Wizard section.
		Appendices B, C, and D	- Updated NETWARS System Adminstrator's Manual and moved to new Appendices in this manual.

Release Date	Product Version	Chapter	Description of Change
October 13, 2006	6.2 Draft	1	- Removed references to the IER Editor.
		2	- Updated System Editor and Sign-in figures to reflect version 6.2. - Removed File > Open Editor and Open Editor dialog box. - Added File > New Project, File > Open Project, File > Open Subordinate Query, File > New Other, File > Open Other, File > Manage Model Files, and File > Recent Projects. - Added IER Database menu and options. - Added Help > Show All Logs.
		3	- Added File > Close, File > Page Setup, File > Print Graphs, and File > Manage Model Files. - Removed File > Close Project and File > Close Subordinate Query. - Added Edit > Save Object Selection Set, Edit > Load Object Selection Set, Edit > Edit Objects Using Template, and Edit > Edit Attribute Template. - Replaced Advanced Preferences dialog box with new Preferences Editor dialog box. - Added View > Show Logical Views, View > Show Network Topology, View > Show Network Showcase, View > Open Network Showcase Window, View > Demands, View > Paths, View > Annotations, View > Layout > Lay Out Nodes Interactively, View > Zoom > To Window, and View > Zoom > To All. - Updated Edit Visualization Preferences dialog box. - Changed View > Layout > Scale Selected Icons to View > Layout > Scale Node Icons Interactively. - Removed View > Network Views > Logical. - Updated Scenario menu to reflect name change to Scenarios menu. Added Scenarios > Scenario Components, Scenarios > Reports > CCSD Summary, and Scenarios > Reports > SLD Summary. - Added Protocols > Servers, Protocols > Mainframes, Protocols > TCP, Protocols > MPLS, Protocols > Ethernet, Protocols > FDDI, Protocols > MANET, Protocols > UMTS, and Protocols > Wireless LAN. - Removed Topology > Import > Template OPFAC into Library from XML. - Added DES > Choose Statistics (Advanced), DES > Expert Service Prediction, DES > Configure/Run Discrete Event Simulation (Advanced), DES > Restart Discrete Event Simulation, DES > Results > View Application Delay Tracking, and DES > Panel Operations > Reload Data Into All Panels. - Removed DES > Results > Compare Results. - Updated Choose Results dialog box.
		4 & 5	- Removed IER Editor chapter (functionality now accessible via System Editor's IER Database menu) and renumbered chapters.
		Glossary	- Removed references to the Scenario Conversion Module.

Release Date	Product Version	Chapter	Description of Change
March 30, 2006	6.1 Final	3	- Updated Traffic Flow Preferences dialog box figure to reflect checkbox options design. - Removed "Rename Scenarios " feature from Manage Scenarios section. - Added Import Scenario > TNAPS to XML. - Removed Import Scenario > From Visio. - Added shortcut keys for configuring/viewing TSSP circuits. - Added Configuration Utilities > Multiplexer > Configure Multiplexer Circuits and View Multiplexer Circuits. - Noted the default username and password for the IER database.
		Appendix A	- Removed Multiplexer Utility Nodes section.
March 3, 2006	6.1 Draft	2	- Updated System Editor and Sign-in figures to reflect version 6.1. - Added Edit > Preferences > Device Model Map > *(options).* - Removed Help > Generate Support Info. - Renamed Help > Error Log > View as Help > Error Log > Open. - Renamed Help > Message Log > View as Help > Message Log > Open. - Added Help > Session Log > *(options)*.
		3	- Updated "Deploying Satellite Links" section to reflect new tabbed design of Satellite Bent Pipe Link Deployment dialog box. - Updated TSSP Circuit Deployment dialog box figure (and corresponding text description) to reflect new tabbed design. - Updated View TSSP Circuits dialog box figure (and corresponding text description) to reflect new column names. - Removed Traffic > Convert Traffic > Flows to IERs.
		4	Added Edit > Preferences.
		5	- Updated figure (and text) to reflect name change from "Icon Editor" to "Icon Database", and new toolbar buttons. - Added File > Exit.

Release Date	Product Version	Chapter	Description of Change
November 14, 2005	5.2 Final	Front material	Changed refs in Documentation Conventions section to be NETWARS-specific rather than OPNET-specific (i.e, changed <opnet_user_home>\... ref to <NETWARS path>\Scenario_Builder\...)
		2	- Added statement (“Please contact NETWARS CM to register your license.”) to Starting License Manager section. - Changed Express method description to “(Requires authorized access.)” in Adding a License section. - Updated Task Tracking Workflow GUI figure.
		3	- Updated Edit > Project Defaults > IER figure. - Updated View > Visualize Link Loads > Color by Link Load figure. - Updated View > Network Views > Logical figure. - Added Map > Background > Add CADRG/CIB Raster catalog directories. - Renamed Scenario > Scenario Documentation as Scenario > Set Scenario Documentation. - Corrected ref to old command (Topology > Import > Subordinate Response) in Scenario > Import Scenario > Subordinate Response. - Updated Object Palettes to include Tree view and reflect new icons. - Updated Annotation Palette. - Removed Topology > Link Operations > Deploy TSSP Groups. - Added Deploying Satellite Links section. - Added Topology > Configuration Utilities > TSSP > Configure TSSP Groups (Circuits.) - Added Topology > Configuration Utilities > TSSP > View TSSP Group Configs (Circuits.) - Updated Topology > Import Device Configuration Files. - Removed Advanced Traffic Wizard. - Updated Traffic Wizard figure. - Removed Traffic > Convert Traffic > Link Loads to Flows. - Removed Traffic > Convert Traffic > Open Conversion Log. - Removed Traffic > Convert Traffic > Clear Conversion Log. - Updated Terrain > Edit Parameter Sets figure.

Release Date	Product Version	Chapter	Description of Change
October 10, 2005	5.2 Draft	All	- Updated document footers and applicable screenshots to reflect new NETWARS version number.
		1	- Added (optional) Server Characterization and Mainframe Characterization options to System Editor (File > Open Editor.)
		2	- Added Edit > User Level options. - Updated Figure 2-4 (Open Editor dialog box) to reflect addition of (optional) Server and Mainframe Characterization editors. - Updated Figure 2-9 (Security Classifications dialog box) to reflect addition of Set Default button. - Updated Figure 2-11 (Advanced Preferences dialog box) to reflect new prefs and correct CORE version in title bar.) - Removed figure of License Manager Help dialog box (not necessary to document since Help changes with each version.) - Added Previous Editor and Circulate Editors options to Windows menu. - Added Help > Documentation > User's Guides options.
		3 File menu	- Removed references to 'demands'. - Updated File > Close Project to reflect that both the project and the Scenario Builder window close (instead of the project only.) - Updated Figure 3-6 (Briefings dialog box) to reflect addition of Add Logical View button. - Added Figure 3-7 (Print Preview dialog box.) - Renamed File > Package as File > Package Project Files, and old File > Package submenu 'Project Files' as 'Project'. - Added File > Recent Projects (with submenu) to replace old list of recent projects. - Removed File > Close Scenario Builder. - Renamed File > Exit NETWARS as File > Exit.
		Edit menu	- Renamed Edit > Select All in Subnet as Edit > Select All. - Added Edit > Find Node/Link. - Renamed Edit > Select as Edit > Group Selection/Action > Selection Filter - Renamed Edit > Apply Action as Edit > Group Selection/Action > Action Filter - Renamed Edit > Object Palette as Topology > Open Object Palette - Renamed Edit > Annotation Palette as Topology > Open Annotation Palette - Moved Edit > Device Model Map submenu to Edit > Preferences > Device Model Map. - Added Edit > Preferences > Device Model Map > Open. - Renamed Edit > Defaults > Device Model Map as Edit > Preferences > Device Model Map > Choose. - Removed Edit > Defaults > Demands, Edit > Defaults > Bandwidth Requirements, and Edit > Defaults > Application Conversion.

Release Date	Product Version	Chapter	Description of Change
		View menu	- Added View > Show Scenario Treeview. - Added View > Layout > Geographic Positioning. - Added View > Layout > Layout Nodes Automatically (Balanced). - Renamed & moved View > Edit Visualization Preferences to View > Layout > Edit Display Preferences. - Renamed & moved View > Zoom In to View > Zoom > To Rectangle. - Renamed & moved View > Zoom to Selection to View > Zoom > To Selection. - Renamed & moved View > Zoom Out to View > Zoom > Unzoom. - Renamed View > Redraw as View > Refresh Workspace. - Removed View > Network Views > Organization Systems and View > Network Views > Device Systems. - Renamed View > Network Views > Logical Views as View > Network Views > Logical. - Renamed Figure 3-42 (View Filtering dialog box) as Logical View Selection dialog box & added detail to dialog box options. - Renamed & moved View > Set View Properties to Map > Background > Set Properties. - Added View > Visualize Protocol Configuration > IP Tunnel Configuration.
		Map menu	- Renamed Map > Set Area of Operations as Map > Set Area of Interest. - Replaced Figure 3-45 (Set Area of Operations dialog box) with new Set Area of Interest dialog box. - Updated Figure 3-46 (Setting background view properties) to reflect addition of Translucency column and Edit CADRG/CIB Properties button. - Replaced Figure 3-47 (Set Area of Operations dialog box) with new Choose Border Map dialog box. - Renamed Map > Background > Import Image as Map > Background > Add Image Map. - Renamed Map > Background > Import Background Image as Map > Background > Add Image. - Removed Map > Background > Import JMTK Data. - Renamed Map > Background > Import MIF Data as Map > Background > Add MIF Map. - Added Figure 3-50 (MIF Import Log dialog box.) - Updated Figure 3-57 (Define Trajectory dialog box) to reflect changes. - Added Figure 3-58 (Trajectory Segment Information dialog box.)
		Scenario menu	- Added Scenario > Manage Scenarios and Figure 3-64 (Manage Scenarios dialog box.) - Removed Scenario > Remove Scenario. - Renamed Scenario > Import as Scenario > Import Scenario. - Renamed Scenario > Import > NETWARS Scenario as Scenario > Import Scenario > From NETWARS.

Release Date	Product Version	Chapter	Description of Change
			- Renamed Scenario > Import > OPNET Modeler Scenario as Scenario > Import Scenario > From OPNET Modeler. - Renamed Scenario > Import > Scenario from XML as Scenario > Import Scenario > From XML. - Renamed Scenario > Import > Scenario from Visio XML as Scenario > Import Scenario > From Visio. - Removed Scenario > Export > Export to HTML. - Renamed Scenario > Export as Scenario > Export Scenario. - Renamed Scenario > Export > Scenario to XML as Scenario > Export Scenario > To XML. - Added Scenario > Export Scenario > To JNMS. - Renamed Scenario > Export > Scenario to Visio XML as Scenario > Export Scenario > To Visio. - Renamed Scenario > Summary Tables as Scenario > Reports. - Renamed Scenario > View Task Organization Report as Scenario > Reports > Task Organization. - Added Scenario > Reports > User-Defined Reports > Open Live Report Table and Scenario > Reports > User-Defined Reports > Generate Report from Template. - Added Scenario > Generate Scenario Web Report and Scenario > Generate Scenario Bitmap. - Added Figures 3-87 (Generate Scenario Web Report dialog box), 3-88 (Displaying Scenario Web Report in Web Browser), and 3-89 (Model Image Capture Preview dialog box.)
		Topology menu	- Added Topology > Open Library Treeview and Figure 3-95 (Library Treeview.) - Renamed Topology > Device Creator as Topology > Create Custom Device Model. - Renamed Topology > Import > Subordinate Response as Scenario > Import Scenario > Subordinate Response. - Renamed Topology > Export > Subordinate Response as Scenario > Export Scenario > Subordinate Response. - Added Topology > Link Operations > Deploy TSSP Groups and Figure 3-101 (TSSP Circuit Deployment.) - Removed Topology > Link Operations > Redeploy Link.
		Traffic menu	- Added Traffic submenus for Traffic Wizard, IERs, Flows, Device/Link Loads, and Convert Traffic. - Added descriptions of Traffic Wizards (Simple & Advanced.) - Removed Traffic > Collector Import Settings and Traffic > Visualize submenus. - Renamed Traffic > IER/Demand Import Options as Traffic > IERs > Set Import Options. - Renamed Traffic > Specify Demands as Traffic > IERs > Specify IERs. - Added Traffic > IERs > Import from IER Report. - Renamed Traffic > Visualize > Export Traffic Report as Traffic > IERs > Export IER Report. - Added Traffic > IERs > Export IERs to XML. - Renamed Traffic > Import Flows > (options) as Traffic > Flows > Import > (options). - Renamed Traffic > Create Traffic Flows as Traffic > Flows > Create Flows. - Replaced Create Traffic Demands dialog box in Figure 3-132 with Create IP Unicast Traffic Flows dialog box.

Release Date	Product Version	Chapter	Description of Change
			- Renamed Traffic > Visualize > Open Flows Browser as Traffic > Flows > Open Flows Browser. - Renamed Traffic > Visualize > Export Traffic Flows as Traffic > Flows > Export to Spreadsheet. - Updated Figure 3-139 (Export Traffic Flows dialog box.) - Renamed Traffic > Import Device/Link Baseline Loads > (options) as Traffic > Device/Link Loads > Import > (options). - Removed Traffic > Convert NETWARS Applications to Demands. - Renamed Traffic > Convert Applications to Flows as Traffic > Convert Traffic > Applications to Flows. - Updated Figure 3-140 (Convert Applications to Flows dialog box) to remove Time Step settings. - Renamed Traffic > Convert Link Loads to Flows > Configure/ Convert as Traffic > Convert Traffic > Link Loads to Flows. - Renamed Traffic > Convert Traffic Flows to NETWARS Demands as Traffic > Convert Traffic > Flows to IERs. - Renamed Traffic > Convert Link Loads to Flows > Open Conversion Log as Traffic > Convert Traffic > Open Conversion Log. - Renamed Traffic > Convert Link Loads to Flows > Clear Conversion Log as Traffic > Convert Traffic > Clear Conversion Log. - Added Traffic > Generate Traffic Web Report. - Renamed Traffic > Visualize > Visualization Aggregate Traffic Preferences as Traffic > Set Aggregate Traffic Preferences. - Renamed Traffic > Collector Import Settings > Specify Core/Edge Devices as Traffic > Specify Core/Edge Devices.
		Protocols menu	- Added Protocols > LDP, Protocols > RSVP, Protocols > VLAN, and Protocols > STP. - Removed Protocols > MPLS and Protocols > DPT. - Updated Figure 3-149 (Terrain Data Directory Preferences dialog box) to reflect addition of Format drop-down list and support for terrain data type ‘OpenFlight’. - Updated Figure 3-151 (Import Elevation Lines dialog box) to reflect reorganization of dialog box fields, etc.. - Updated Figure 3-153 (Evaluation Settings dialog box) to reflect reorganization of dialog box buttons.
		NetDoctor menu	- Added NetDoctor > Auto-Generate Report Template..., NetDoctor > Suppress Messages, NetDoctor > Rule Development..., and NetDoctor > Options. - Removed NetDoctor > Security.
		DES menu	- Removed DES > Record Animation for Subnet. - Added DES > Record Packet Flow Animation for Subnet and DES > Record Node Movement Animation for Subnet. - Removed DES > Color Links by Utilization. - Removed DES > Clear Link Visualization. - Removed DES > Panel Operations > Export Panels > To HTML.
		Windows menu	- Added Previous Editor and Circulate Editors options to Windows menu.
		Glossary	- Removed ‘Demands’ from Glossary.

Release Date	Product Version	Chapter	Description of Change
April 1, 2005	5.1 Final	Cover	Changed release date to April 1, 2005, and removed OPNET contact info from Identification section.
March 21, 2005	5.1 Final	1	- Added ACE to ACE Whiteboard section.
		2	- Added ACE option to System Editor (File > Open Editor.)
		2 and 3	- Corrected definition of functional names in Edit > Preferences > Functional Profiles.
		3	- Added View > Zoom to Selection and View > Layout options. - Added Scenario > New/Duplicate/Previous/Next Scenario. - Updated Figure 3-18 to reflect new button placement.
February 21, 2005	5.1 Final		No changes.
February 7, 2005	5.1 Draft	All	Ported existing Word document (NETWARS v 4.2) into Adobe FrameMaker 7.1, and applied commercial OPNET product documentation style template to improve quality and update manual. Reformatted text as needed to fit new format and style. Content changes: - Updated applicable screenshots to reflect new NETWARS version number. - Added reference to ACE doc in chapter 1, and added ACE Whiteboard option to System Editor (File > Open Editor) and Scenario Builder (Traffic > Import Flows.) - Removed Capacity Planning > Restore Link Colors and Capacity Planning > Show Link Legend. - Removed Alternate Link Cost field from Capacity Optimization Settings Advanced Parameters dialog box.

Contents

List of Figures

List of Tables

1 Introduction

The *JCSS Technical Reference Manual* is a comprehensive reference and guide that covers all aspects of JCSS operation and use. It contains over 200 figures that provide visual examples of the JCSS interfaces, step-by-step procedures, and other important JCSS features and concepts.

Organization of the chapter topics is menu-based, meaning topics are grouped and ordered according to the various menus and menu options available in the respective JCSS interfaces.

Editor Interfaces

The capabilities of the JCSS software are organized into separate editors:

- The System Editor,
- The Scenario Builder, and
- Other optional editors that require an extra license (To learn more about available editors, refer to the IT Guru documentation suite.)

System Editor

The System Editor acts as an entry point to the remaining editors. It allows you to access user and profile information as well as the task assistant interface.

Scenario Builder

The Scenario Builder allows you to build template OPFACs and organizations, define infrastructure and Information Exchange Requirements (IERs) between OPFACs and organizations, specify movement and geographical context for OPFACs and organizations, and run discrete event simulations for developed scenarios. You can open and work with multiple projects simultaneously, and easily cut, copy and paste supported objects (Organizations, OPFACs, devices, links, etc.) between them.

Additionally, the Scenario Builder provides you with the option of deploying a communications infrastructure using a top-down approach. More specifically, the Lead Planner sets up the planning view and sends the initial scenario to the Subordinate Planners. The Subordinate Planners modify their portion of the initial plan and then send it back to the Lead Planner who integrates them into a complete plan.

The Scenario Builder allows you to optimize link and broadcast network capacities as well as traffic load. The optimization of the link and broadcast network capacities is implemented by changing the size of links and broadcast networks while the optimization of the traffic load is implemented by changing the load caused by the IERs.

This editor also allows you to view results of the simulation. After the simulation is complete, the Simulation Domain creates text files that capture the results of the simulation. The Scenario Builder reads these files and displays the results graphically as Measures of Performance (MOPs). The MOPs focus on the ability of selected communication equipment to send and receive information (e.g. file transfers, situation awareness updates, e-mail messages, etc.).

2 System Editor

The System Editor is the editor that opens first when you start JCSS. From the System Editor, you can open any of the other JCSS editors, like the Scenario Builder. The System Editor allows you to access user and profile information, as well as the task assistant interface.

Accessing the System Editor

Starting up the JCSS software launches the System Editor and a Sign In dialog box, as shown below.

Figure 2-1 System Editor

Figure 2-2 Sign In dialog box

1) By clicking the **New** button in the Sign In dialog box, the system launches the New Profile dialog box as shown below. The * designates a required field and must be filled-in to continue the sign-in process.

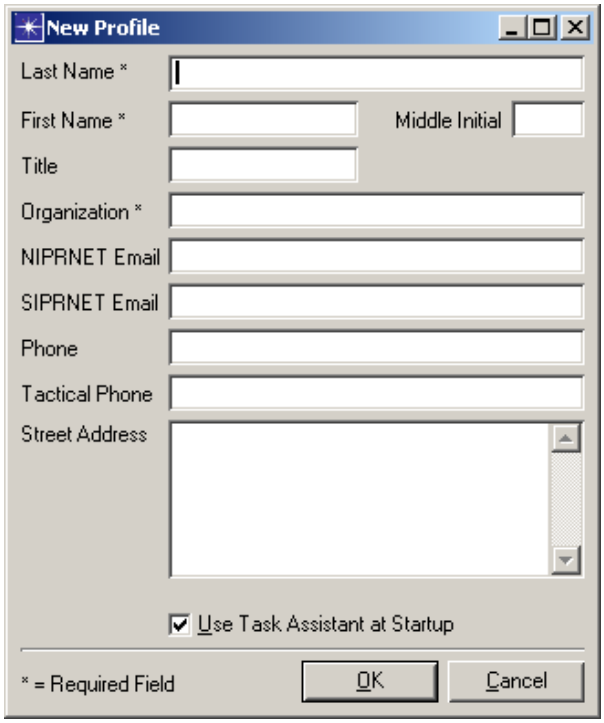


Figure 2-3 New Profile dialog box

2) Enter required information and any optional information. Click **OK**.

   JCSS will save up to five (5) user names so that multiple users can use the software without re-typing their user information. This information is used to track the changes made in the scenario by the planner. The Subordinate Planners also use this information when they need to send the subordinate response files back to the Lead Planner. If you have logged in before, select your name from the drop-down menu. To enable the Task Assistant to open upon sign-in, ensure that the **Use Task Assistant at Startup** is checked.

3) The Sign In dialog box displays again with the new user name in the Profile field. Sign in by clicking the **Sign In** button or, if you elect to edit your profile, you may do so at this time by clicking the **Edit** button.

File Menu

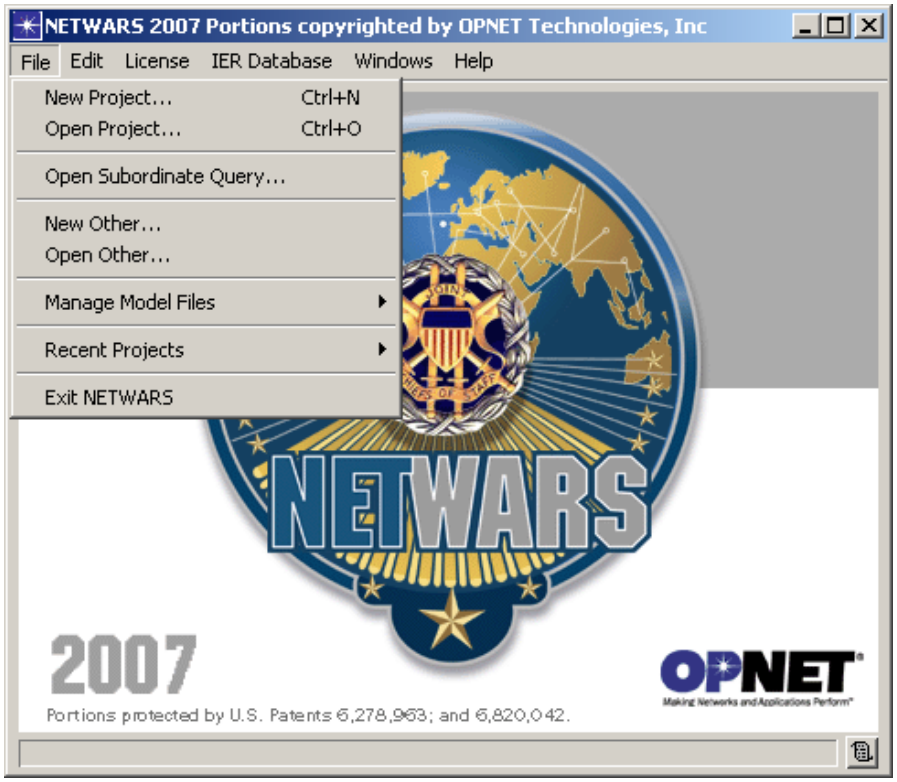


Figure 2-4 System Editor's File menu

New Project **File > New Project...**: Opens a new Scenario Builder and displays the Create Project dialog box. Create a new project, the first step in the Lead Planner's workflow. You can simply name and launch a generic project, or use the New Project Wizard to select and set the area of operations, top-level units (including their locations and templates), an equipment list for each organization, an owner for each unit, and the types of equipment to use in the new project.

1) Specify the names for the project and the initial scenario in the Create Project dialog box.

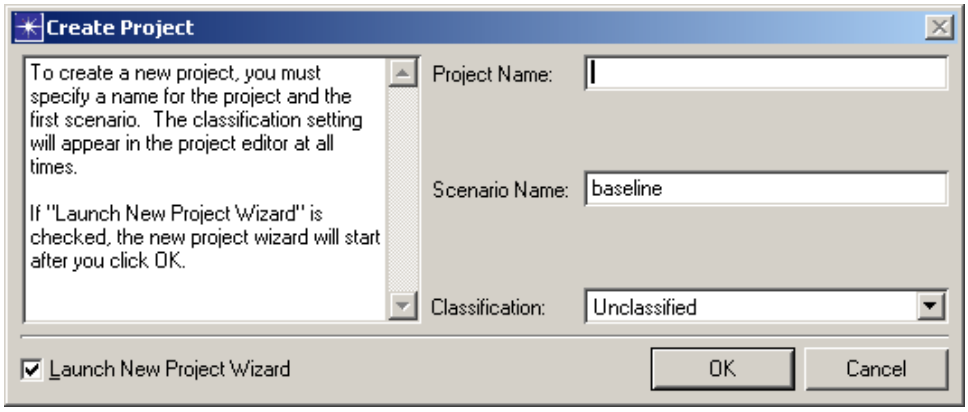


Figure 2-5 Creating a new project

2) Select a classification level for the project in the Create Project dialog box.

3) Click **OK**. If **Launch New Project Wizard** is not checked, then JCSS creates a generic project using the names you have designated. If **Launch New Project Wizard** is checked, then JCSS displays a sequence of dialog boxes that enable you to make selections specific to your new project.

Note—The names for both the project and the scenario cannot contain any spaces or special characters. In addition, the scenario name cannot be the same as any existing OPFAC; it must be unique.

Open Project

File > Open Project...: Displays the Open Project dialog box in which you can select a project file to open, and then opens the selected file in a new Scenario Builder.

1) Navigate to the NETWARS Project directory, and select the .prj file corresponding to the project that you wish to open.

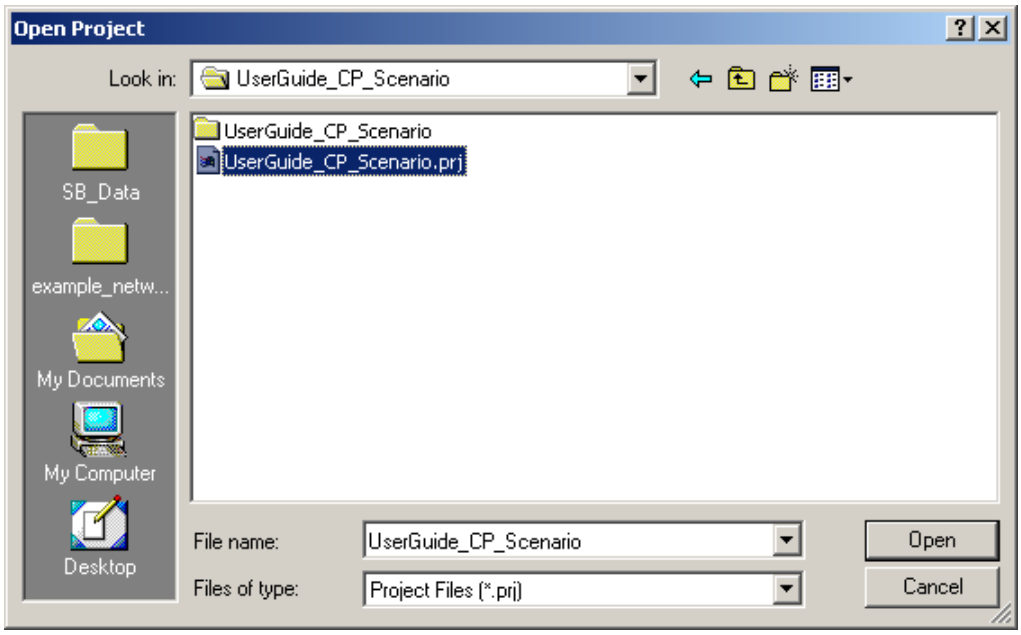


Figure 2-6 Opening a Project in Scenario Builder

Open Subordinate Query

File > Open Subordinate Query...: Displays the Open Subordinate Query dialog box in which you can select a query file to open, and then opens the selected file in a new Scenario Builder.

New Other

File > New Other...: Displays the New dialog box which lists standard OPNET editors from which you can select to create.

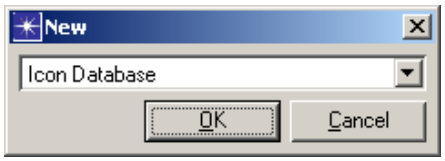


Figure 2-7 New dialog box

Open Other

File > Open Other...: Displays the Open dialog box which lists standard OPNET files from which you can select to open.

Manage Model Files

File > Manage Model Files > *(option)*: This submenu lists options which allow you to manage model files.

Manage Model Files > Delete Model Files

File > Manage Model Files > Delete Model Files...: Displays a list of models from which you can select to delete files.

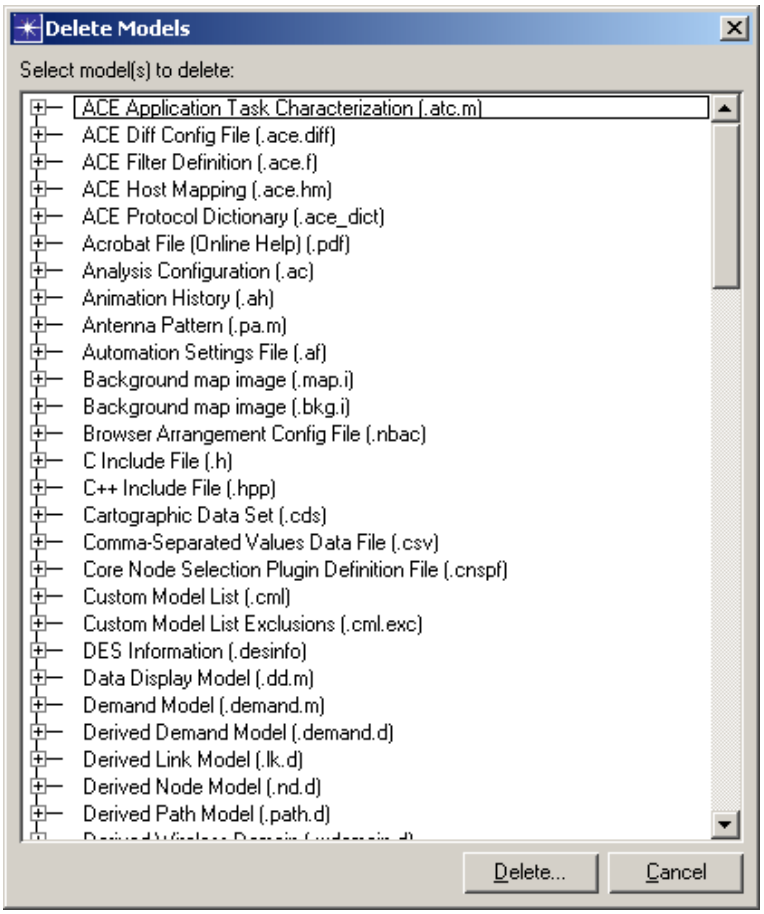


Figure 2-8 Delete Models dialog box

Manage Model Files > Add Model Directory

File > Manage Model Files > Add Model Directory...: Displays the Directory Browser in which you can create a new directory for models.

Manage Model Files > Refresh Model Directories

File > Manage Model Files > Refresh Model Directories: JCSS maintains a list of folders called the model directories (or mod_dirs for short) where the models are stored. Models include OPFACs, organizations, projects and scenarios, icon databases, and device models. When you move such models around on the hard drive using Windows Explorer, the software does not recognize the change. To make the software aware of the changes, use this option.

Recent Projects

File > Recent Projects > *(option)*: This submenu lists recently opened projects for easy re-opening.

Exit NETWARS

File > Exit NETWARS: Exit NETWARS.

Edit Menu

Figure 2-9 System Editor's Edit menu

Login Information

Edit > Login Information: Edit user profile information.

1) After modifying profile information, click **OK**.

Change User

Edit > Change User: Change the current signed in user and sign in information at any time by using the System Editor.

1) To change the user's profile, choose the user's name from the drop-down menu, as shown in the figure below.

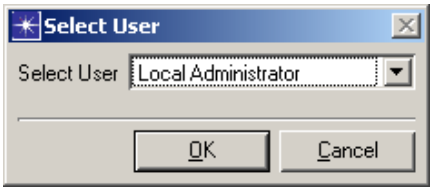


Figure 2-10 Select User dialog box

2) Click **OK** when modifications are complete.

User Level

Edit > User Level > *(option)*: Switch between JCSS interfaces (simple to complex) by selecting one of the available User Level options. JCSS menus change to reflect the selected User Level. Higher User Levels display more menu items and menu headers. You can switch between User Levels at any time, so that all menu items can be available if you so choose. Switching the User Level affects all open editors.

User Level > Beginner

Edit > User Level > Beginner: Select this User Level to access a simplified JCSS interface. The Beginner interface hides access to many of the functionalities of the system without removing the capabilities themselves. This results in an interface that is reduced in complexity, and is easier to navigate for beginner JCSS users.

User Level > Intermediate

Edit > User Level > Intermediate: Select this User Level to access a JCSS interface that exposes more functionalities of the system than the Beginner level, but less than the Advanced level. This results in an interface that is reduced in complexity, and is easier to navigate for intermediate users.

User Level > Advanced

Edit > User Level > Advanced: Select this User Level to access the full-featured JCSS interface. The Advanced interface exposes all of the functionalities of the system. This results in an interface that is more complex, and best utilized by more experienced JCSS users.

Preferences

Edit > Preferences > *(option)*: Display and edit JCSS preference settings.

Note—These menu options perform the same functions as the **Preferences** options under the Edit menu in the Scenario Builder.

Preferences > Device Model Map

Edit > Preferences > Device Model Map > *(option)*: Set default device model map files.

Preferences > Device Model Map > New...

Edit > Preferences > Device Model Map > New...: Using the Choose Source and Target Model Name Lists dialog box, select the paths to where you want new source and target model list .txt files to reside, and then click **OK**.

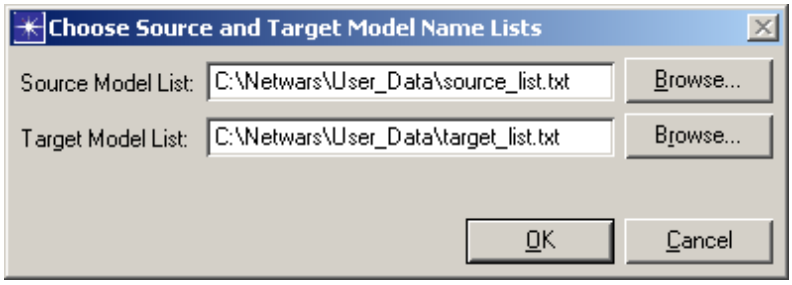


Figure 2-11 Choose Source and Target Model Name Lists dialog box

Preferences > Device Model Map > Open...

Edit > Preferences > Device Model Map > Open...: Using the Device Model Map Browser, navigate to and select an existing JCSS device model map .xml file to open, and then click **Open**.

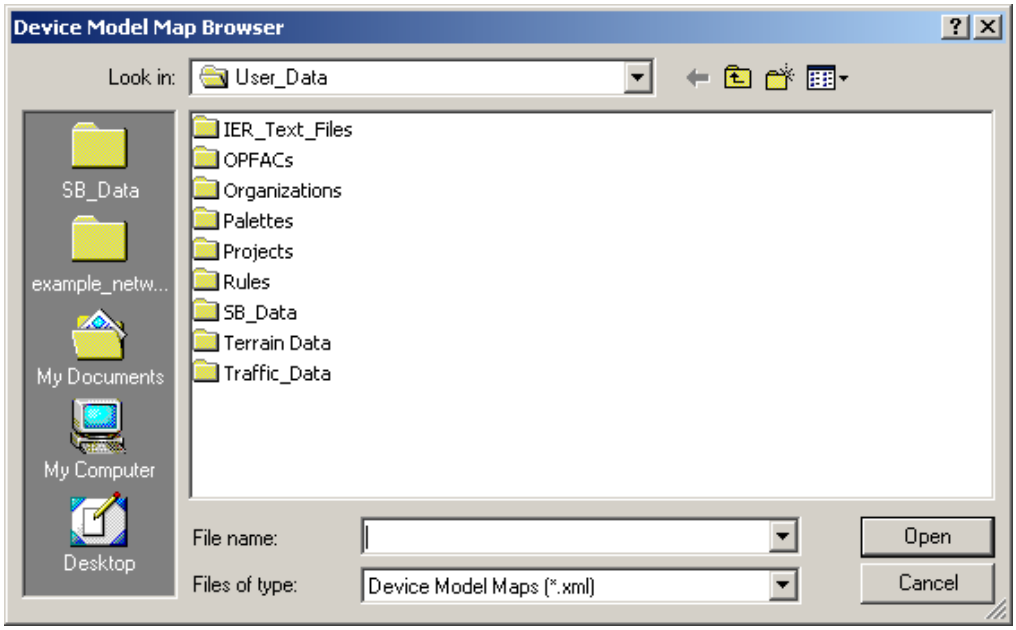


Figure 2-12 Device Model Map Browser

Preferences > Device Model Map > Choose...

Edit > Preferences > Device Model Map > Choose...: Using the Device Model Map Preferences dialog box, navigate to and select a device model map .xml file to import into JCSS as needed, and then click **OK**.

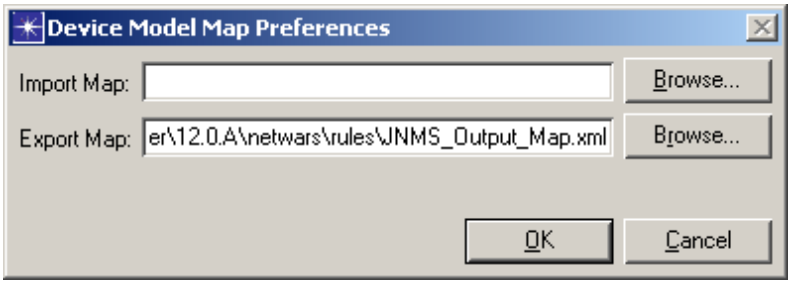


Figure 2-13 Device Model Map Preferences dialog box

Preferences > Functional Profiles

Edit > Preferences> Functional Profiles: Set default functional profiles and functional names. Every OPFAC has one or more functional names. A functional name maps an IER or series of IERs to a producer/consumer OPFAC pair. A collection of such functional names is called a functional profile. A functional profile can be defined locally on an individual OPFAC, or globally, so that it is available to all OPFACs and across different scenarios.

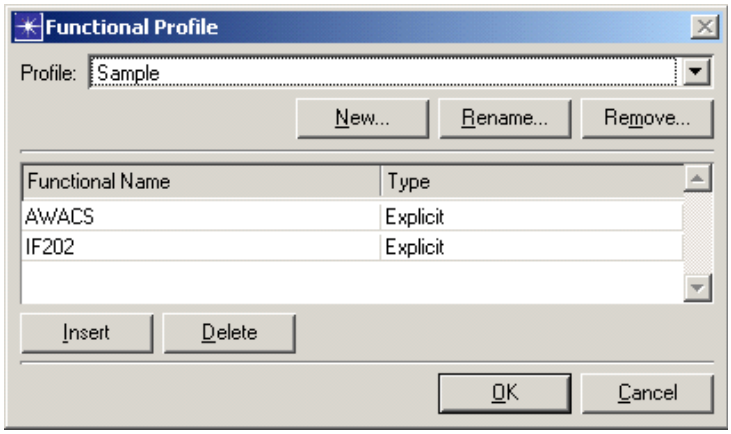


Figure 2-14 Functional Profile dialog box

1) Add, rename, or remove functional profiles as needed.

2) Insert, delete, or edit functional names as needed.

3) Click **OK**.

Preferences > Geographical Coordinates

Edit > Preferences> Geographical Coordinates: Sets the default geographical coordinate system view displayed in the workspace of the Scenario Builder. There are three types of geographical coordinate systems available in JCSS: Lat/Long, UTM, and MGRS.

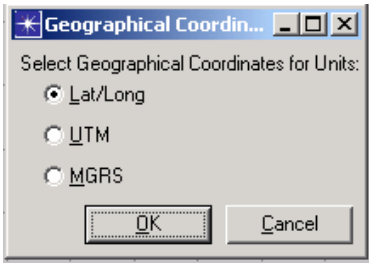


Figure 2-15 Geographical Coordinates dialog box

1) Select the **Lat/Long**, **UTM**, or **MGRS** radio button.

2) Click **OK**.

Preferences > Owners List

Edit > Preferences> Owners List: Set owner information for units marked for subordinate query.

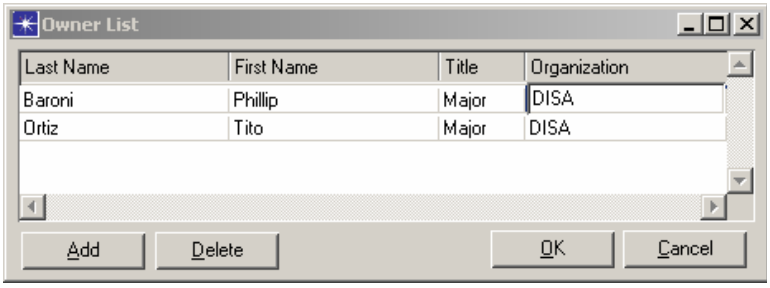


Figure 2-16 Owner List dialog box

Owner information contains four fields: Last Name, First Name, Title and Organization. All fields, except the Title field, are required fields.

- Modify existing records or add/delete records to the list as needed.

Preferences > Security Classifications

Edit > Preferences> Security Classifications: Set security classification for a variety of items in a scenario, such as devices and IERs. This dialog allows you to add, remove, or change values in the list of recognized security classification values. The list is used to populate drop-down lists in edit fields and table cells that specify security classification values. Since some fields and cells only allow values from this list, you will need to modify this list in order to specify custom security classifications. Any value is permitted, although duplicates are not allowed in the list.

The list order indicates the classifications' levels of restriction. Less restrictive classifications appear toward the top of the list.

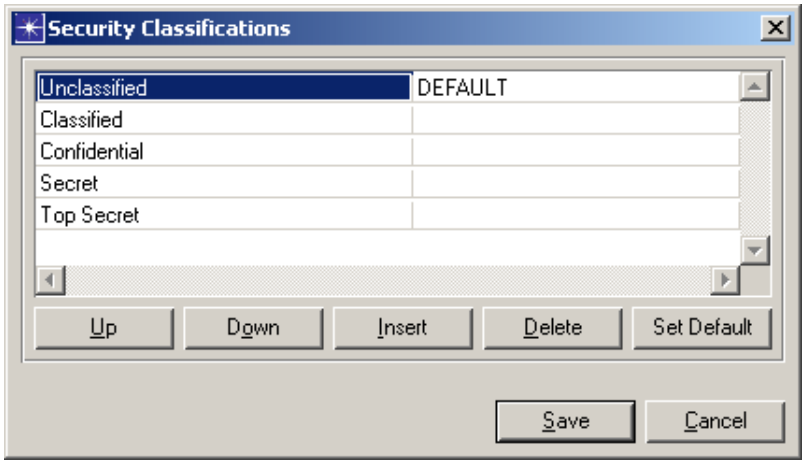


Figure 2-17 Security Classifications dialog box

A value in the list may be modified by selecting and editing it. The **Up** button moves the selected value toward the top of the list. The **Down** button moves the selected value toward the bottom of the list. The **Insert** button inserts a blank entry above the selected value. The **Delete** button deletes the selected value. The **Set Default** button sets the selected classification as your default. The **Save** button saves all changes. Once saved, the changes will affect all edit fields and table cells, but the values stored in those attributes will not be modified.

Preferences > Traffic Flow Thresholds

Edit > Preferences> Traffic Flow Thresholds: Specify a minimum value and the colors to be used for each threshold; traffic flow thresholds are used in conjunction with the Aggregate Traffic Flows feature. Flows are colored based on the threshold they fall under. If the load of an aggregate flow is more than the highest threshold, the flow is colored black. If the load is lesser than the minimum threshold, the flow is not displayed. The default minimum value is 0 Kbps.

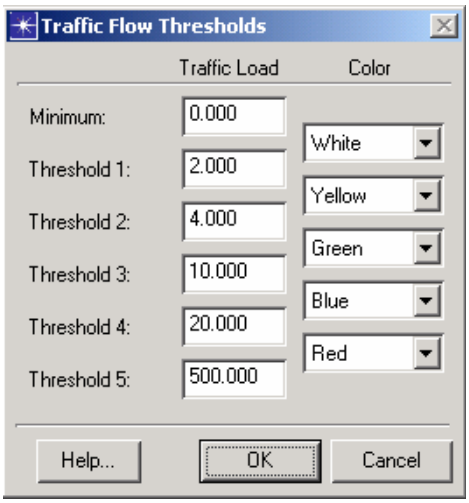


Figure 2-18 Traffic Flow Thresholds dialog box

- Set desired traffic load/color combinations, and then click **OK**.

Preferences > Advanced

Edit > Preferences> Advanced: View and/or set environment attributes, or "preferences", that control program operation. These values are stored in the environment database file (env_db for short).

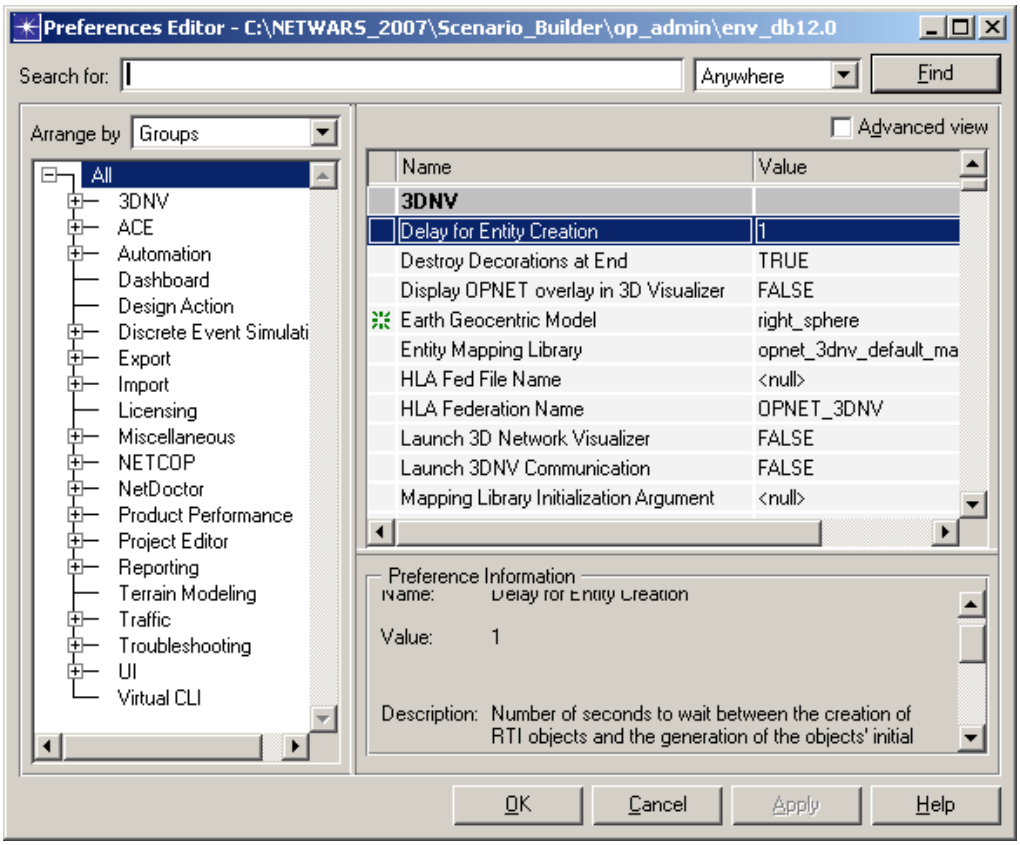


Figure 2-19 Sample Preferences Editor

1) Use the **Seach for** field (and drop-down list) to search for preferences that contain specific words or values (**Anywhere** specifies to search Tags, Names, Values and Descriptions; **In Names** specifies to search Tags and Names only; **In Values** specifies to search Values only.)

2) Use the **Arrange by** drop-down list to display preferences by category (**Groups** specifies to arrange preferences by group; **Source** specifies to arrange preferences by source.)

3) Check (or uncheck) the **Advanced view** checkbox to toggle between showing an advanced or basic view of the preferences table (basic view shows the name and value of each preference; advanced view shows the name, tag, source of the preference's value and the value itself.)

4) Edit desired preference values by clicking in the corresponding **Value** cell, selecting or typing the new value, and then pressing the **<Enter>** key on your keyboard.

5) Click **OK** to save edits and close the Preferences Editor.

License Menu

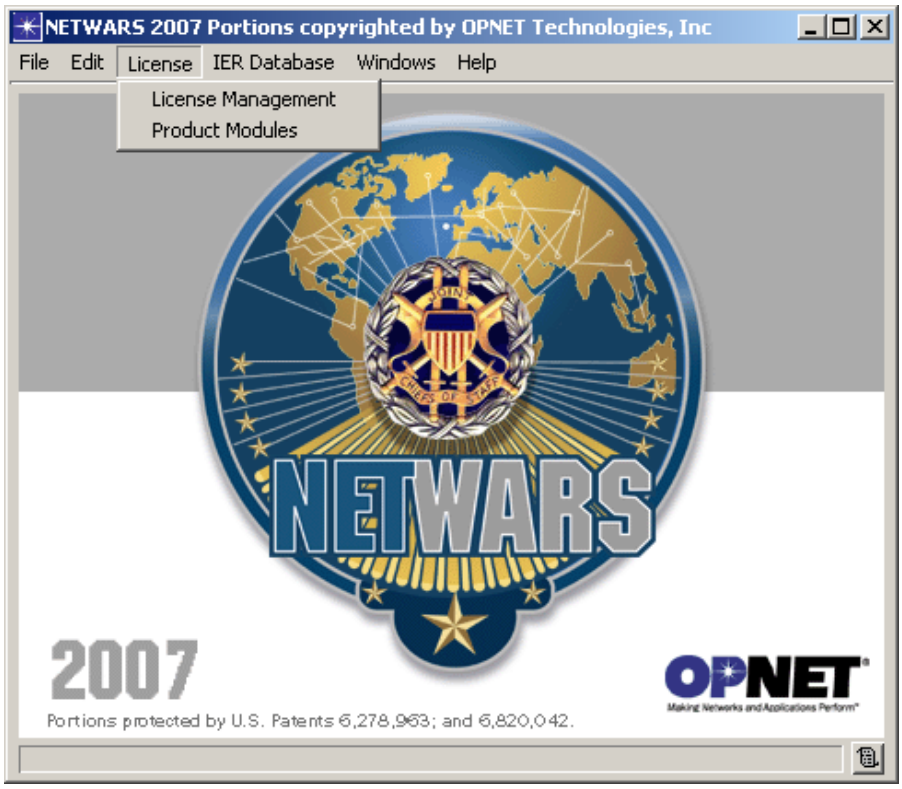


Figure 2-20 System Editor's License menu

License Management

License > License Management: A license conveys the right to use an application. A single license typically bundles one or more applications (such as Modeler), the simulation program, and one or more modules.

Licenses are allocated on a per-component basis. A component that is part of a license (perhaps a module such as TMMGUI) may be free and can be given to any requester, even if another component of the same license (most likely an application program) is in use.

License Terminology

- **Add license –** Add a license to the machine's license file.
- **Deregister License –** Remove a license from a machine's license file (it is then free to be installed on a different machine).
- **License –** Authorization to use an OPNET product.
- **License Manager –** Software interface for performing license operations.

- **License Server –** A service running on a machine that will distribute licenses to other computers.
- **Revoke License –** Free a license that is currently in use.
- **Standalone Mode –** A licensing mode in which only the local machine can obtain the licenses.

License Attributes

- **License number –** Specifies the software license. A single license may bundle several programs (for example, 100/1 that includes the programs IT Guru, Expert Service Prediction, and Multi-Vendor Import will be shown as three separate lines in the License Manager interface.)
- **Program name –** Specifies which programs are included in the license (such as IT Guru and the Radio module).
- **Status –** Identifies the state of the program.
  - A *free* program (shown in green) is available to the next user who requests it.
  - An *in use* program (shown in white) is unavailable.
  - An *expired* program (shown as yellow with red X) is no longer valid.
- **License expiration –** Displays a date or the word “permanent,” depending on the type of license purchased.
- **Maintenance expiration –** An expiration date. It is not possible to buy perpetual maintenance.
- **User –** Shows the user ’s name if the license is in use.
- **Host –** Shows the machine where the license is being used.

License Files

Files for dedicated licenses (as opposed to floating licenses) are stored in the OPNET_license directory.

Note—When it is time to install a new version of the software, do not delete the license files.

Starting License Manager

The License Manager can be accessed from the Start Menu or the System Editor. If you need to perform license maintenance, it is better to access the License Manager from the Start Menu (JCSS does not need to obtain a license).

Note—Please contact NETWARS CM to register your license.

- To access License Manager from the Start Menu, simply select it from the **Start/Programs/NETWARS/License Manager** menu option.
- To access License Manager from the System Editor, select it from the License menu.

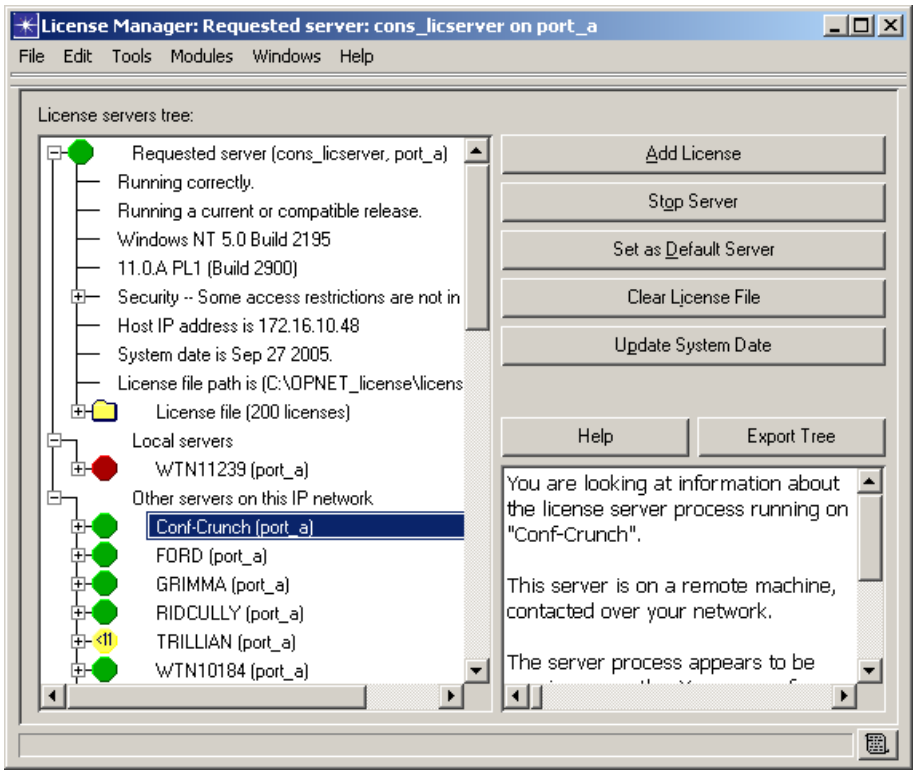


Figure 2-21 License Manager: Requested Server dialog box

The License Manager contains details about the license server that you requested during the installation of JCSS. The License Manager also provides a Help menu that can be accessed by clicking the **Help** button.

Adding a License

1) Click the **Add License** button on the License Manager to display the Choose Transaction Method dialog box.

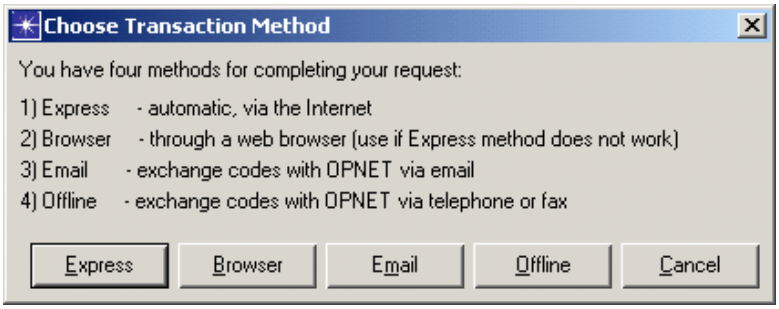


Figure 2-22 Choose Transaction Method dialog box

There are four transaction methods:

- **Express –** Automatic via the Internet (Requires authorized access.)
- **Browser –** Through a Web browser if the Express method does not work.
- **Email –** Exchange Codes with OPNET via Email.
- **Offline –** Exchange codes with OPNET via Telephone or Fax.

Product Modules

License > Product Modules: Select active modules. Product Modules refer to additional licensed OPNET functionality. For example, JCSS uses the Terrain Modeling Module GUI (TMMGUI) and Multi-Vendor Import (MVI) product options, which are licensed under the NETWARS license. (JCSS can also make use of NetDoctor and Simulation licenses to further enable the JCSS-resident functionality.)

If a module is deactivated using this option (by deselecting the checkbox next to a module name) the corresponding features of that module will not be active in JCSS. For example, the Terrain menu will not be available in JCSS if the TMMGUI module is deactivated.

IER Database Menu

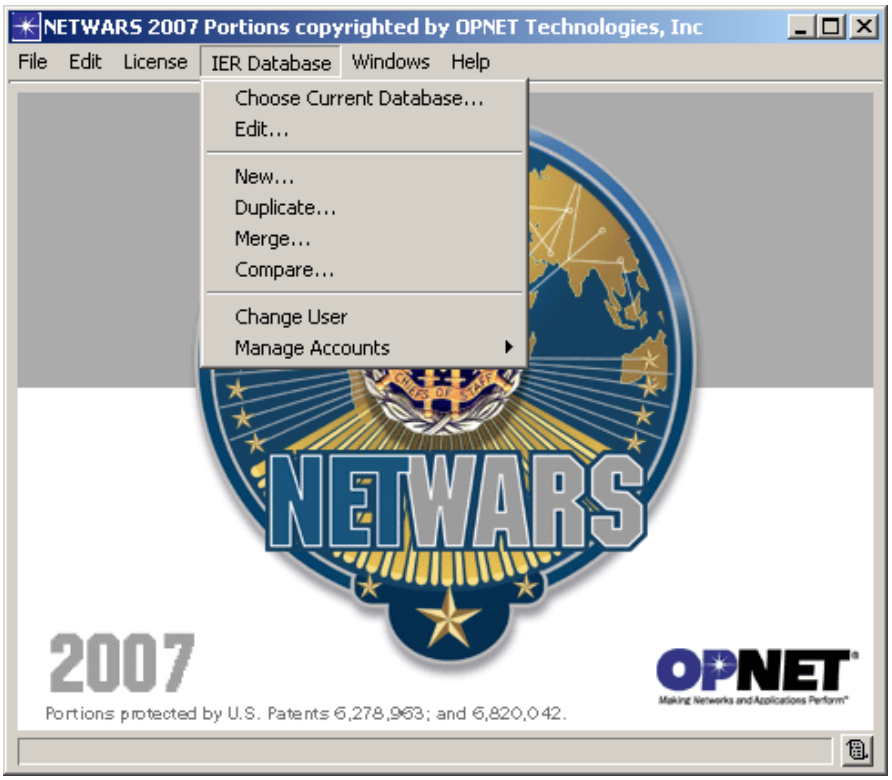


Figure 2-23 System Editor's IER Database menu

Database Selection

By default, JCSS will install a database service. The name of the service would be the hostname of the system followed by **\NW2004_1**. JCSS allows users to select between multiple databases on the local server or to select any network accessible database server as a datasource.

Choose Current Database

IER Database > Choose Current Database: Choose the current database server and database within that server for use as a default data source for the IER Editor. Each selected server requires login for a user account registered at that database server, so you may be prompted for a login when changing the selected server.

Edit **IER Database > Edit...**: View and edit IERs in the IER Database Editor table. This table will contain the current working set of IERs, which you may view, insert into, delete from and modify. The current working set contains IERs that have been queried from the database, imported from text files, or entered manually.

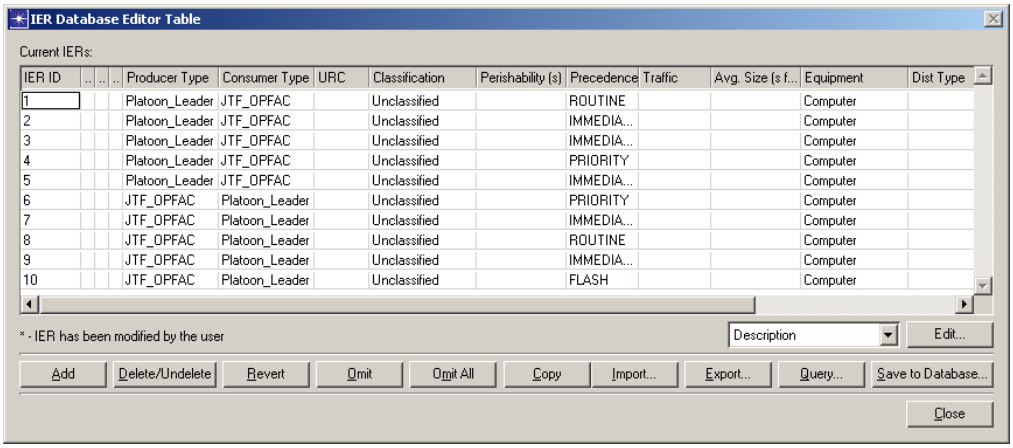


Figure 2-24 IER Database Editor Table

New **IER Database > New...**: Create a new database on the specified server. Creating a database requires Administrator access, so you will be prompted for an administrator login. Logging in as the administrator also selects the destination server.

Database names cannot contain spaces or duplicate an existing database name on the selected server. Differences in capitalization are not enough to make database names unique; for example, Database1 is considered the same as both database1 and DATABASE1.

Duplicate **IER Database > Duplicate...**: Copy/duplicate the current default database. If no default database has been selected, you will be required to select one. You will also be required to provide a Database Administrator ID and password for the destination server. A new database will be created, and its name must be unique as described above. The newly created database will contain copies of all of the records in the source.

Merge **IER Database > Merge...**: Merge one database into another. All of the records from the source database will be copied into the destination database. Any existing records in the destination database whose identifier fields match records in the source database will be overwritten by the source records.

Compare **IER Database > Compare...**: Produce a record-by-record comparison report for two selected databases. The resulting report can be in text or HTML format.

Change User **IER Database > Change User**: Clear all retained user passwords; requires you to login again to the default database server.

Manage Accounts

IER Database > Manage Accounts > *(option)*: Database Administrators have the authority to create databases or accounts, or to modify passwords or account access permissions. Administrator access allows control over any database or user account on a specified server.

Database users may read the contents of database tables and store changes to them. Users only have access to selected databases. When users login to a server, they will be given the list of databases on that server that they are allowed to access.

User accounts may differ for each database server, so switching servers may require a new login.

Manage Accounts > Admin

IER Database > Manage Accounts > Admin: Manage JCSS administrator accounts.

Manage Accounts > User

IER Database > Manage Accounts > User: Manage JCSS user accounts. rant or deny access to designated servers and databases, add and delete users, and change user passwords via the Database User Account Management dialog box shown below.

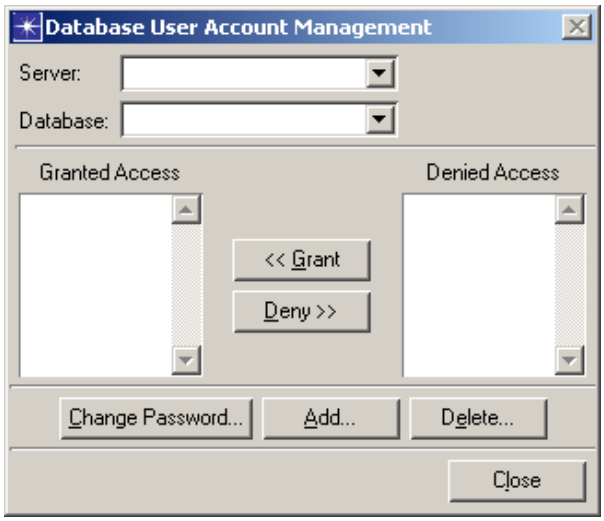


Figure 2-25 Database User Account Management dialog box

Password Retention

User logins are retained for the duration of the session until you select to change the user. For typical operations, password retention will minimize repetition of password prompts. JCSS will not retain administrator passwords. Each time a user conducts an administrator operation, JCSS will ask for the administrator login.

Windows Menu

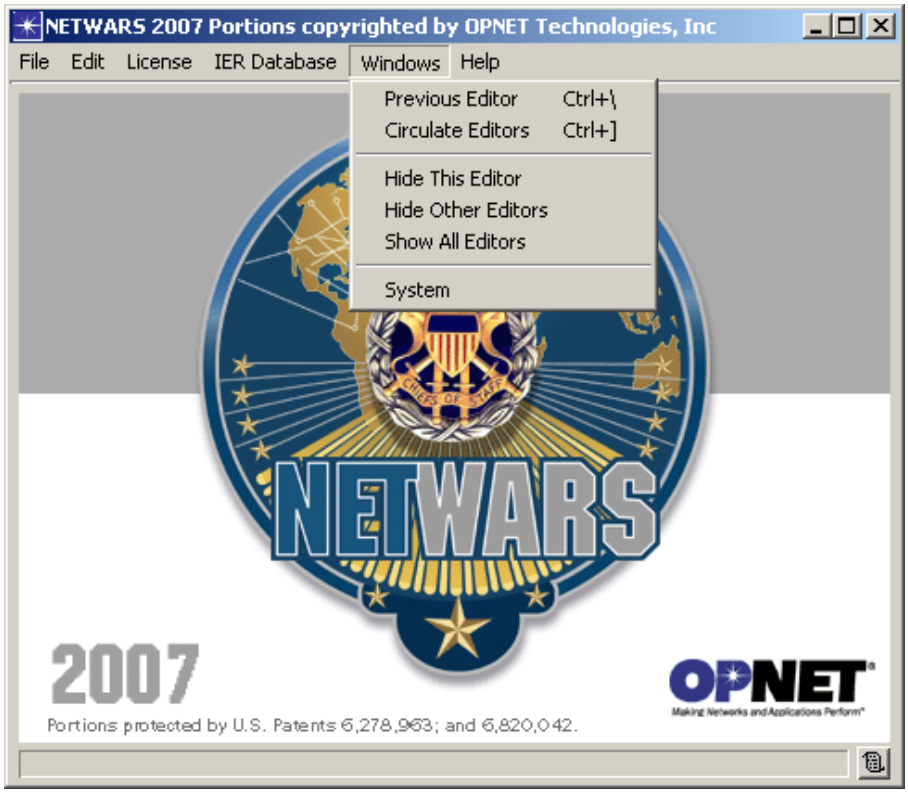


Figure 2-26 System Editor's Windows menu

Previous Editor — **Windows > Previous Editor**: Make the previously displayed editor the current editor.

Circulate Editors — **Windows > Circulate Editors**: One at a time, make each open editor the current editor.

Hide This Editor — **Windows > Hide This Editor**: Hide the current editor. If another editor is open, that editor is made the current editor.

Hide Other Editors — **Windows > Hide Other Editors:** Hide all open editor windows except the current one.

Show All Editors — **Windows > Show All Editors:** Re-display any hidden editors.

Help Menu

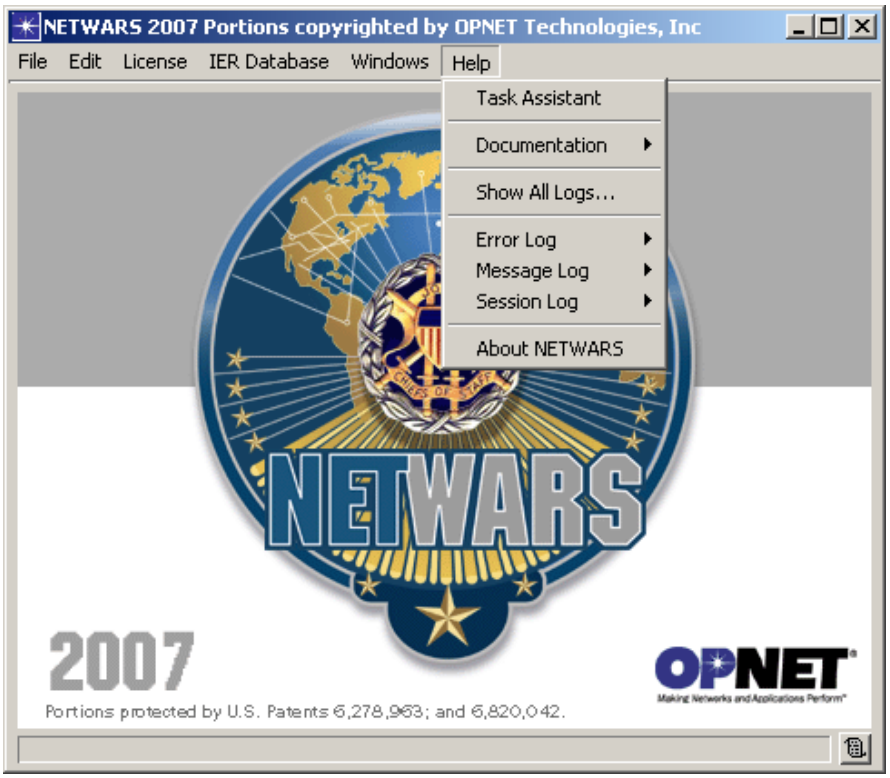


Figure 2-27 System Editor's Help menu

Task Assistant **Help > Task Assistant**: Display a list of workflows available to the planners. A workflow is simply an outline of steps to guide the planner through the flow of tasks associated with the particular mission being addressed.

The Task Assistant dialog box is shown below. The treeview on the left hand side of the dialog box shows the list of workflows available, and the explanation for those is provided in the pane on the right.

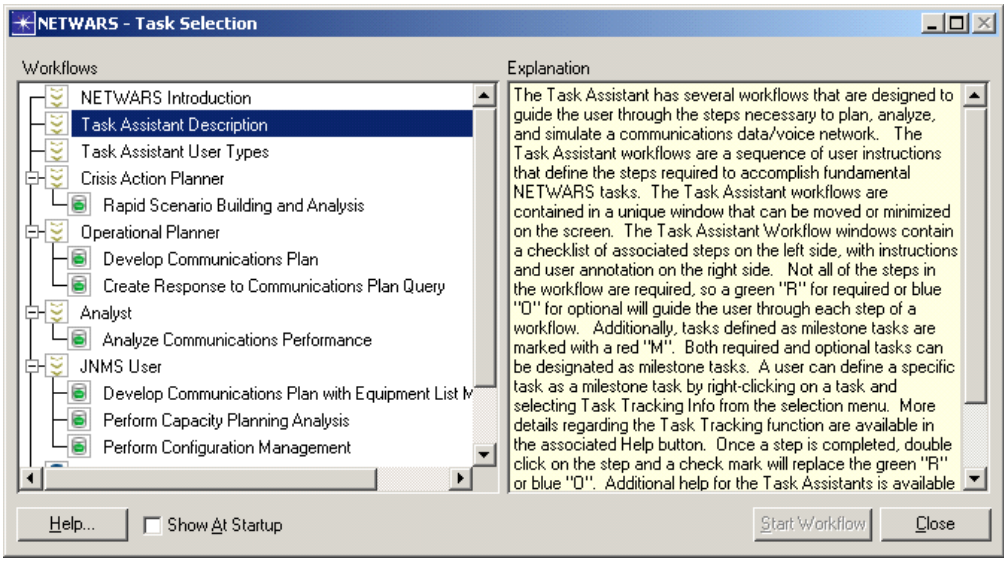


Figure 2-28 Task Assistant dialog box

The list of workflows displayed in the Task Assistant dialog box are organized based on categories of user tasks and most recently used workflows.

1) To start a workflow, first select the workflow and then use either the **Start Workflow** button or simply double-click the workflow of interest.

2) To review the items within each workflow, click the plus sign to the left of the workflow category. This expands that category as shown below.

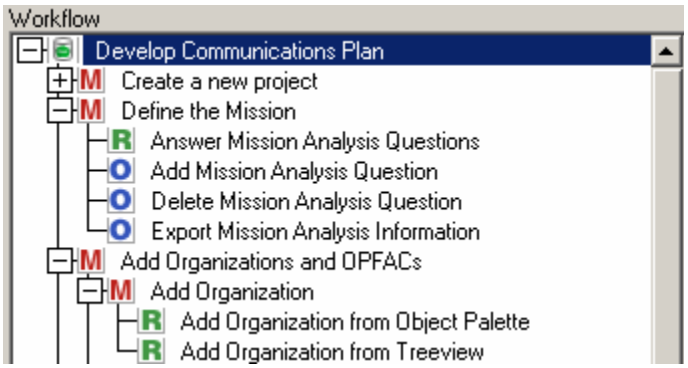


Figure 2-29 Expand a workflow category

Assume that you have decided to develop a communications plan. You would select Develop Communications Plan by double-clicking the icon to the left of the text. This calls the Develop Communications Plan task.

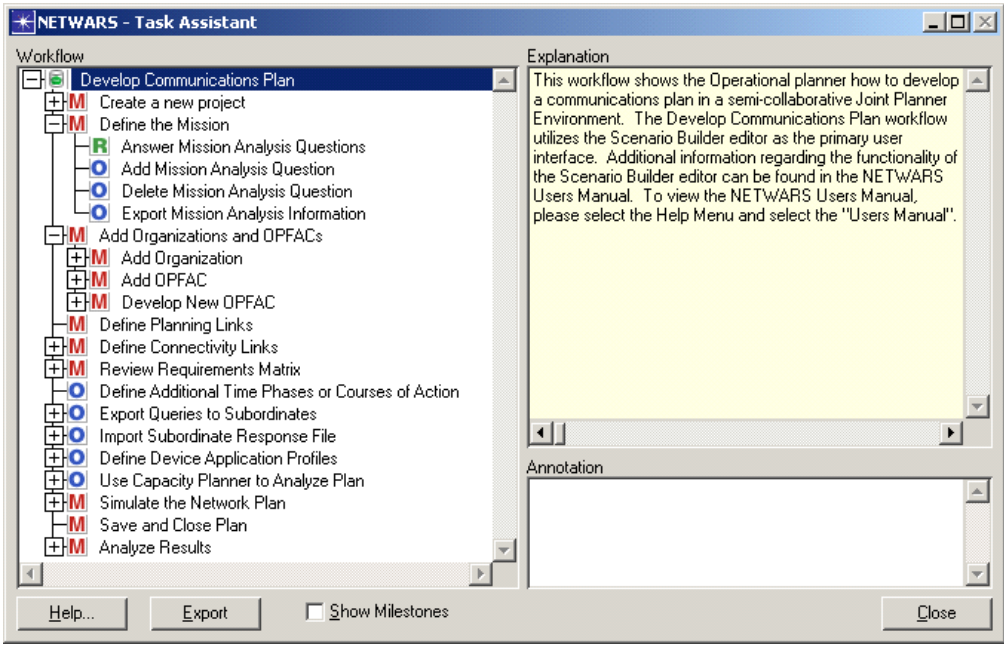


Figure 2-30 Develop Communications Plan task

In the figure above, each line item of the workflow has a letter to the left of the text indicating whether it is a milestone (M), an optional (O) or a required (R) task. In addition, many of the tasks can be expanded, by clicking the plus sign to reveal details on what steps are necessary to complete each task. Additionally, each line of the workflow is supplemented by details on how to complete that task in the Explanation pane located on the right side of the Task Assistant dialog.

1) If you do not find the workflow of interest, choose the **Browse for Workflow** option under the Recent Workflows in the Task Assistant dialog box to locate the desired workflow.

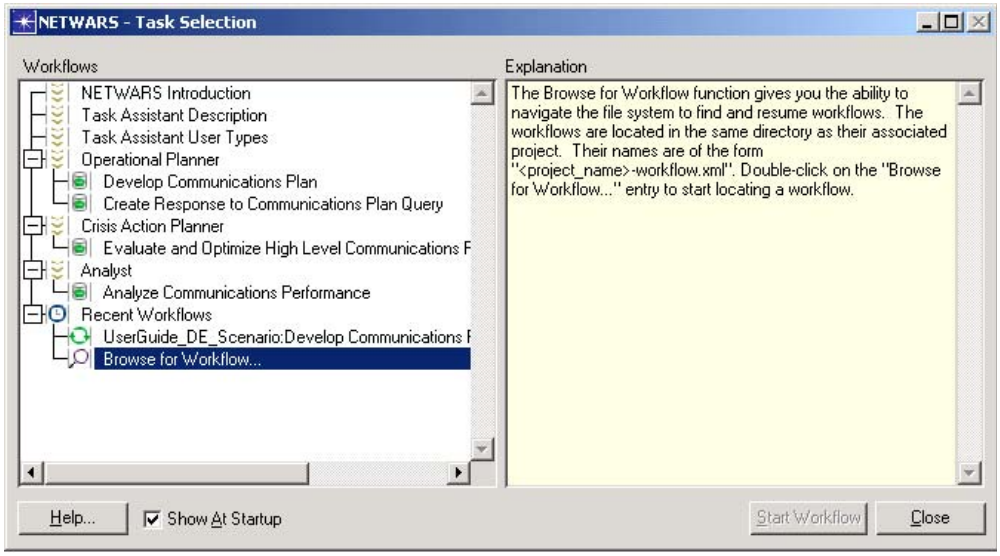


Figure 2-31 Task Assistant browse for workflow

2) If you chose not to use the Task Assistant workflows, exit by clicking the **Close** button.

Task Tracking

This feature, available from the Task Assistant, provides you with the capability to track timeline, dates due, milestones, and percent done for the planning tasks associated with the plan being developed.

1) Task tracking information is available from the Task Tracking dialog box. Right-click on a particular task or on the workflow in the Task Assistant, and select **Task Tracking Information**. If you click on the workflow name and select **Task Tracking Information**, the workflow dialog box launches.

2) Check the **Edit Workflow Time Information** and enter the number of days for the entire workflow.

3) Click **OK** to save changes and close the dialog box. The figure below shows workflow time information.

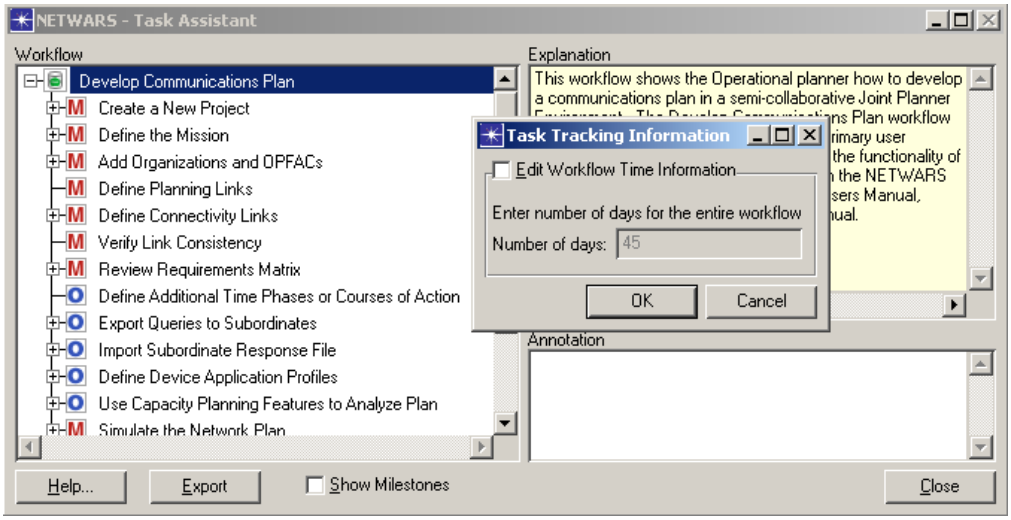


Figure 2-32 Entering number of days for the entire workflow

4) Right-click on a task and choose **Task Tracking Information** to launch the Task Tracking dialog box, as shown below.

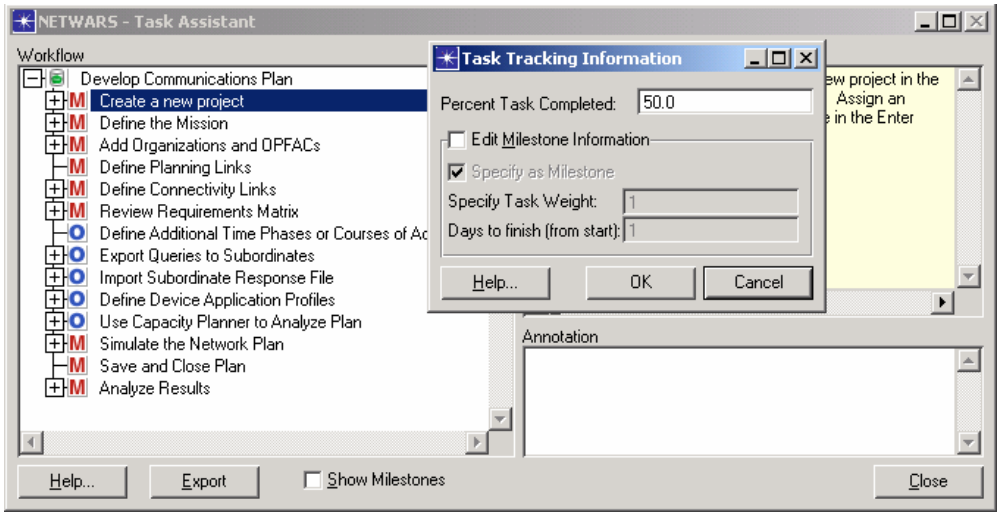


Figure 2-33 Task Tracking dialog box

The Task Tracking dialog box shows the milestone information, and the percent task completion. The Task Tracking dialog box provided above shows that the task Create a New Project is a milestone task (also known from the “M” icon for this task). The task is currently 50% complete. It has a task weight of 1, and days to finish (from start of the workflow) is 1.

5) To edit this milestone information, check **Edit Milestone Information**. You can then change task weight and days to finish.

6) Update the percent task completed accordingly.

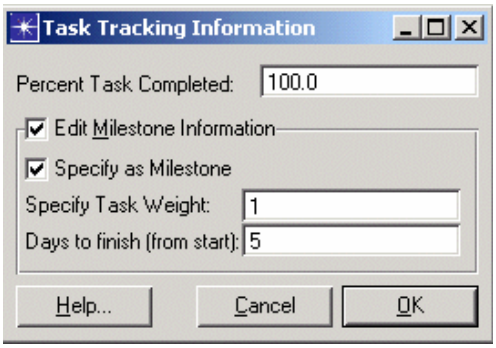


Figure 2-34 Editing milestone information

A particular task can be completed by setting its task completion percent to 100 (from the Task Tracking dialog box, as shown above), or by double-clicking the task from the Task Assistant. When a milestone task is completed, the icon in the Task Assistant dialog box changes from “M” to display a check mark.

In order to complete a milestone task, all of its required and milestone sub-tasks need to be complete. If they are not complete, you will be prompted to do so. Similarly, a completed task can be marked incomplete, either by entering a value for task completion percent that is less than 100, or by double-clicking the completed task from the Task Assistant dialog box.

In the figure below, task Define the Mission is complete (as indicated by the check-mark), and percent task completion is set to 100 in the Task Tracking Information dialog box.

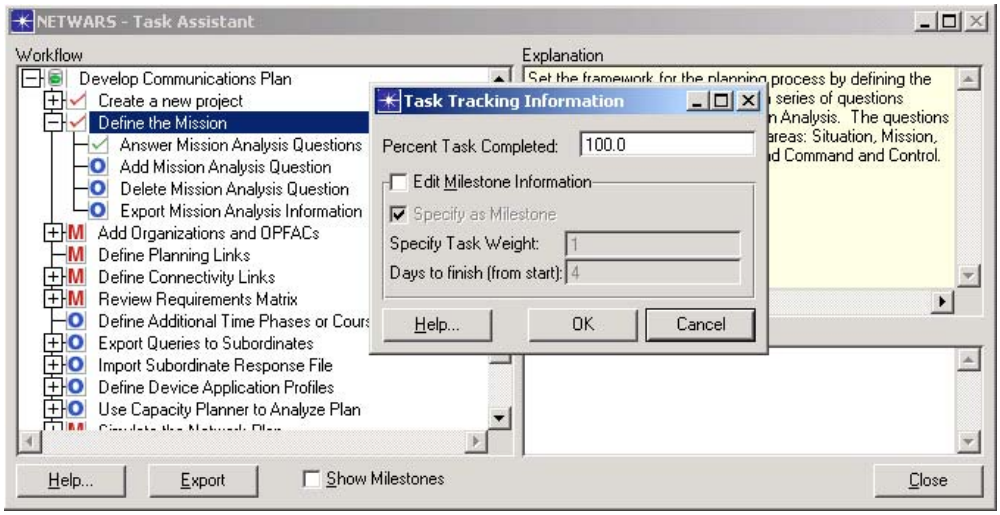


Figure 2-35 Completing a task

If a milestone is overdue, it is flagged with a red “X”.

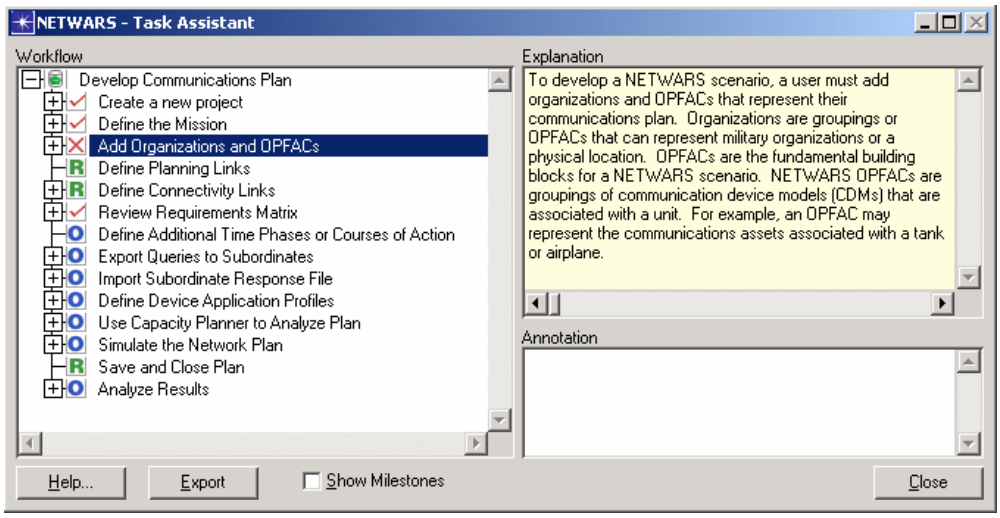


Figure 2-36 Overdue milestone

1) Toggle the milestone display on or off using the **Show Milestones** checkbox at the bottom of the Task Assistant. This will display/hide the completion percentage and days remaining for the milestone tasks, and for the entire workflow. The completion percentage for the entire workflow is calculated as a function of the completion percentage of each of the milestone tasks, and their relative task weights.

2) By clicking the **Export** button, you will be prompted for a filename and location to save the task information as a tab-delimited text file. This file can be opened in MS Project (choose **Default Task Information** for the **Import Mapping** when opening the file).

Documentation

Help > Documentation > (*option*): Access the JCSS documents discussed below:

Documentation > User’s Manual

Help > Documentation > User’s Manual: Open the *JCSS User’s Manual* using Adobe Acrobat Reader. Acrobat Reader software can be optionally installed with JCSS. The *JCSS User’s Manual* is a guide that covers basic aspects of JCSS operation and use. It contains figures that provide visual examples of the JCSS interfaces, step-by-step procedures, and other important JCSS features and concepts.

Documentation > Technical Reference Manual

Help > Documentation > Technical Reference Manual: Open this document, the *JCSS Technical Reference Manual,* using Adobe Acrobat Reader.

Documentation > Software Release Bulletin (SRB)

Help > Documentation > Software Release Bulletin (SRB): Open the *JCSS Software Release Bulletin* using Adobe Acrobat Reader. Acrobat Reader software can be optionally installed with JCSS. The *SRB* contains release notes for the current version of JCSS.

Documentation > NETWARS Acronyms

Help > Documentation >NETWARS Acronyms: Open the *NETWARS Acronyms* table using Adobe Acrobat Reader. Acrobat Reader software can be optionally installed with JCSS. The *NETWARS Acronyms* table contains definitions for acronyms commonly used with JCSS.

Documentation > IT Guru Documentation

Help > Documentation > IT Guru Documentation: Open the complete IT Guru document set. IT Guru is an OPNET commercial-off-the-shelf (COTS) product that models the behavior of networks—many JCSS features are based on IT Guru features. In this *JCSS Technical Reference Manual*, many references will ask you to access the IT Guru documentation for further details on certain features.

The IT Guru documentation contains a global index and a search query feature that enables you to quickly find needed information.

Show All Logs

Help > Show All Logs...: Launches the Log Viewer.

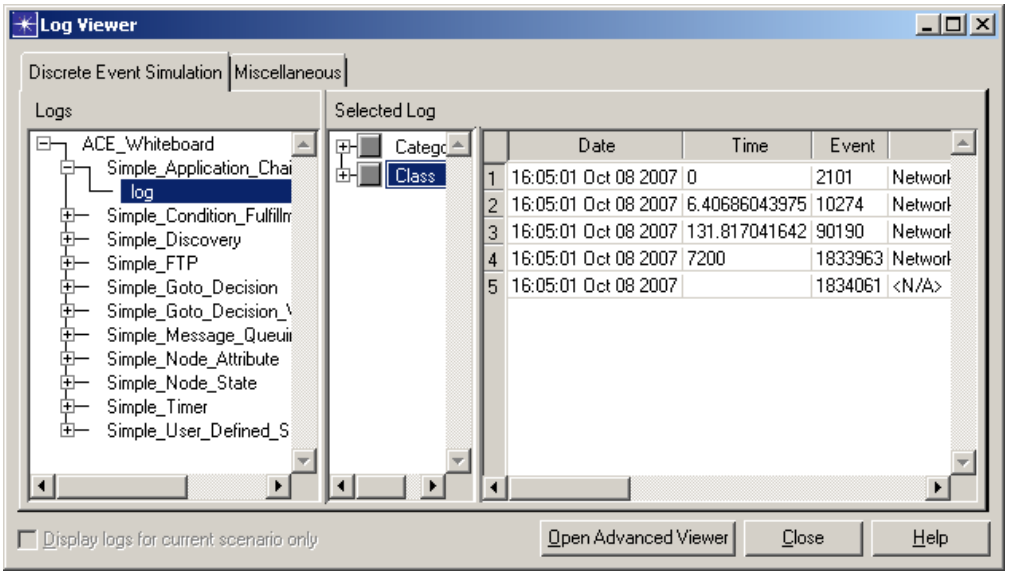


Figure 2-37 Log Viewer

Error Log

Help > Error Log > (*option*): If errors occur during the operation of JCSS, they are logged in this error log file.

Error Log > Open

Help > Error Log > Open: Using this menu option, you can open the error log file to view the error messages in a text editor.

Error Log > Clear...

Help > Error Log > Clear...: Clear the error log file of the selected range of error messages.

Message Log

Help > Message Log > (*option*): Status updates that monitor the performance of JCSS are logged in this message log file.

Message Log > Open — **Help > Message Log > Open:** Using this menu option, you can open the message log file to view the status messages in a text editor.

Message Log > Clear — **Help > Message Log > Clear:** Clear the message log file of all status messages.

Session Log — **Help > Session Log > (*option*)**: Basic file operations during the current session are logged in this session log file.

Session Log > Open — **Help > Session Log > Open:** Using this menu option, you can open the session log file to view the session messages in a text editor.

Session Log > Clear... — **Help > Session Log > Clear...:** Clear the session log file of the selected range of session messages.

About NETWARS — **Help > About NETWARS**: View information about the OPNET core software version, the models version, and copyright information.

3 Scenario Builder

The Scenario Builder is the JCSS interface for creating, modifying, and saving projects and scenarios; it allows you to develop a complete communications plan.

The Scenario Builder is also used to evaluate and optimize network performance. Given a network and the traffic, it runs the optimization and comes up with suggestions for optimal link and network capacities.

Additionally, this interface analyzes the results of a simulation session. It converts output from a simulation session into data that is viewable and grouped into Measures of Performance (MOPs).

Accessing the Scenario Builder

When you choose to either open an existing project in JCSS or create a new project, the Scenario Builder displays.

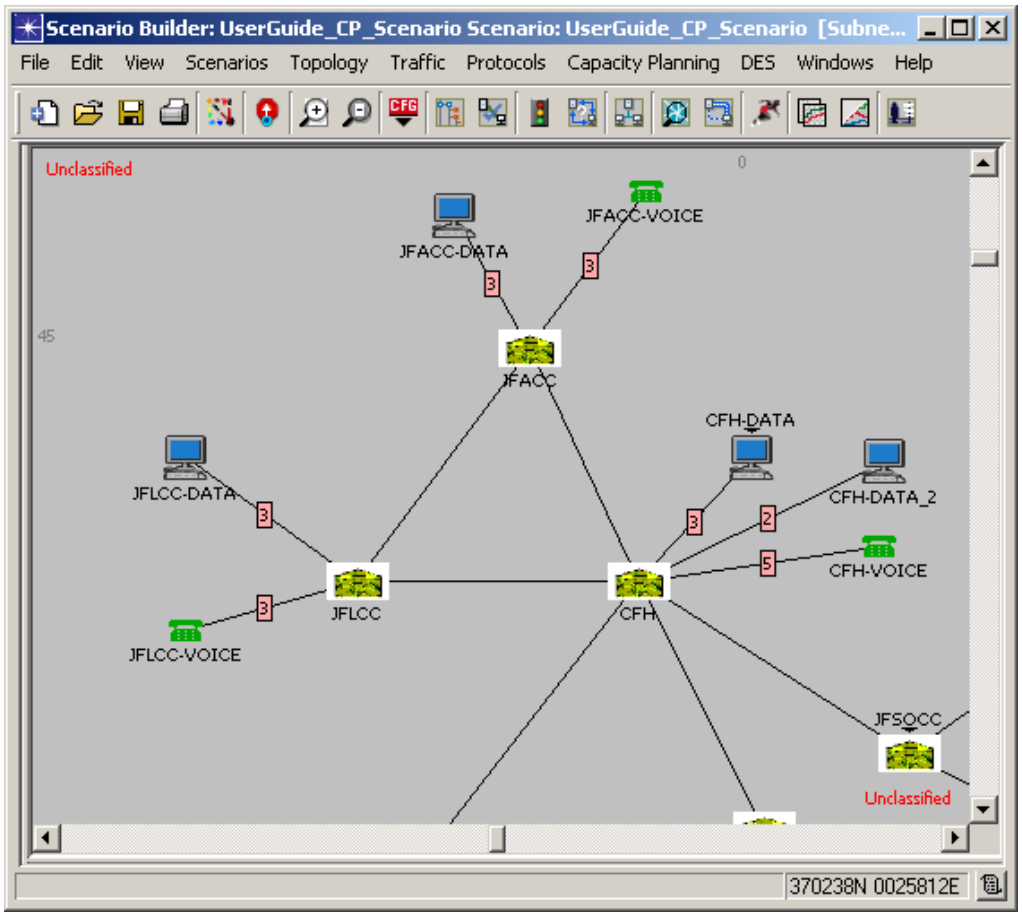


Figure 3-1 Scenario Builder

File Menu

New Project

File > New Project: Opens a new Scenario Builder and displays the Create Project dialog box. Create a new project, the first step in the Lead Planner's workflow. You can simply name and launch a generic project, or use the New Project Wizard to select and set the area of operations, top-level units (including their locations and templates), an equipment list for each organization, an owner for each unit, and the types of equipment to use in the new project.

1) Specify the names for the project and the initial scenario in the Create Project dialog box.

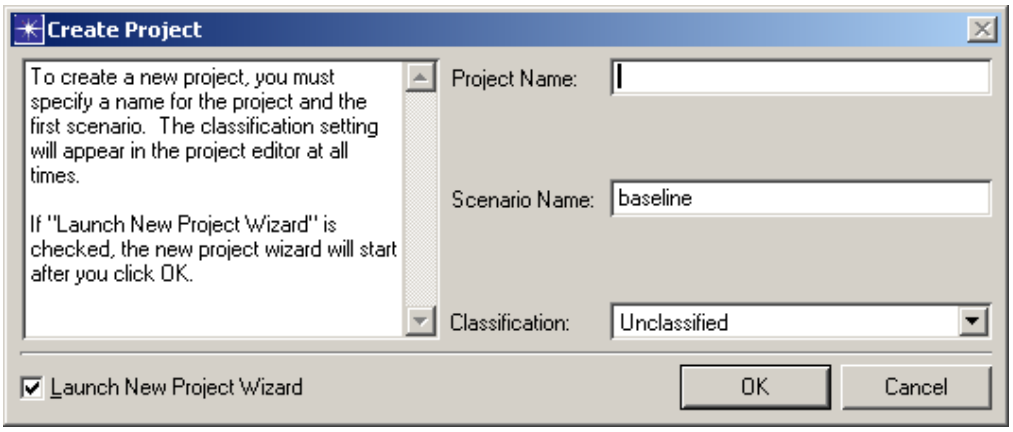


Figure 3-2 Creating a new project

2) Select a classification level for the project in the Create Project dialog box.

3) Click **OK**. If **Launch New Project Wizard** is not checked, then JCSS creates a generic project using the names you have designated. If **Launch New Project Wizard** is checked, then JCSS displays a sequence of dialog boxes that enable you to make selections specific to your new project.

Note—The names for both the project and the scenario cannot contain any spaces or special characters. In addition, the scenario name cannot be the same as any existing OPFAC; it must be unique.

Open Project

File > Open Project or click the **Open an Existing Project** toolbar button: Displays the Open Project dialog box, and then opens the selected project in a new Scenario Builder.

1. Double-click on the project folder of interest and select the corresponding .prj file.

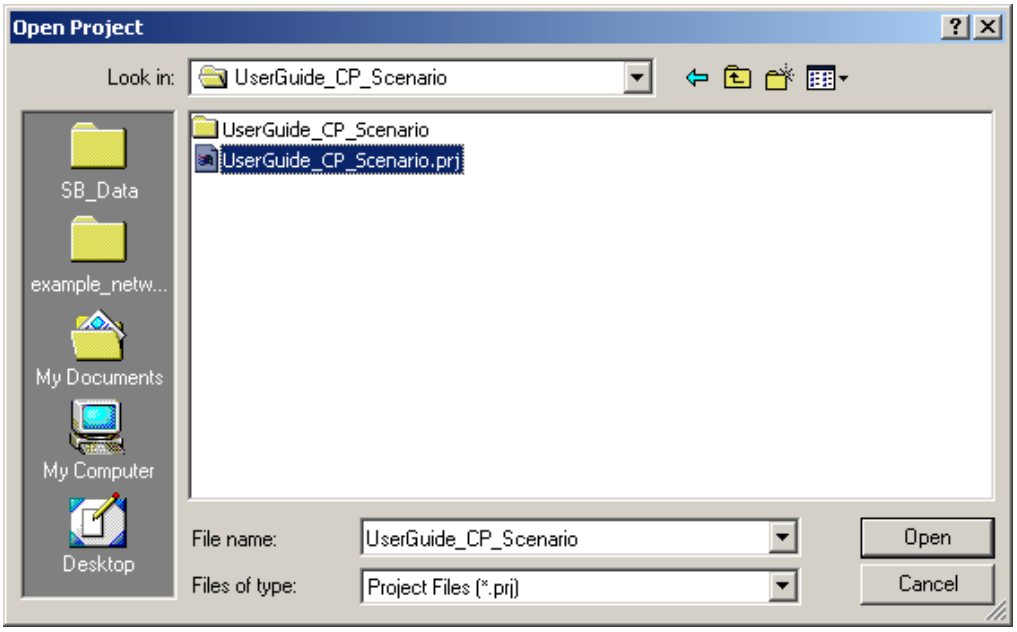


Figure 3-3 Opening a Project in Scenario Builder

To create a new project, click the **Create a New Project** button.

Save Project/ Save Project As

File > Save Project or **File > Save Project As**: Save changes to existing projects in the Scenario Builder by using either the Save or Save As option.

1) To save the changes you made to an existing project in the Scenario Builder, select the **Save** option.

OR

1) To save the changes you made as a separate project in the Scenario Builder, select the **Save As** option.

2) Specify the desired name in the subsequent dialog box, and then click **OK**.

Open Subordinate Query

File > Open Subordinate Query: Displays the Open Subordinate Query dialog box, and then opens the selected subordinate query file in a new Scenario Builder.

1) The Subordinate Planner will be prompted to select the appropriate file to be edited. Select the file by double-clicking on it.

2) The project that you select will then open in a new Scenario Builder. The green “e” icon on top of the units indicates that the Senior-level Planner has marked these units to be edited by the Subordinate Planner. Other units that have a red lock icon on top indicate that the Subordinate Planner may only view these units and cannot edit them. Lock units are hidden by default upon opening.

3) To view all units, select **View > Filter > Restore Full View**.

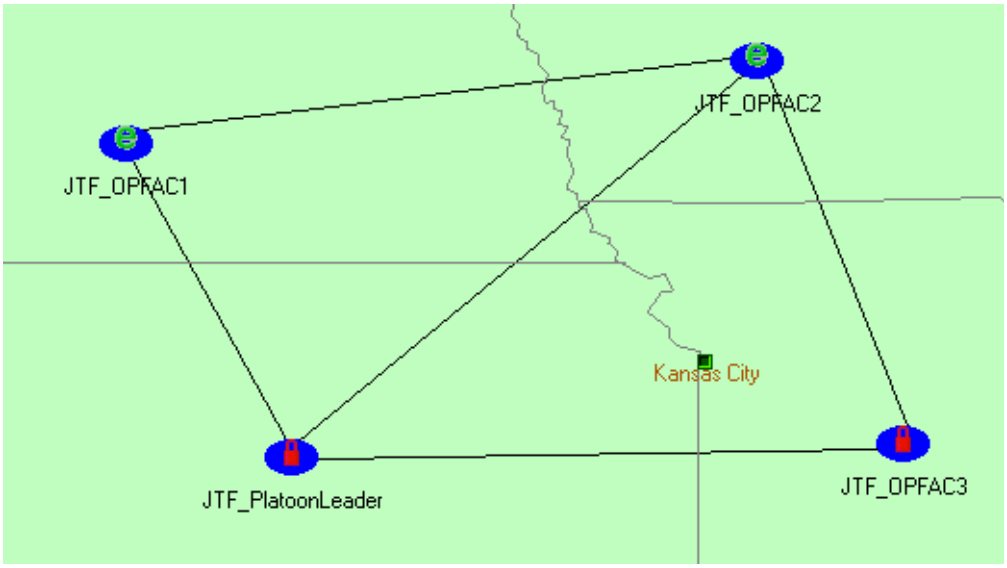


Figure 3-4 Opened Subordinate Query

The Subordinate Planner also has the capability to export subordinate query files, allowing this process to be repeated through many levels of hierarchy.

Save Subordinate Query

File > Save Subordinate Query: Save the Subordinate Planner’s changes to the subordinate query file.

Close

File > Close: Close the open project or subordinate query file.

1) You will be prompted to save your project. Choose the desired option.

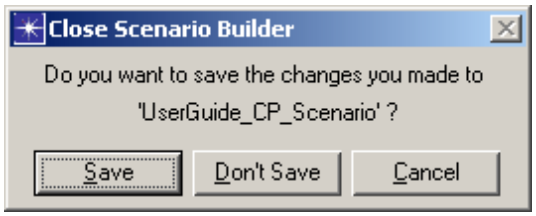


Figure 3-5 Closing a Project

The selected project and its Scenario Builder window close.

Generate Scenario Briefing

File > Generate Scenario Briefing: Export scenario information directly to a PowerPoint (.ppt) file. Customize the briefings by selecting desired components from a set of elemental briefing components.

A graphical user interface (GUI) is provided to enable you to choose the briefing components that you want to include, and then generate the briefings.

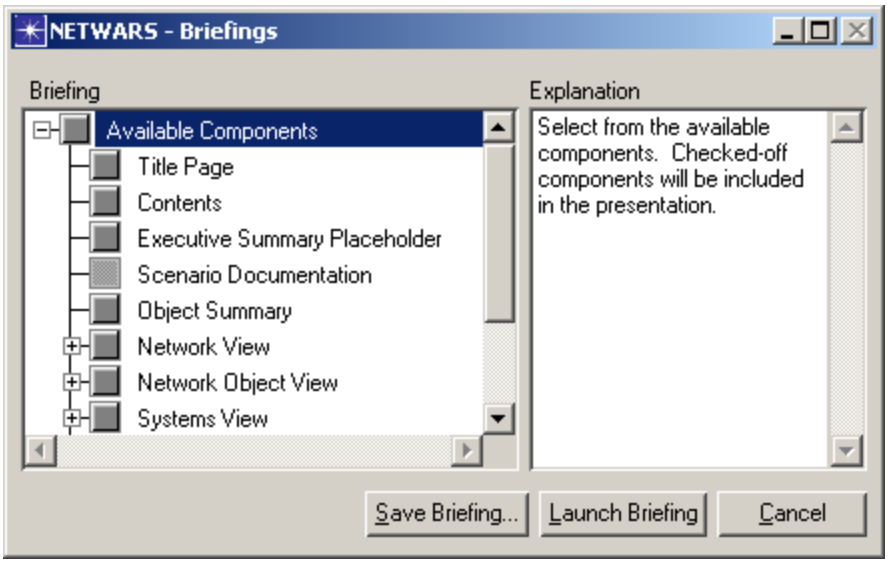


Figure 3-6 Briefings dialog box

1) Choose from the available components shown on the left hand side of the dialog box. The explanation of the selected components displays on the right hand side text box. Some of the components may not be available and will be shown disabled in the dialog box accordingly.

2) After selecting the components, choose **Launch Briefing** or **Save Briefing**.

   a) **Launch Briefing** launches the PowerPoint application and opens the briefing generated.

   b) **Save Briefing** prompts you to specify a filename and location for saving the briefings PowerPoint file.

Page Setup

File > Page Setup: Displays the Print Setup dialog box from which you can set printing options.

Print Scenario

File > Print Scenario: Prints the contents of a JCSS scenario. A preview of the scenario displays as it will appear when it is printed.

When the scenario is printed, the contents of each subnet are printed on a separate page. Every page has a default header, which is the hierarchical name of the subnet.

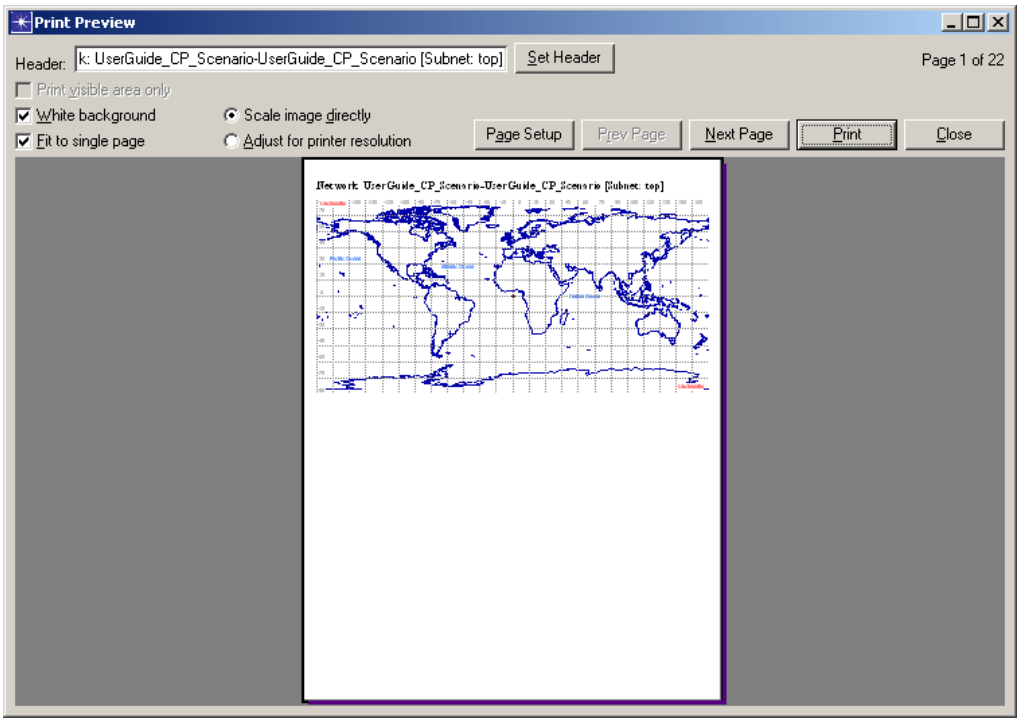


Figure 3-7 Print Preview dialog box

1) From the preview, browse through the pages and make choices about the following:

   a) Set the header to be used on each printed page

   b) Use the background color as it appears on the workspace, or click the **White background** checkbox to use a white background.

   c) If you check **Fit to single page**, the contents of a subnet print in one page; if not, each subnet prints across multiple pages. The total number of pages displays at the top right corner of the preview window.

   d) Choose to either scale the image directly, or adjust for printer resolution.

   e) Click **Page Setup** if you want to select the mode (landscape or portrait), paper size, etc.

2) Click **Print**, and select the desired printer.

Print Graphs

File > Print Graphs: After you have defined and displayed graph panels, you can print the visible graphs.

Refresh IER Text Files

File > Refresh IER Text Files: IERs are queried from various sources; text files are one of them. Text files are queried when you view the IERs for an OPFAC or run a simulation. If changes are made to the text files, they are not immediately reflected in the scenario that is currently loaded in the Scenario Builder. In such cases, you can use this menu option to force the list of IERs to be refreshed.

Manage Model Files

File > Manage Model Files > *(option)*: This submenu lists options which allow you to manage model files.

Manage Model Files > Delete Model Files

File > Manage Model Files > Delete Model Files: Displays a list of all models from which you can select to delete files.

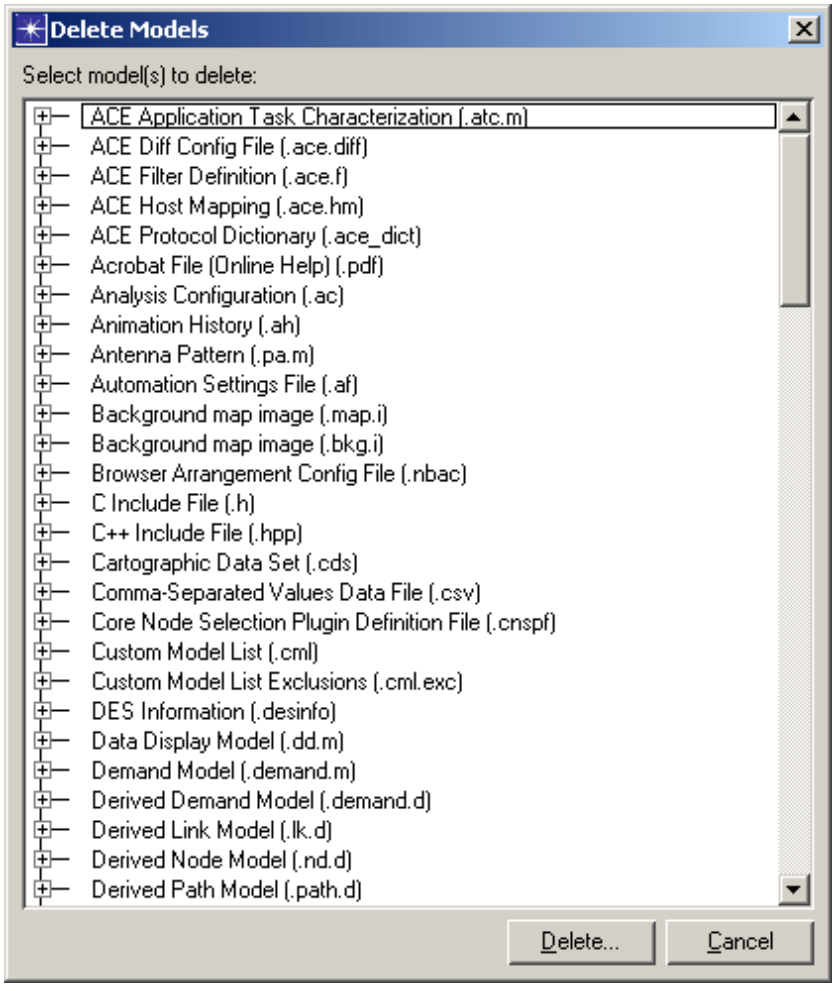


Figure 3-8 Delete Models dialog box

Manage Model Files > Add Model Directory

File > Manage Model Files > Add Model Directory: Displays the Directory Browser in which you can create a new directory for models.

Manage Model Files > Refresh Model Directories

File > Manage Model Files > Refresh Model Directories: JCSS maintains a list of folders called the model directories (or mod_dirs for short) where models are stored. Models include OPFACs, organizations, projects and scenarios, icon databases, and device models. When you move models around on the hard drive using Windows Explorer, the software does not recognize the change. To make the software aware of the changes, use this option.

Recent Projects

File > Recent Projects: This submenu lists recently opened projects for easy re-opening.

Package Project Files

File > Package Project Files: Provides you with the ability to package and import JCSS data files. It serves as a convenient mechanism to exchange JCSS data with other users, and with NETWARS Support.

There are different options for including different files in the package. These various options are available from the Package Project Files menu. Four different options are available for packaging the files; and there is one menu item for importing packaged files.

A zipped archive is created as a result of choosing one of the package menu items. The zipped archive contains the following files for the various options:

- **Project –** Project, scenario, subordinate query, traffic, trajectory, OPFAC and organization files, simulation model files (if they exist.)
- **Project with Simulation Results –** Project files as indicated in the Project Files option and simulation results files.
- **Project for NETWARS Support –** Project files as indicated in the Project Files option and other files needed by NETWARS Support (env_db files, various log files etc.).
- **Custom OPFACs and Organizations –** All OPFAC and organization files under the Custom OPFACs and Custom Organizations folders.

To import package file, choose **Import** from the Package Project Files menu. A file chooser is presented to browse for the zipped archive containing JCSS files. After you select the archive file to be imported, a dialog box with the different import options displays (see figure below.)

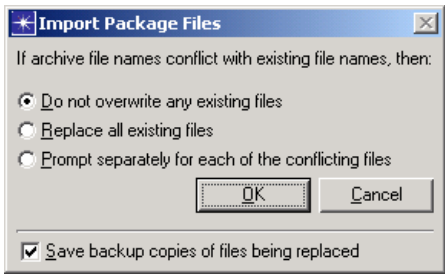


Figure 3-9 Import Package options

Backup copies of files being replaced are created by default. You can turn that off by un-checking the check box in the above dialog box.

If you choose to be prompted for each file separately, then for every conflicting file, a dialog box displays. From this dialog box, you can choose to not overwrite the existing file, to replace the existing file, or to quit the import process.

Info-ZIP's Zip and UnZip[1] are used to create and import the zip file.

Exit **File > Exit**: Shut down the entire JCSS program. If there are editors that are open, you will be prompted to save the contents of those editors. When all open editors have been acknowledged and closed, the software is shut down.

Edit Menu

Undo **Edit > Undo**: Undo editing operations.

Only the following actions can be undone:

- Adding a unit (includes OPFAC and organization), device, or association (includes intra OPFAC link, inter OPFAC link, satellite link, broadcast network, relationship and circuit) from the palette or library.
- Moving a unit, device, association, or any group of these objects in the workspace.
- Deleting a unit, device, association, or any group of these objects in the workspace or using Scenario Builder treeview.
- Modifying attributes on devices; all attribute changes made while the Edit Attributes dialog box is open will be considered a single action.

Some actions may prevent certain, or all, prior actions from being undone (for example, saving a project file.) Immediately following a save to disk, there will be no undoable actions.

There is no limit on the number of actions that can be undone; however, the actions can only be undone in the reverse order in which they were performed.

Redo **Edit > Redo**: Redo the undone operations. The actions can only be redone in the reverse order in which they were undone, which means that the last undone action will be the first one to be redone.

1. Copyright (c) 1990-1999 Info-ZIP. All rights reserved. "Info-ZIP" software is provided "as is," without warranty of any kind, express or implied. In no event shall Info-ZIP or its contributors be held liable for any direct, indirect, incidental, special or consequential damages arising out of the use of or inability to use this software. Info-ZIP retains the right to use the names "Info-ZIP," "Zip," "UnZip,""WiZ," "Pocket UnZip," "Pocket Zip," and "MacZip" for its own source and binary releases.

Any action (editing operation which includes add, delete, move, modify) performed after performing **Undo** will clear the **Redo** stack, and the undone operations cannot be redone.

Cut

Edit > Cut : Cut selected, supported objects on the workspace to the clipboard, and remove them from the workspace. If unsupported objects are attached to a supported object, then the unsupported objects are also removed from the workspace but not maintained on the clipboard.

Supported Objects:

- Organizations
- OPFACs
- Devices
- External links
- Internal links
- Satellite links
- Broadcast networks
- OPNET demand objects

Unsupported Objects:

- Radio links
- Circuits
- Relationships
- OE
- Configuration OPFAC

Copy

Edit > Copy: Copy selected, supported objects on the workspace to the clipboard.

Supported Objects:

- Organizations
- OPFACs
- Devices
- External links
- Internal links
- Satellite links
- Broadcast networks
- OPNET demand objects

Unsupported Objects:

- Radio links
- Circuits
- Relationships
- OE
- Configuration OPFAC

Paste **Edit > Paste**: Paste all objects on the clipboard to the workspace.

Delete **Edit > Delete**: Delete the objects selected on the workspace.

1) Select the objects that need to be deleted. To select multiple objects, press the **<Ctrl>** key on the keyboard while clicking on the objects.

2) Once the objects are selected, use this menu option or use the **<Del>** key on the keyboard to delete them.

3) Confirm the deletion.

Select All **Edit > Select All**: Select all the objects in the current subnet. Once the objects are selected, perform edit operations (like move, delete, etc) on these objects as needed.

Find Node/Link **Edit > Find Node/Link**: You can search for an object (such as a node, subnetwork, link, or path) based on its name (part or all of the name.) Every object whose name contains the string you specified is selected and listed. Or you can leave the search field blank to find all objects in the network.

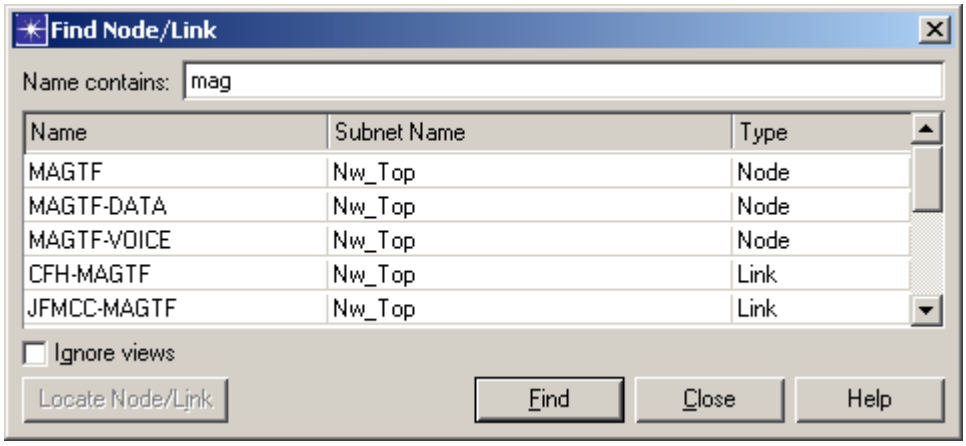


Figure 3-10 Find Node/Link dialog box

1) Enter a text string in the **Name contains** field to search for specific names. Leave this field blank to find all objects in the network.

2) Click **Find**. Found objects are highlighted in the display (any objects in the display that were previously highlighted are de-selected.)

   The Find Node/Link dialog box displays a list of the seached objects, including their names, subnet names, and types.

3) Select the name of an object in the list, and click **Locate Node/Link**. The display changes to locate and highlight the seleted object.

Group Selection/Action > Selection Filter

Edit > Group Selection/Action > Selection Filter: Select multiple objects based on their attributes.

1) Select the type of object by clicking the appropriate check box.

2) Choose the attribute to be used for selecting objects from the drop-down menu. The list of attributes changes dynamically with the type of objects chosen for selection. From this list, select one or more attributes and specify the matching criteria. If there are objects of the selected type that match the attribute selection criteria specified, they are marked as selected.

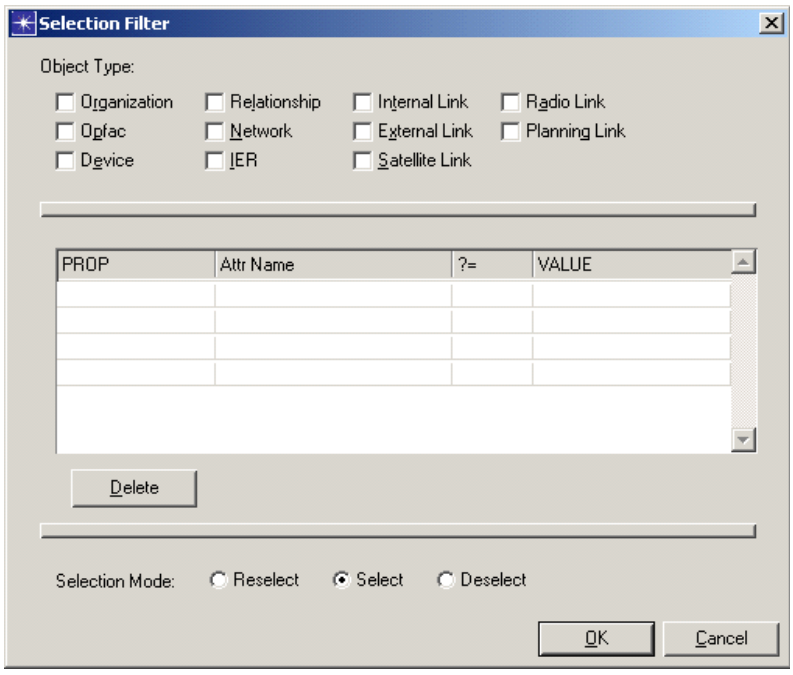


Figure 3-11 Selection Filter dialog box

The table used for specifying the matching criteria has four columns.

— **PROP –** If set to **Require**, an object must satisfy *all* attribute criteria for selection; if set to **Consider**, an object must satisfy at least *one* of the attribute criteria for selection.

— **Attr Name –** Name of the attribute used as the selection criteria.

— **?= –** Specifies the type of matching to be used. For a string, the only option is a complete match or **equal to**. For numeric values, the satisfying condition could be **less than**, **greater than**, or **equal to** the specified value.

— **VALUE –** Value that the attribute must match for selection.

Multiple attributes can be used as matching criteria. Attributes can also be deleted as needed.

3) Click the **OK** button; the software goes through all the objects in the scenario looking for those that match the specified criteria. When it finds objects that match the given criteria, it marks those objects as selected.

Group Selection/Action > Action Filter

Edit > Group Selection/Action > Action Filter: Used in conjunction with the object selection feature. A similar interface is provided for applying action on the selected objects.

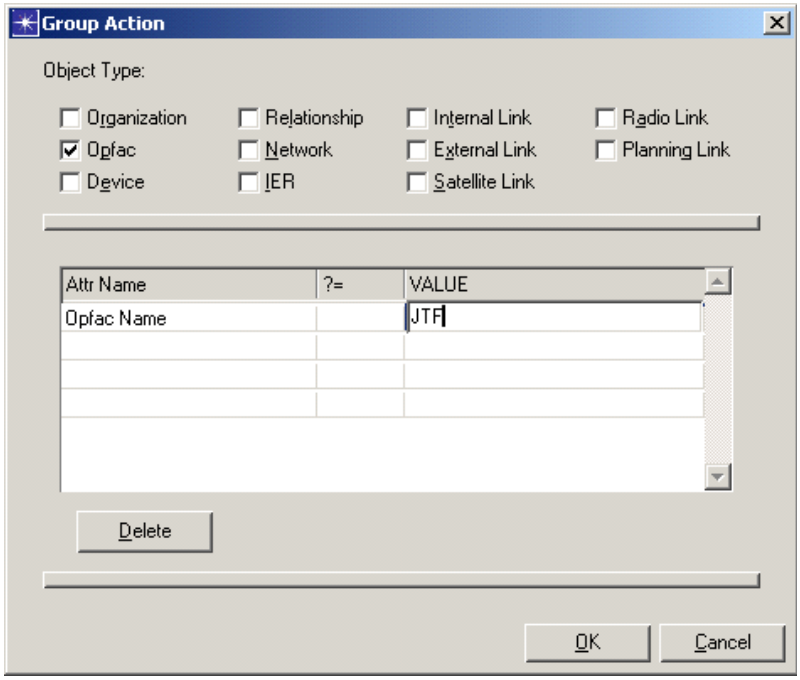


Figure 3-12 Group Action dialog box

1) Select the type of object to which you want to apply group action.

2) Select the attribute that needs to be changed. In the above example, the user selected the **Opfac Name** attribute.

3) Specify the value. In the previous example, the user chose **JTF** as the name to be set on all selected OPFACs.

4) Click **OK**; the software goes through all the selected organization objects in the scenario and sets the trajectory attribute on them.

Save Object Selection Set

Edit > Save Object Selection Set...: Allows you to save an object selection set to a .selset file in the SB_Data directory.

Load Object Selection Set

Edit > Load Object Selection Set...: Allows you to choose a saved object selection set file from the SB_Data directory to load.

Edit Objects Using Template

Edit > Edit Objects Using Template...: Allows you to specify the content you want to include in your report. When you generate a report (by clicking **Generate**), your content settings are saved and will be reloaded the next time you generate a report.

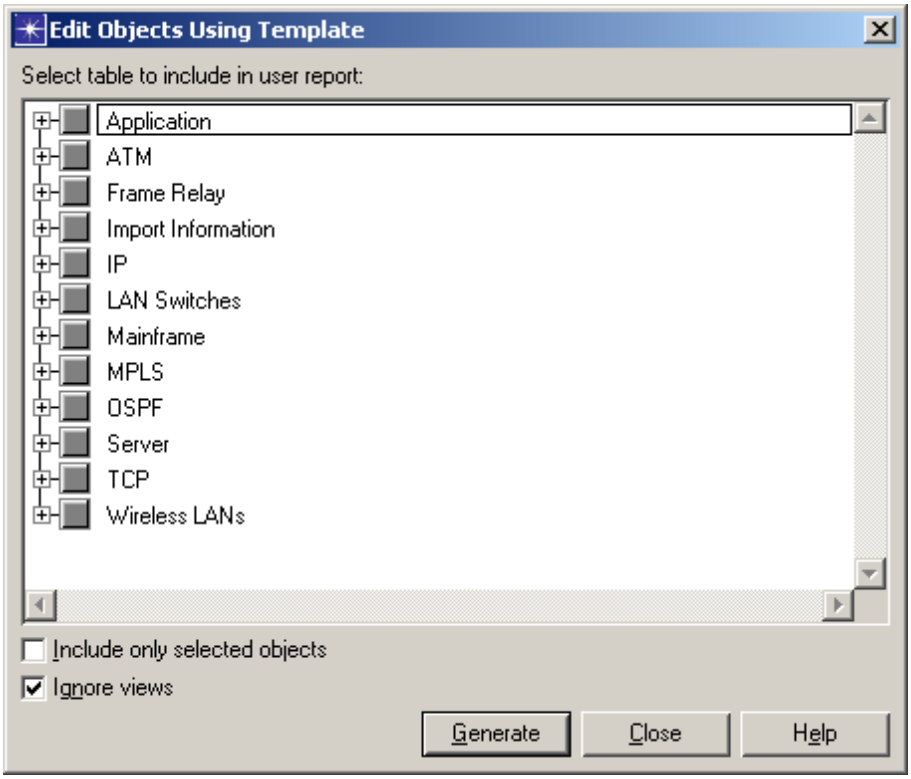


Figure 3-13 Edit Objects Using Template dialog box

- **Select tables to include in user report** treeview: Specifies which tables of which categories should be included in the report. To select an entire category of tables, click on the top-level category entries. To select an individual table, expand the top-level category subtree and click on the table entry. Selected tables and categories show a green check mark. Categories with some but not all of its tables selected display a green dot instead.
- **Include only selected objects in report** checkbox: Indicates that table data will be collected only for objects that are currently selected in the network.
- **Ignore views** checkbox: Indicates that table data will be collected even for objects that are not in the current view or views of the network.
- **Generate** button: Generates the table data for all of the selected tables.
- **Close** button: Closes the dialog box but retains any table selections.

Edit Attribute Template

Edit > Edit Attribute Template...: Allows you to edit the objects in your attribute templates.

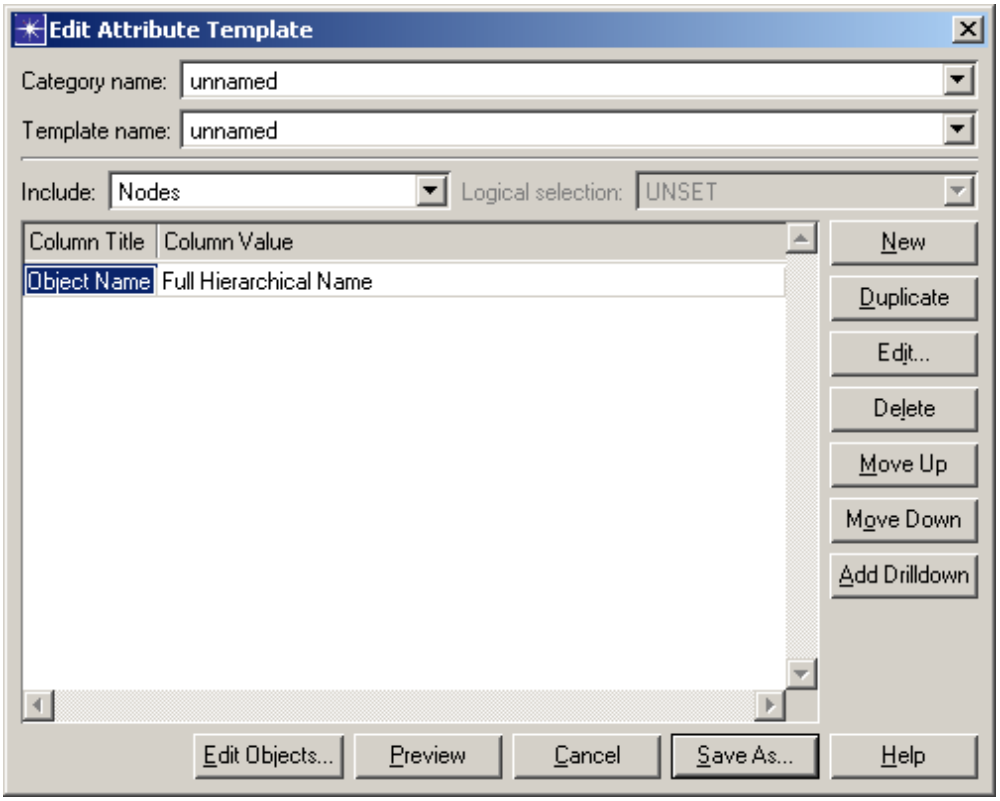


Figure 3-14 Edit Attribute Template dialog box

Project Defaults > Link

Edit > Project Defaults > Link: Specify the default values for a link's attributes based on the link type.

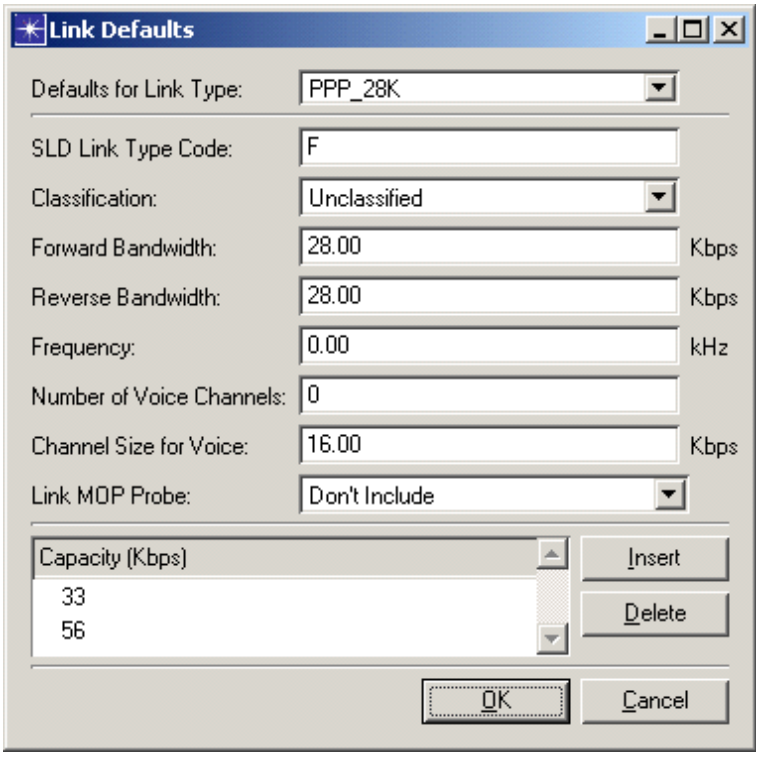


Figure 3-15 Link Defaults dialog box

1) Specify the desired default values for a particular link type.

2) If needed, change the default values for another type by choosing the desired type from the **Defaults for Link Type** drop-down menu.

3) Click the **OK** button to save the default values.

Project Defaults > Network

Edit > Project Defaults > Network: Specify the default values for a broadcast network's attributes based on the network type.

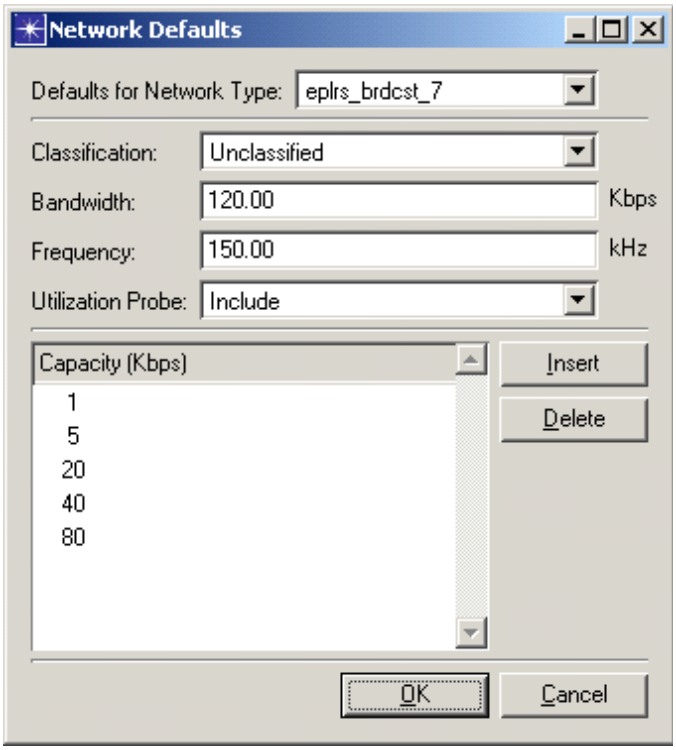


Figure 3-16 Network Defaults dialog box

1) Specify the desired default values for a particular network type.

2) If needed, change the default values for another type by choosing the desired type from the **Defaults for Network Type** drop-down menu.

3) Click the **OK** button to save the default values.

Project Defaults > IER

Edit > Project Defaults > IER: Specify the default values for an IER's attributes based on the IER's traffic type.

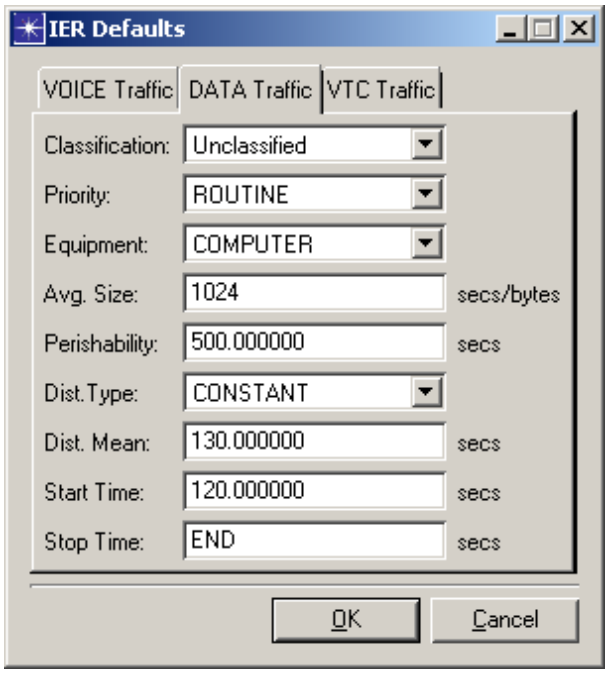


Figure 3-17 IER Defaults dialog box

1) Specify the desired default values for a particular traffic type.

2) If needed, change the default values for another type by choosing the desired type from the **Defaults for Traffic Type** drop-down menu.

3) Click the **OK** button to save the default values.

Project Defaults > Requirements Matrix

Edit > Project Defaults > Requirements Matrix: Edit the default requirements matrix questions that are applied to all new projects and scenarios created in the Scenario Builder. All fields are editable and can be modified by clicking on a cell and entering new values.

The **Applications** column displays application names for the specified service. If there is more than one application for the service, a comma separates application names with no space after the comma separator.

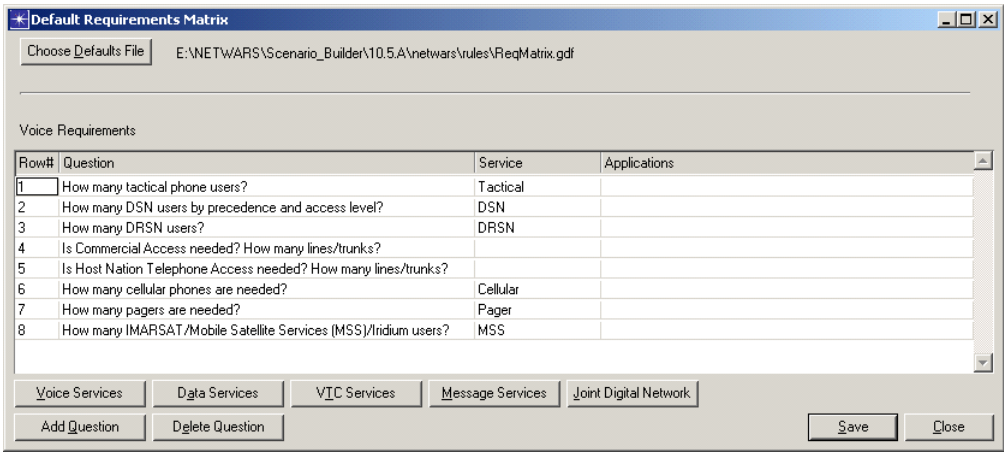


Figure 3-18 Requirements Matrix Default dialog box

Clicking on the buttons below the table displays different functional areas. Questions can be added or deleted via the **Add Question** and **Delete Question** buttons.

By default, the **Save** button saves the changes and replaces the ReqMatrix.gdf file located in the Rules directory. To change the default file, click the **Choose Defaults File** button. Selecting this button launches a file chooser dialog box. Select the desired default file; the contents of the alternate file import into the Defaults Requirements Matrix dialog box.

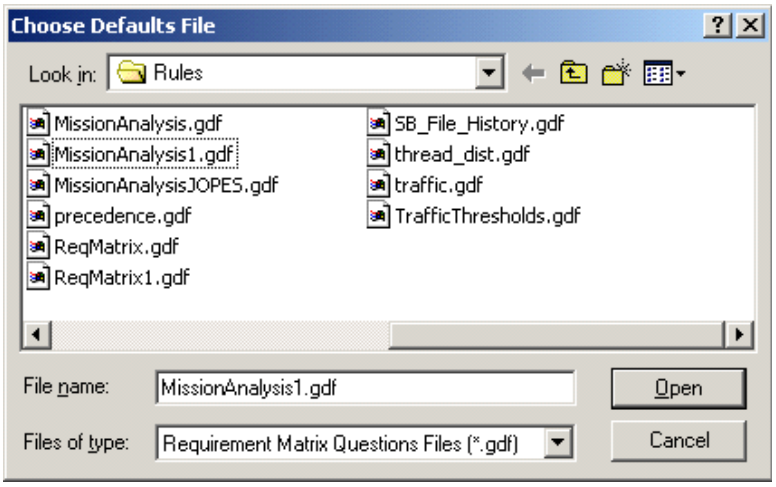


Figure 3-19 Specifying the Requirements Matrix defaults file

Project Defaults > Mission Analysis

Edit > Project Defaults > Mission Analysis: Edit the default mission analysis questions that are applied to all new projects created in the Scenario Builder.

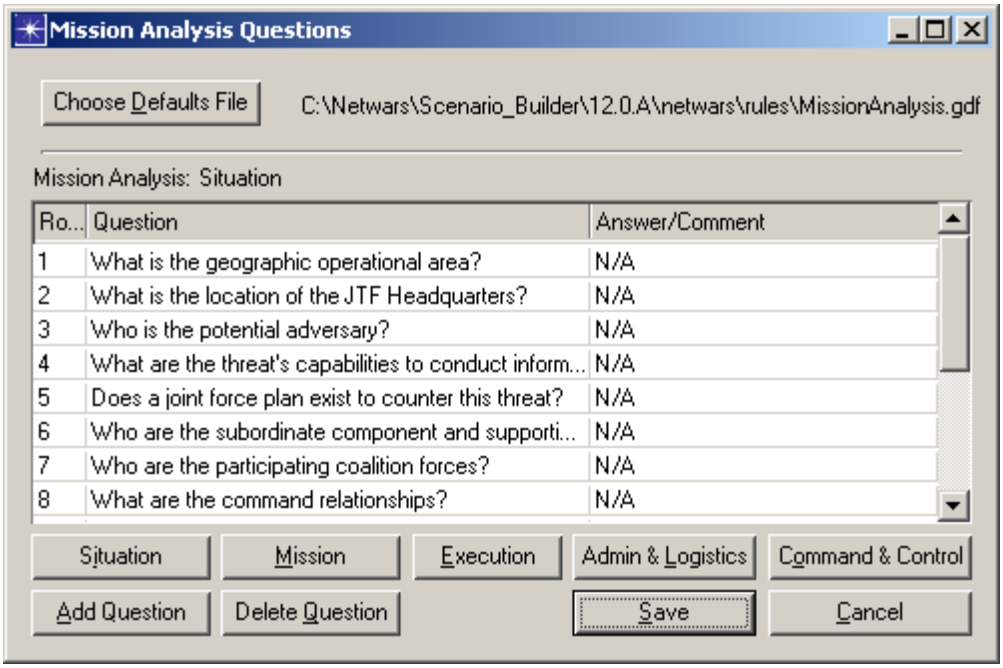


Figure 3-20 Mission Analysis defaults

Answer questions by clicking on a cell and entering values.

Clicking on the buttons below the table displays different functional areas. Questions can be added or deleted via the **Add Question** and **Delete Question** buttons.

Saving the changes will modify the `MissionAnalysis.gdf` file located in the Rules directory. To change the default file, click the **Choose Defaults File** button. Selecting this button launches a file chooser dialog box. Select the desired default file; the contents of the alternate file import into the Mission Analysis Defaults dialog box.

Project Defaults > Distance Between OPFACs

Edit > Project Defaults > Distance Between OPFACs: Configure the preferences for the Distance Between OPFACs feature. See field descriptions below.

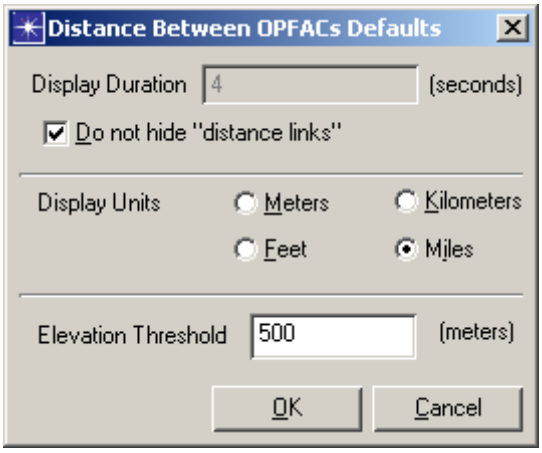


Figure 3-21 Distance Between OPFACs Defaults

- **Display Duration** – Specifies the amount of time that the OPFAC Distance Links will display if you do not have the **Do not hide "distance links"** preference enabled.
- **Do not hide "distance links"** – Specifies whether the Scenario Builder will automatically hide the OPFAC Distance Links after a specified amount of time (refer to the **Display Duration** field above).
- **Display Units** – Sets the units in which the OPFAC Distance Links will display their distances.
- **Elevation Threshold** – Determines whether the distance calculation will consider the Earth's curvature. If both OPFACs' altitudes fall below this value, it will consider the Earth's curvature, but it will not if either OPFACs' altitude exceeds or equals this value.

Preferences > Device Model Map > New

Edit > Preferences > Device Model Map > New: Bridge the differences in model names/naming conventions that exist between JCSS and Network Engineer. Device model mapping allows you to define a mapping between two sets of device model names. During export of data, the JCSS XML processing code can substitute model names in the source data with alternate names in the target data.

1) From the Scenario Builder, select **Edit > Preferences > Device Model Map > New**. The Choose Source and Target Model Name Lists displays. Use this dialog box to select a source model list and a target model list; this means that device model names in the source list will be replaced with designated model names in the target list.

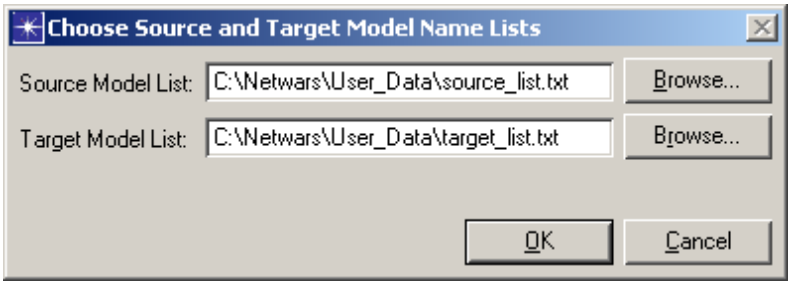


Figure 3-22 Choose Source and Target Model Name Lists dialog box

2) Click the **Browse** button for the **Source Model List**. The Source Model List Browser dialog box displays.

3) Open the appropriate **Rules** folder.

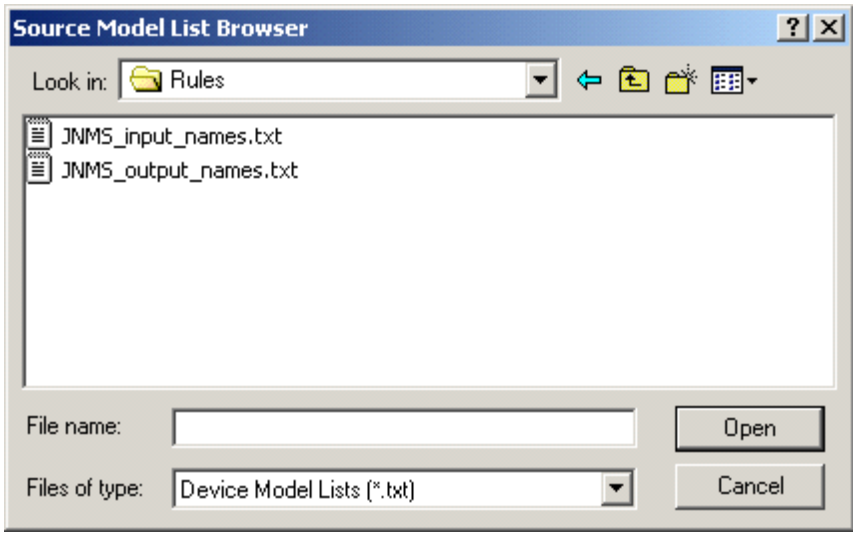


Figure 3-23 Source Model List Browser

Note—JCSS provides template source and target model list files from which you can choose in the Scenario_Builder/12.0.A/netwars/rules directory. You can select a template file, modify it, and save it under a new name in the User_Data/Rules folder to use again. Remember: Save your rules files to the default User_Data/Rules folder so you can easily find them again.

4) Select the desired input file and then click **Open**.

5) Click the **Browse** button for the **Target Model List**. Repeat steps 3-4, this time selecting an output file as the target model list.

6) Click **OK**. The Edit Device Model Mapping File dialog box displays.

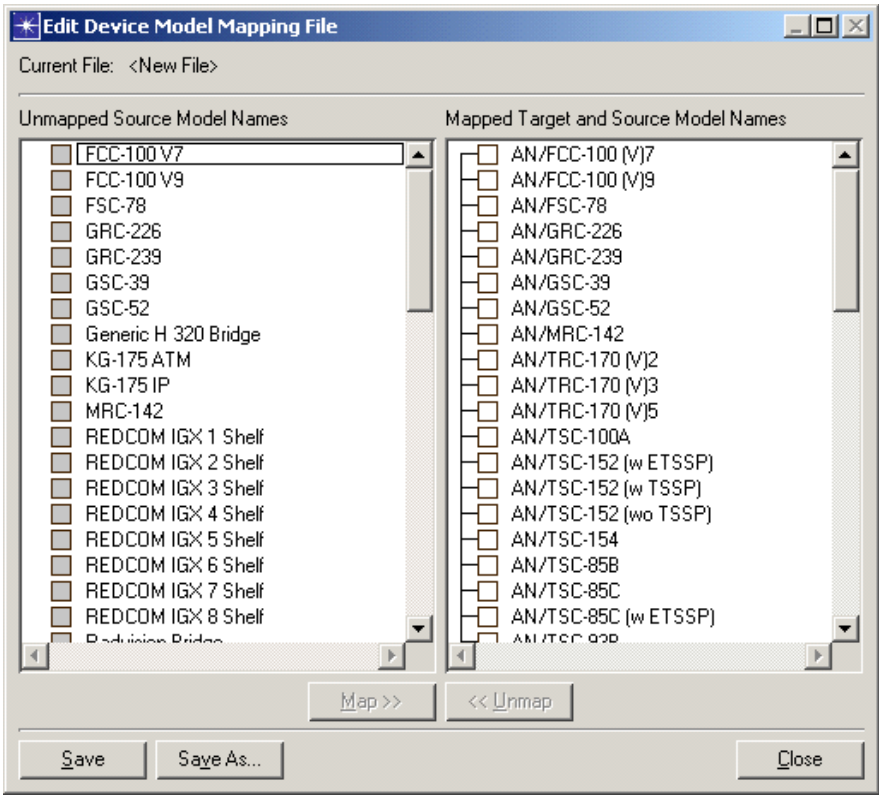


Figure 3-24 Edit Device Model Mapping File dialog box

Devices in the **Unmapped Source Model Names** panel (from the source model list) can be mapped to only one device in the **Mapped Target and Source Model Names** panel (from our target model list). However, it is possible to map more than one source name to the same target name (many-to-one mapping).

7) Select the desired model in the **Unmapped Source Model Names** panel. Next, in the **Mapped Target and Source Model Names** panel, select one or more models.

8) Click the **Map>>** button. The source device name displays under the target device name in the **Mapped Target and Source Model Names** panel.

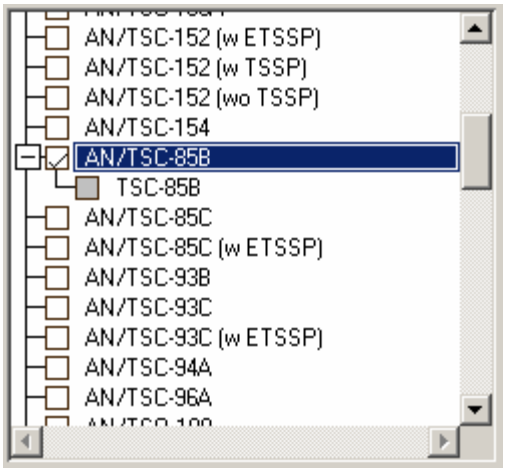


Figure 3-25 Mapped devices

9) Click the **Save** button. The Device Model Map Browser dialog displays. Name the file in the **File Name** field and then click **Save**.

Preferences > Device Model Map > Open

Edit > Preferences > Device Model Map > Open: Launches the Device Model Map Browser dialog box which enables you to select a previously created device model map to edit using the Edit Device Model Mapping File dialog box.

Preferences > Device Model Map > Choose

Edit > Preferences > Device Model Map > Choose: Use this dialog box to select an import map (source model list) and an export map (target model list) defaults for device model mapping.

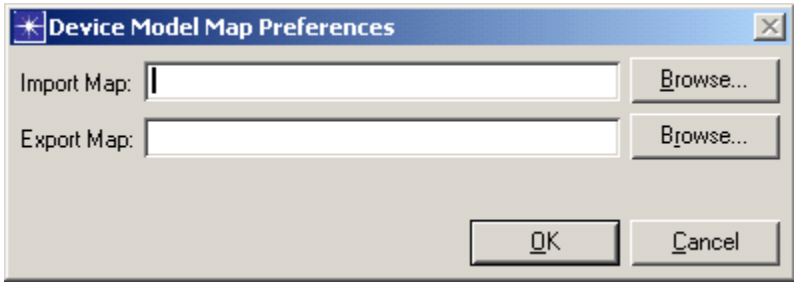


Figure 3-26 Device Model Map Preferences dialog box

1) Click the **Browse** buttons for the **Import Map** and **Export Map** fields to select desired device model maps.

2) Click **OK**.

Preferences > Functional Profiles

Edit > Preferences > Functional Profiles: Set default functional profiles and functional names. Every OPFAC has one or more functional names. A functional name maps an IER or series of IERs to a producer/consumer OPFAC pair. A collection of such functional names is called a functional profile. A functional profile can be defined locally on an individual OPFAC, or globally, so that it is available to all OPFACs and across different scenarios.

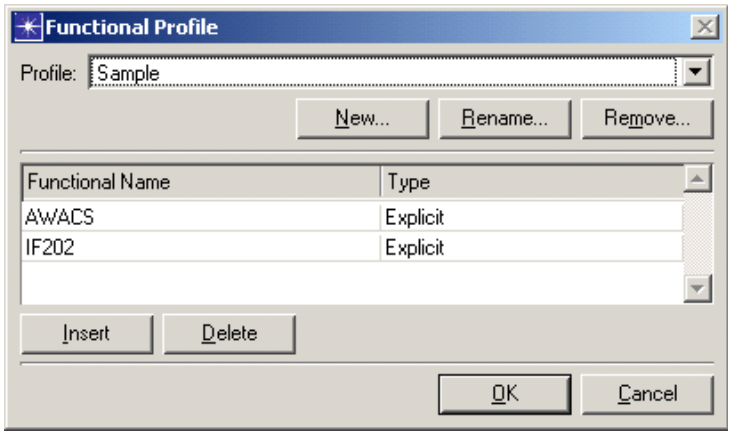


Figure 3-27 Functional Profile dialog box

1) Add, rename, or remove functional profiles as needed.

2) Insert, delete, or edit functional names as needed.

3) Click **OK**.

Preferences > Geographical Coordinates

Edit > Preferences > Geographical Coordinates: Sets the default geographical coordinate system view displayed in the workspace of the Scenario Builder. There are three types of geographical coordinate systems available in JCSS: Lat/Long, UTM, and MGRS.

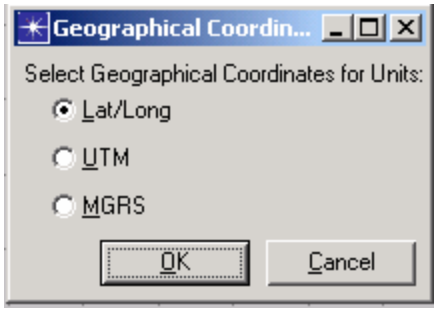


Figure 3-28 Geographical Coordinates dialog box

1) Select the **Lat/Long**, **UTM**, or **MGRS** radio button.

2) Click **OK**.

Preferences > Owners List

Edit > Preferences > Owners List: Set owner information for units marked for subordinate query. Modify existing records or add/delete records to the list as needed.

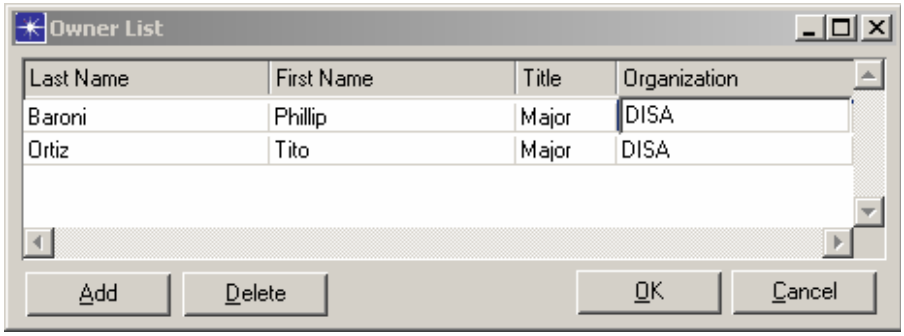


Figure 3-29 Owner List dialog box

Owner information contains four fields: Last Name, First Name, Title and Organization. All fields, except the Title field, are required fields.

Preferences > Security Classifications

Edit > Preferences > Security Classifications: Set security classification for a variety of items in a scenario, such as devices and IERs. This dialog allows you to add, remove, or change values in the list of recognized security classification values. The list is used to populate drop-down lists in edit fields and table cells that specify security classification values. Since some fields and cells only allow values from this list, you will need to modify this list in order to specify custom security classifications. Any value is permitted, although duplicates are not allowed in the list.

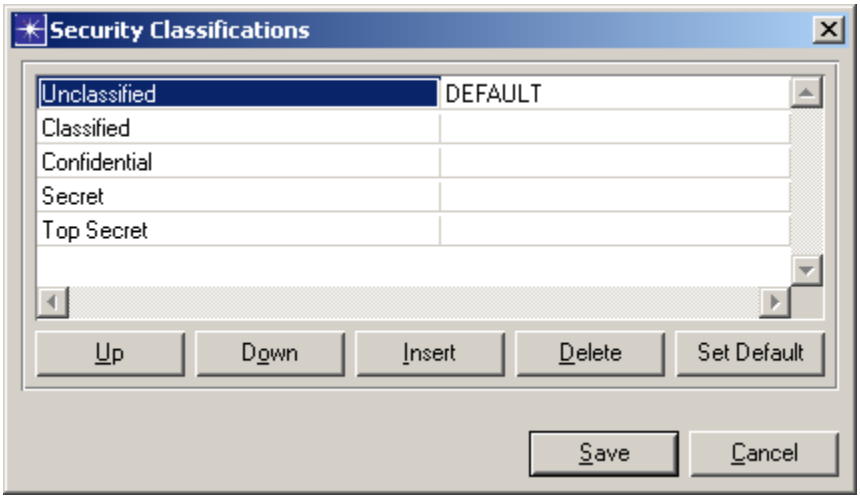


Figure 3-30 Security Classifications

The list order of an entry indicates the classification's level of restriction. Less restrictive classifications appear toward the top of the list.

A value in the list may be modified by selecting and editing it. The **Up** button moves the selected value toward the top of the list. The **Down** button moves the selected value toward the bottom of the list. The **Insert** button inserts a blank entry above the selected value. The **Delete** button deletes the selected value. The **Set Default** button sets the selected classification as your default. The **Save** button saves all changes. Once saved, the changes will affect all edit fields and table cells, but the values stored in those attributes will not be modified.

Preferences > Traffic Flow Thresholds

Edit > Preferences > Traffic Flow Thresholds: Specify a minimum value and the colors to be used for each threshold; traffic flow thresholds are used in conjunction with the Aggregate Traffic Flows feature. Flows are colored based on the threshold they fall under. If the load of an aggregate flow is more than the highest threshold, the flow is colored black. If the load is lesser than the minimum threshold, the flow is not displayed. The default minimum value is 0 Kbps.

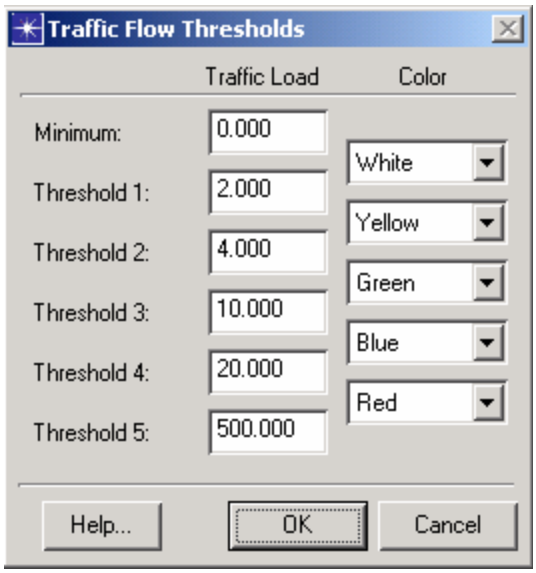


Figure 3-31 Traffic Flow Thresholds dialog box

Set desired traffic load/color combinations and then click **OK**.

Preferences > Advanced

Edit > Preferences > Advanced: Display the Preferences Editor and set environment attributes that control program operation. These values are stored in a file called the environment database file (env_db for short).

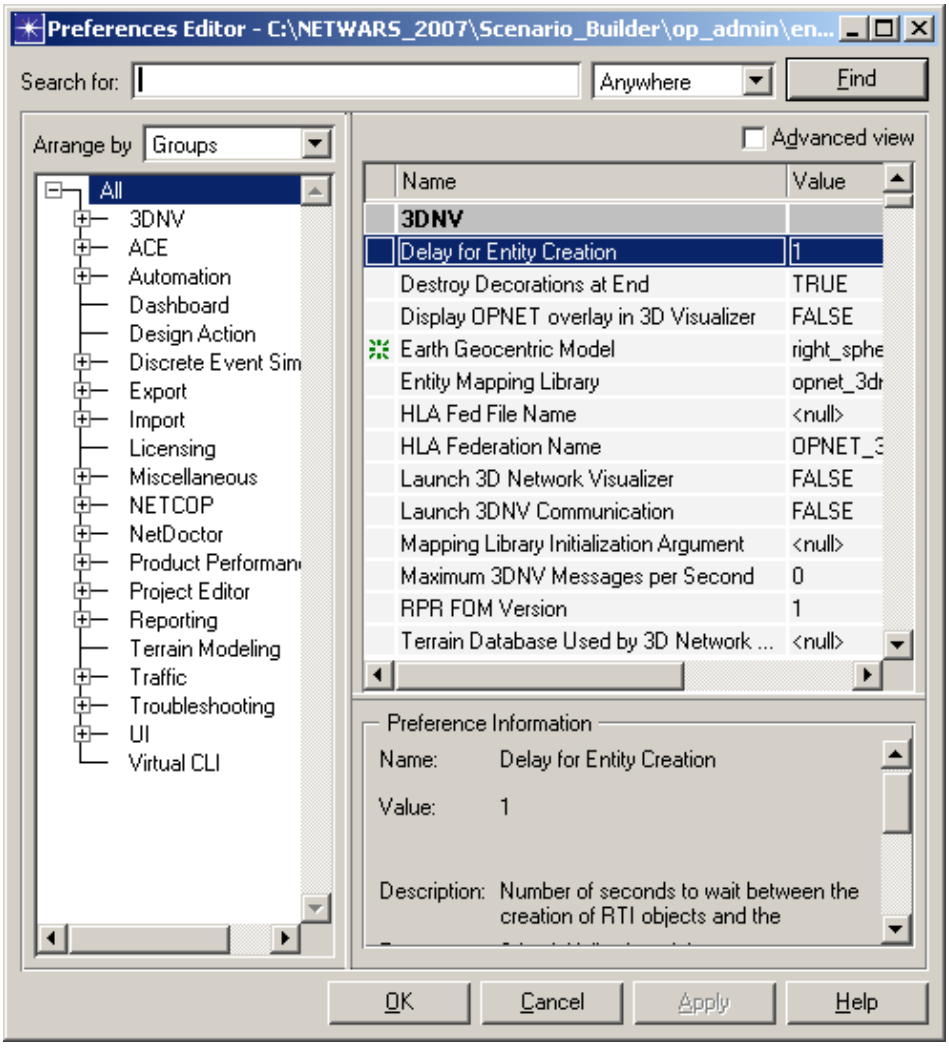


Figure 3-32 Preferences Editor dialog box

- **Search for**: Enables you to search for preferences. The system searches for words or values containing the specified text in whole or in part (a wildcard search is assumed.) From the pull-down menu, select the preference information to include in the search: **Anywhere** specifies to search Tags, Names, Values and Descriptions; **In Names** specifies to search Tags and Names only; and **In Values** specifies to to search Values only.

- **Arrange by**: Enables you to display preferences by category. From the pull-down menu, select **Groups** to arrange preferences by group (filters the list of displayed preferences to a specific group by selecting the group in the treeview,) or **Source** to arrange preferences by source (preferences are classified based on the source of their values.)

- **Advanced view checkbox**: Toggles between showing an advanced or basic view of the preferences table. In the basic view, the table shows the name

and value of each preference. In the advanced view, the table shows the name, tag, source of the preference's value and the value itself.

- **OK** button: Saves changes and closes the Preferences Editor.
- **Cancel** button: Closes the Preferences Editor without saving changes.
- **Apply** button: Saves changes.

View Menu

Show Logical Views

View > Show Logical Views...: Specify filtering parameters for logical views.

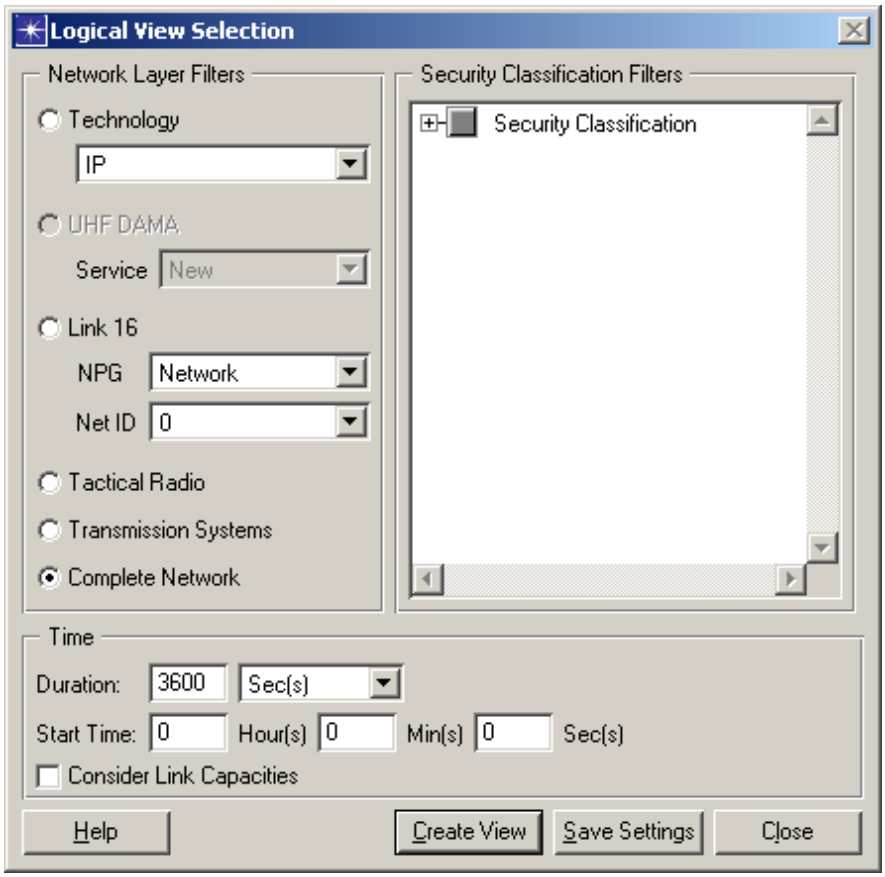


Figure 3-33 Logical View Selection dialog box

- **Network Layer Filters**: Enables you to hide all devices except for those of a particular layer of the network. (Note that this doesn't necessarily mean OSI network layer, but instead, it means the various devices that use a particular protocol.) Network Layer Filters shown are only those which apply to the deployed network.
- **Security Classification Filters**: Enables you to filter elements of the network based on their security classifications. Any custom classifications added in the Operational view will be availabe here as well for filtering.
- **Duration and Start Time**: These parameters are used to get the Capacity Planner graphs that are used for filtering for various layers.

- **Consider Link Capacities** checkbox: This is another parameter used to get the Capacity Planner graphs. A link capacity considered filter will result in the Capacity Planner creating a graph that may not show complete physical connectivity of the devices due to link load. A link capacity ignored filter will result in a graph that shows complete physical connectivity of the devices regardless of link load.
- **Create View** button: Creates the type of view that you specified and loads it into a Network Showcase window. The filter specifications are applied to the logical view model.
- **Save Settings** button: This button is availabe from the Operational View and gives you the option to save the settings instead of creating a logical view network with the specified settings at this time.
- **Close** button: Closes the dialog without making any changes to the model.

Show Network Browser

View > Show Network Browser: Use the Network Browser, a pane that lists objects (organizations, OPFACs, nodes, devices, links, etc.) in the network, to locate objects in the workspace and edit their attributes.

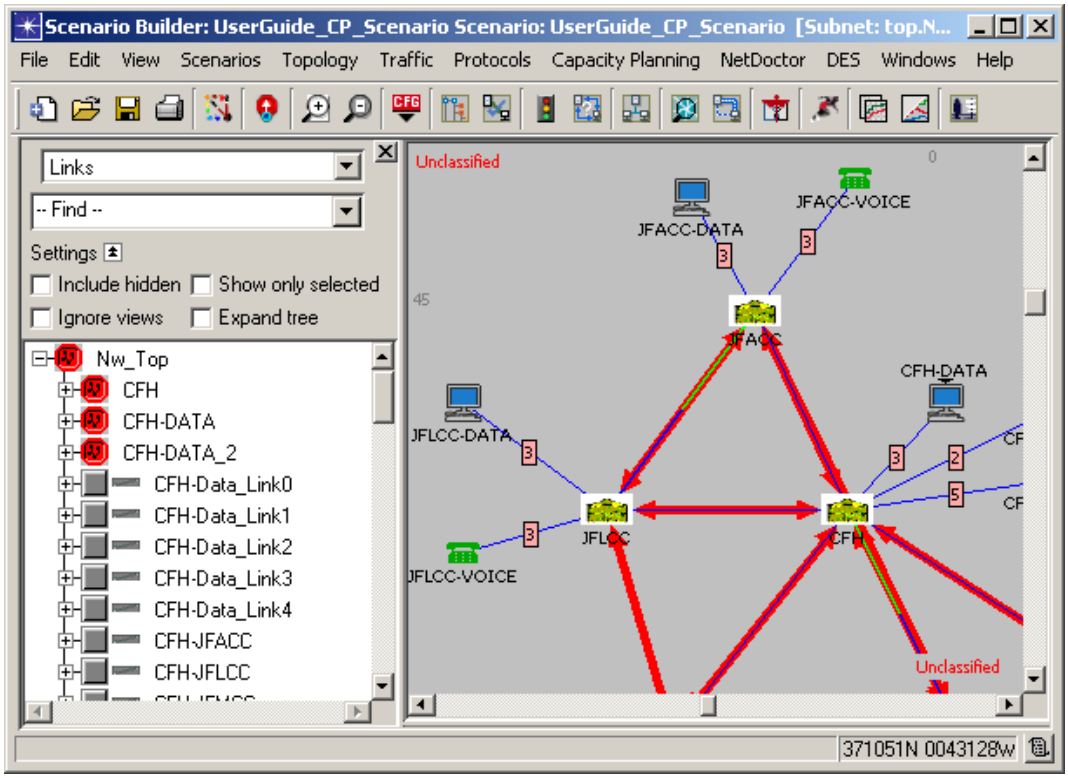


Figure 3-34 Network Browser

You can modify the object list by applying filters. Available filters include:

- **Classification:** Filters based on the classification attribute on each device.
- **Equipment Type:** Filters based on the equipment type attribute on each device.

- **Ownership:** Shows the associated owner for each device in the scenario. Used with Collaborative Planning.
- **Broadcast Networks:** Shows all of the broadcast networks in the scenario and to which OPFACs/devices they are attached.
- **Relationships:** Shows all of the relationships in the scenario and to which OPFACs they apply.
- **Satellite Links** or **Radio Links:** Shows all of the satellite (or radio) links in the scenario and to which OPFACs/devices they are attached.

Click on an object in the object list to display it in the workspace (upper-right.) Select and check an object to display it in a Showcase window (lower-right.) Use options provided on the right-click menu to modify object attributes.

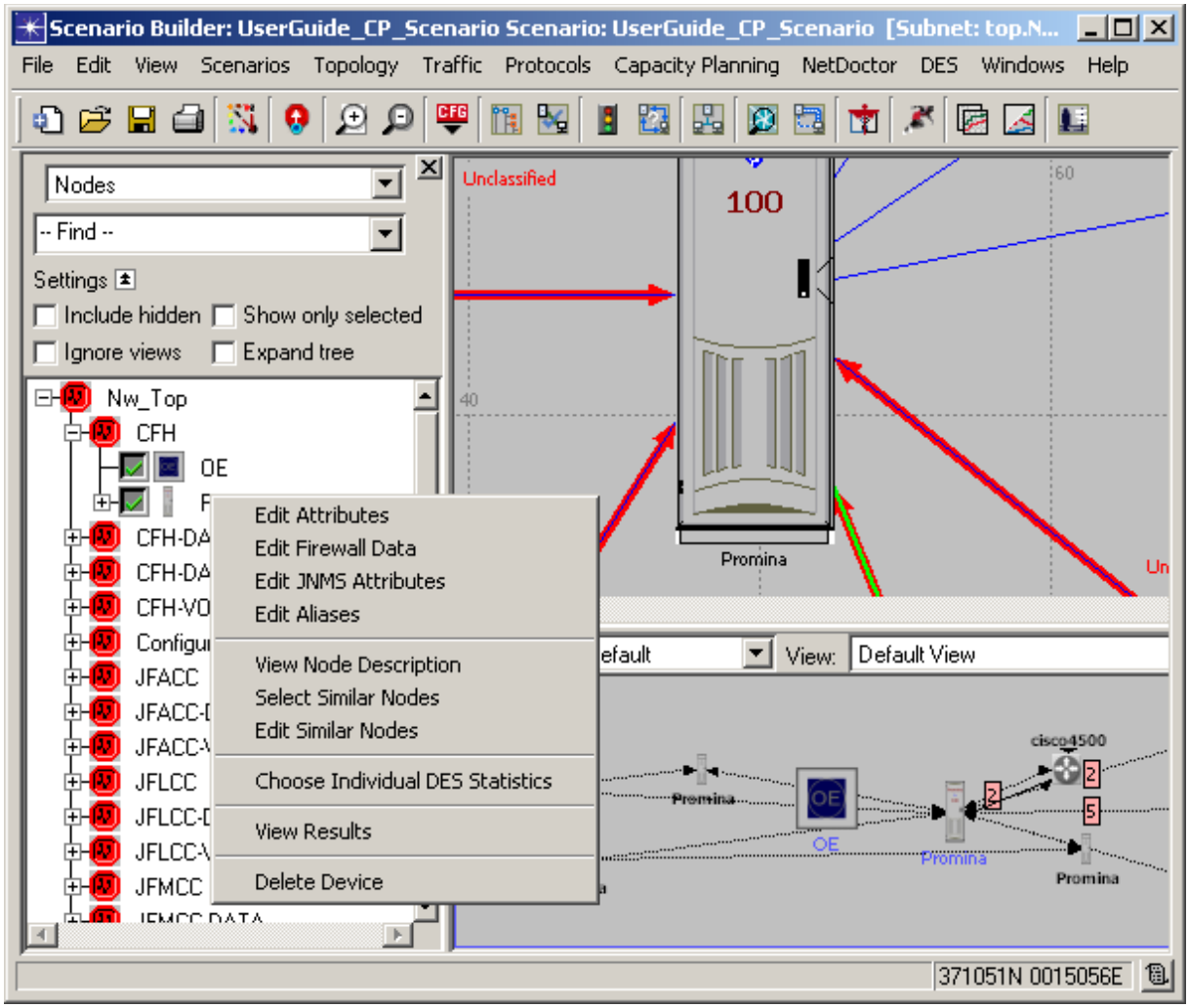


Figure 3-35 Network Browser with Showcase Window and Right-click Menu

If you have an OPNET Modeler license, you can easily open node models in Modeler by double-clicking on a device in the workspace. Note that for derived models, you must open the parent instead.

1) Double-click on a node model. The following message dialog displays.

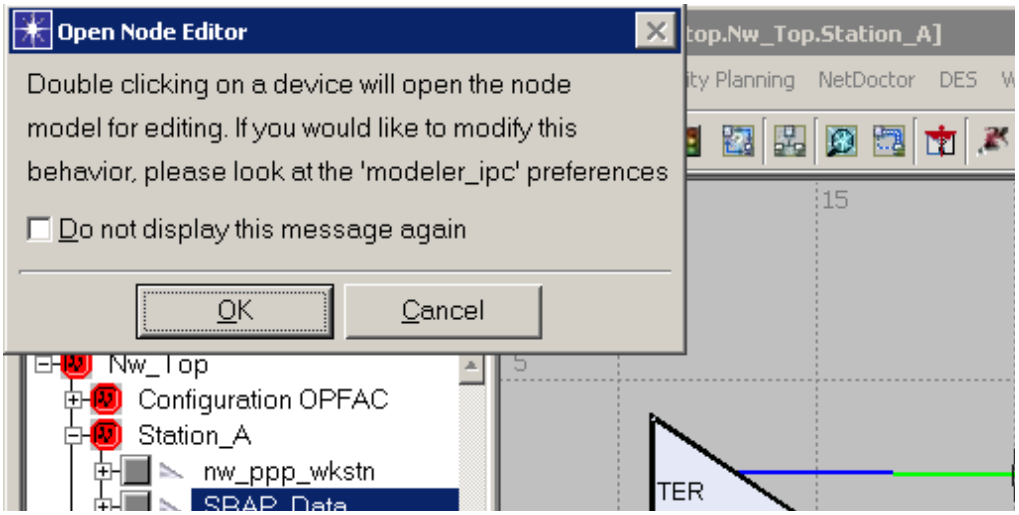


Figure 3-36 Open Node Editor Message Dialog

2) Click **OK**. OPNET Modeler launches and displays the node model.

Note that Modeler will not launch if you do not have a license for it.

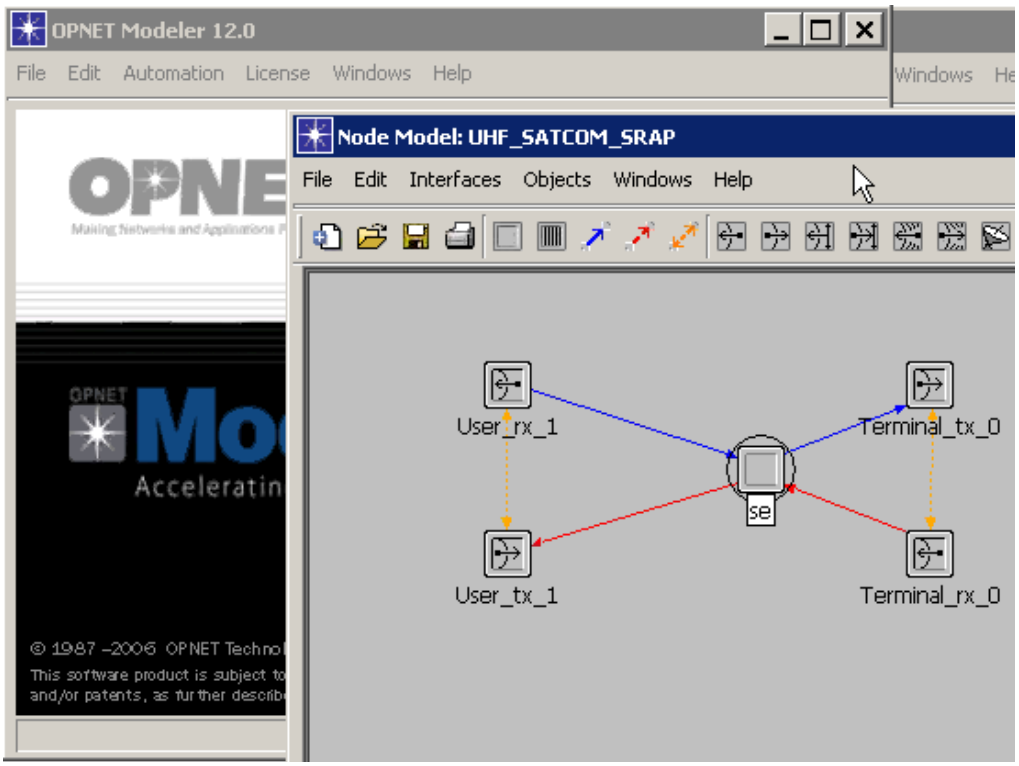


Figure 3-37 Opening a Node Model in OPNET Modeler

Using the Preferences Editor in JCSS, you can set preferences for the Modeler launch feature:

- **modeler_ipc.enable:** Enables (set to TRUE) or disables (set to FALSE) the Modeler launch capability.
- **modeler_ipc.port:** Sets your port preference for Modeler launch. Specify the port number you want to use to connect to Modeler when opening node models.
- **modeler_ipc.show_dialog:** Shows (set to TRUE) or hides (set to FALSE) the Open Node Editor message dialog on Modeler launch.

Consult the documentation for OPNET Modeler for instructions on using Modeler with node models.

Show Network Topology

View > Show Network Topology: Toggles the view of the network topology on and off in the Scenario Builder workspace.

Show Network Showcase

View > Show Network Showcase: Toggles the showcase display on and off in the Scenario Builder window. Allows you to choose to use the default showcase (or select another showcase to use from the Showcase pull-down menu) and the default view (or select another view from the View pull-down menu.) You can also create a new showcase, and import, duplicate, rename or delete a showcase using options in the Showcase pull-down menu.

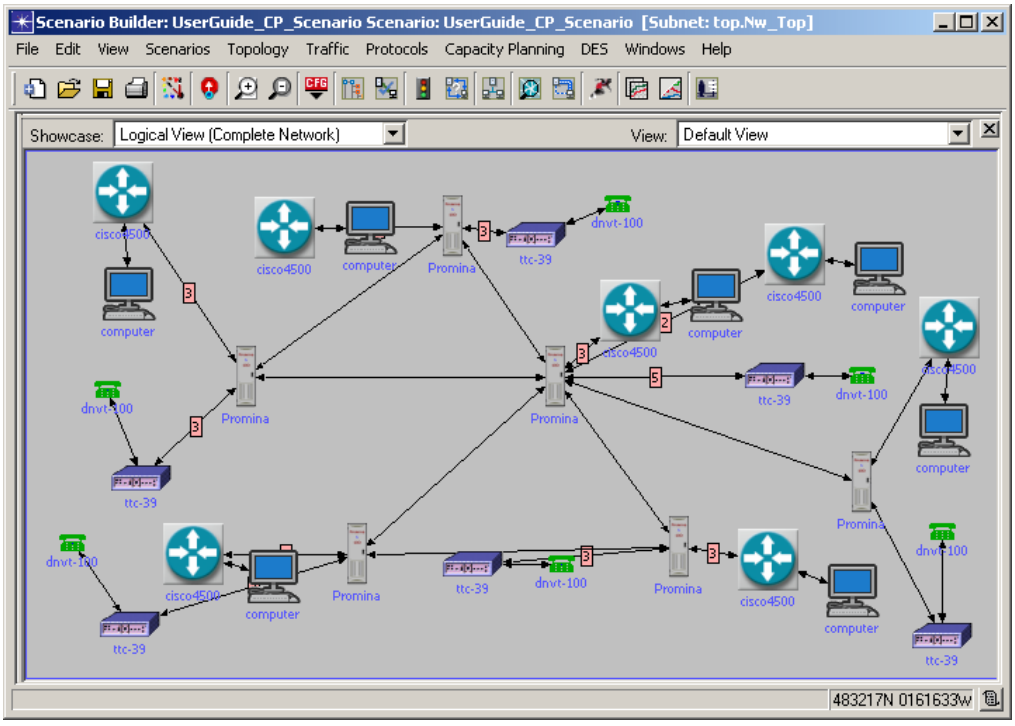


Figure 3-38 Show Network Showcase in Scenario Builder

Open Network Showcase Window

View > Open Network Showcase Window: Allows you to select a saved network showcase and display it in a separate window.

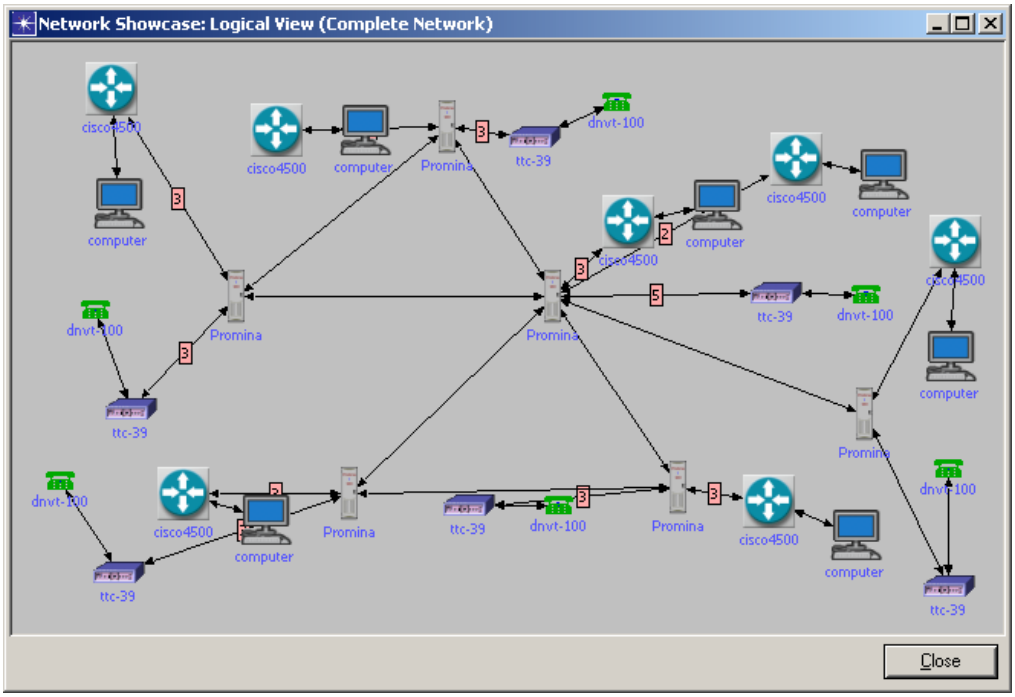


Figure 3-39 Network Showcase window

Show Time Controller

View > Show Time Controller: Displays the Time Controller dialog box that allows you to change the current network time visualized in the editor and animate the time change effect.

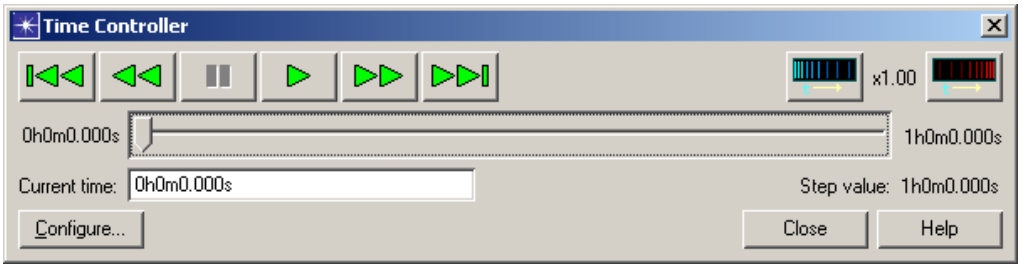


Figure 3-40 Time Controller dialog box

The Time Controller dialog box shows the current network time (text field), value for the time step which is used for stepping forward or backward using the animation buttons, animation control buttons (to **Jump to start** to reset the network time to the slider start time, **Step backward** to decrease network time by one time step, **Play forward** or **Pause animation**, **Step forward** to increase network time by one time step, **Jump to end** to set network time to slider end time, **Slow down play** and **Speed up animation**), and a time slider which provides an alternative way to change the current network time.

Click the **Configure** button to display the Time Controller Settings dialog box, which allows you to set and change Time Controller values.

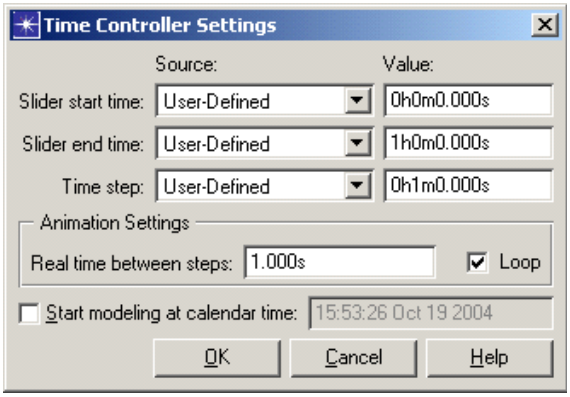


Figure 3-41 Time Controller Settings dialog box

- **Slider start time**, **Slider end time**, and **Time step**: Specifies the source plug-in for the slider start time, end time, and time step values. If **User-Defined** is selected, you can enter the value in the corresponding field.
- **Real time between steps** field: Specifies the speed of the animation.
- **Loop** checkbox: If checked, animation restarts after completion.
- **Start modeling at calendar time**: Check and type a value if you want to specify a calendar time for the network start time. Otherwise, the start time is assigned the symbolic value of 0.0.

Set Area of Interest

View > Set Area of Interest: Provide geographical context to the scenario.

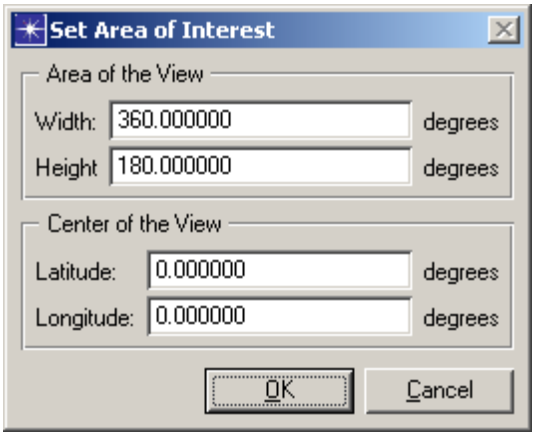


Figure 3-42 Set Area of Interest dialog box

The geographic focus can be specified by entering values for **Area of the View** and **Center of the View**. These attributes determine the location and size of the displayed view of the map. For example, if the center of view is (0,0) and the area of view is (width = 100, height = 50), then the portion of the map that would be displayed will be from (50W, 25N) to (50E, 25S).

The geographic focus also determines the relative range at which OPFACs (and their devices) are located from each other. This is of significance when exchanging IERs/traffic during a simulation. While TSC-85 radios and wire links have no range limitations, TRC-170 radios and tactical broadcast networks only work within a limited range of each other. For example, if two TRC-170 radios are more than 150 miles apart, then all packets transmitted between them will be dropped, resulting in failed IERs.

Go to Parent Organization

View > Go to Parent Organization or click the **Go to Parent Organization** toolbar button: Change the view in the workspace to the next higher level in the network. For example, if this option were selected inside an OPFAC, the next level up would include the parent OPFAC. Similarly, if this option were selected for an OPFAC inside an organization, the next level up would be the parent organization as seen below.

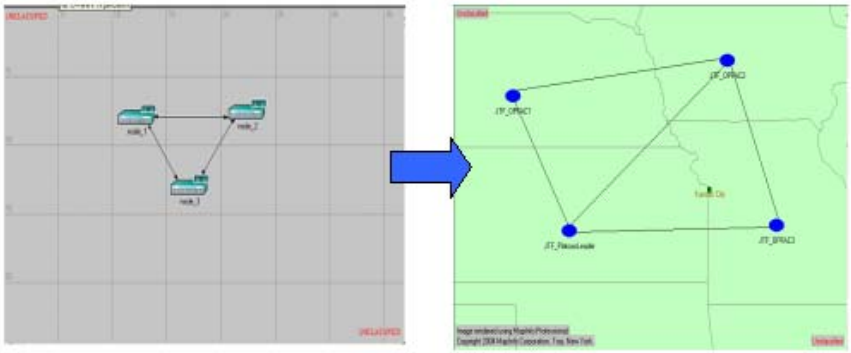


Figure 3-43 Go to Parent Organization

Background > Set Properties

View > Background > Set Properties: Control the look-and-feel of the scenario in the workspace. Specify the units and draw style to be used for the grid lines, the color of the links, the space between grid lines, and the resolution to be used for displaying the network.

1) Select desired view properties from the options at the top of the dialog box.

2) The scenario displayed in the workspace is made up of many different layers. Specify which layers to view and hide using the checkboxes next to each layer.

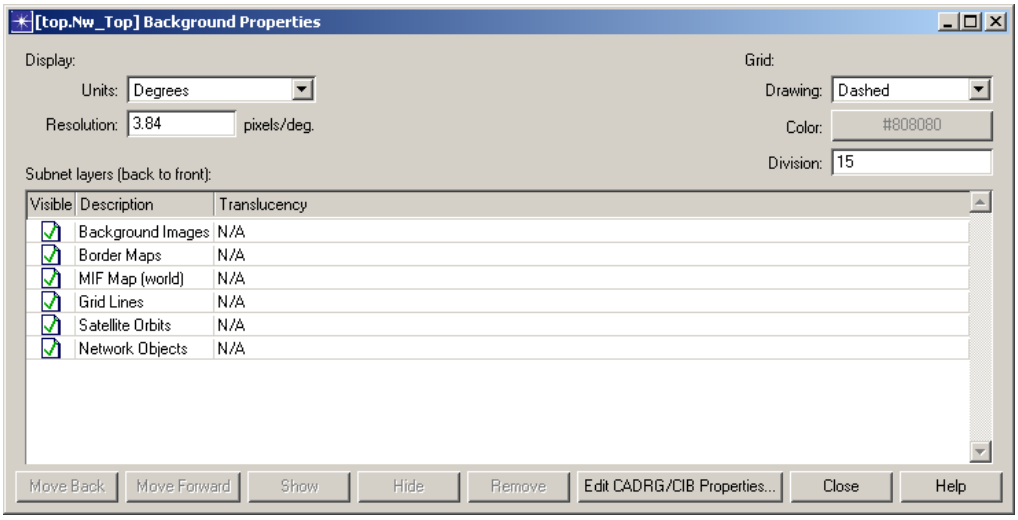


Figure 3-44 Setting background view properties

3) Move layers backward and forward to indicate the layer ordering in the workspace.

4) The **Translucency** column specifies the level of layer content translucency with a value between 0 (opaque) and 100 (transparent).

   NOTE: Presently, only layers with CADRG/CIB and MrSID raster images support translucency.

5) Click the **Edit CADRG/CIB Properties...** button to open a dialog box with configurations of background slices with CADRG/CIB rasters.

   NOTE: Requires an active Terrain Modeling Module license.

Background > Set Border Map

View > Background > Set Border Map: Specify a border map for a scenario. Border maps use stylized lines to define political and geographical areas. JCSS comes with a library of border maps that represent a geographic area described by latitude and longitude.

1) To specify the border map from the Choose Border Map dialog box, choose the border map from the drop-down menu. Note that the border map list is a finite list and may not be modified.

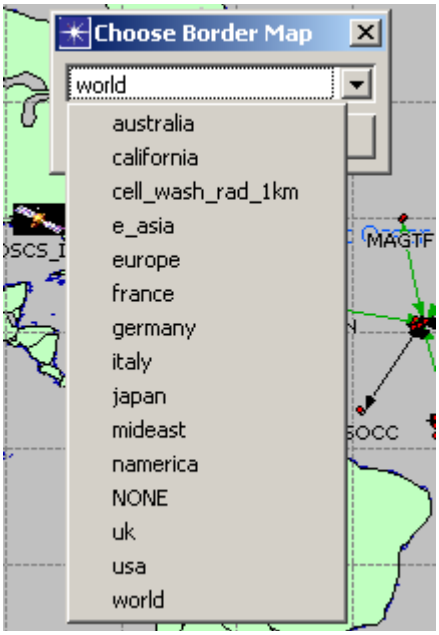


Figure 3-45 Selecting a Border Map

2) If the desired map is not in the list, select the world map and zoom in to the desired location.

Background > Add Image Map

View > Background > Add Image Map: Import an image into the scenario.

1) Choose an option of importing an existing map or specifying the file path to an image map located on the hard drive; select desired map.

2) Click **OK**.

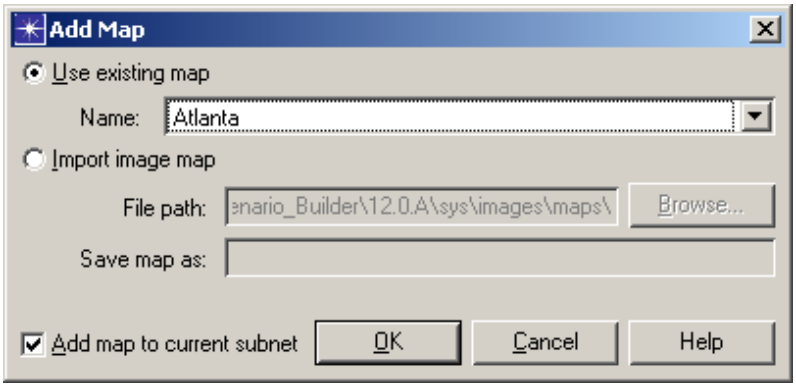


Figure 3-46 Importing an Image Map

Background > Add MIF Map

View > Background > Add MIF Map: Import MIF maps, which are files that provide additional geographical information on a border map, such as roadways, waterways, regions, and counties. Several MIF maps come standard with JCSS.

1) From the MIF menu, select the desired MIF map.

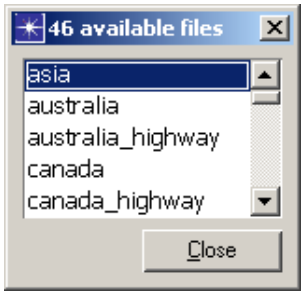


Figure 3-47 Importing MIF Data

2) Click **Close**. The MIF map automatically imports.

Background > Show MIF Log

View > Background > Show MIF Log: Display the MIF Import Log dialog box.

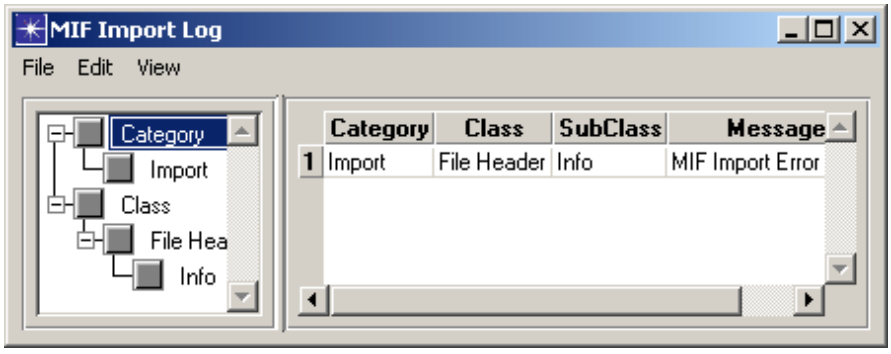


Figure 3-48 MIF Import Log dialog box

Background > Add Image

View > Background > Add Image: Import ordinary background images, i.e., those images that do not include geographical information, into a scenario. Background images differ from background image maps in that they are described by referencing points on an x-y axis.

1) In the Add Background Image dialog box, select the **Import background image** radio button and then click **Browse** to select the TIFF file to import.

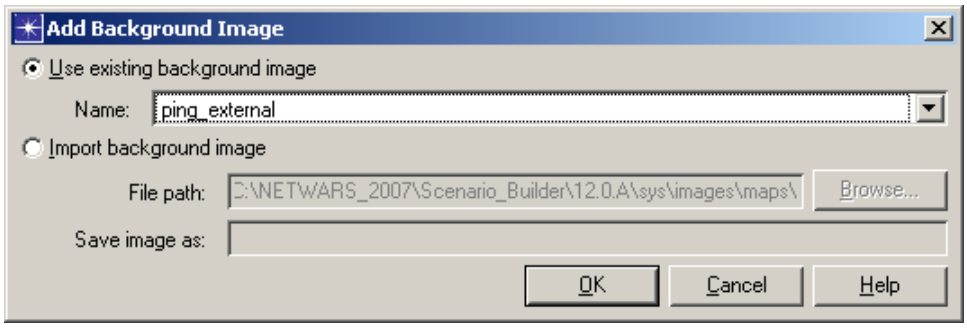


Figure 3-49 Importing a Background Image

2) Select **Open** and then **OK** to import the image.

3) Next, JCSS enters Map/Background Editing mode. In this mode, you can edit image maps and backgrounds, but not border maps. To select/move/resize image maps or backgrounds, simply click, right-click or drag them. You can also copy, cut and paste maps/backgrounds.

   After you are done, exit Map/Background Editing mode by right-clicking on one of the maps or backgrounds and choosing **Exit Map Editing Mode**, or by selecting **View > Background > Map Edit Mode**.

Background > Add CADRG/CIB Raster Catalog Directories

View > Background > Add CADRG/CIB Raster Catalog Directories...: Select a raster catalog directory from which to import.

Background > Add MrSID Image

View > Background > Add MrSID Image...: Select a MrSID image file (*.sid) to import.

Background > Map Edit Mode

View > Background > Map Edit Mode: Modify background images. Once this mode is selected, edit the image as desired using techniques described in the following table.

Table 3-1 Edit Image Map Operations

To...	Follow These Steps...
Resize an image map	Right-click on the image map and drag one of the handles on the image map. Right-click on the image map and choose **Set Location and Size** and fill in the desired values.
Move an image map	Drag the object to the desired location. Right-click on the image map and choose **Set Location and Size** and fill in the desired values.
Remove an image map from the workspace	Click on the image map and press the **<Delete>** key.
Exit map edit mode	Right-click on the image map and select **Exist Map Editing Mode**.

Edit Display Preferences

View > Edit Display Preferences: Edit preferences that control the visualization of the network. Preferences can be controlled on either the global or subnet level. Those preferences specified at the subnet level take precedence over any global settings. The subnet preferences are retained per subnet and can only be changed from this dialog box for the active subnet.

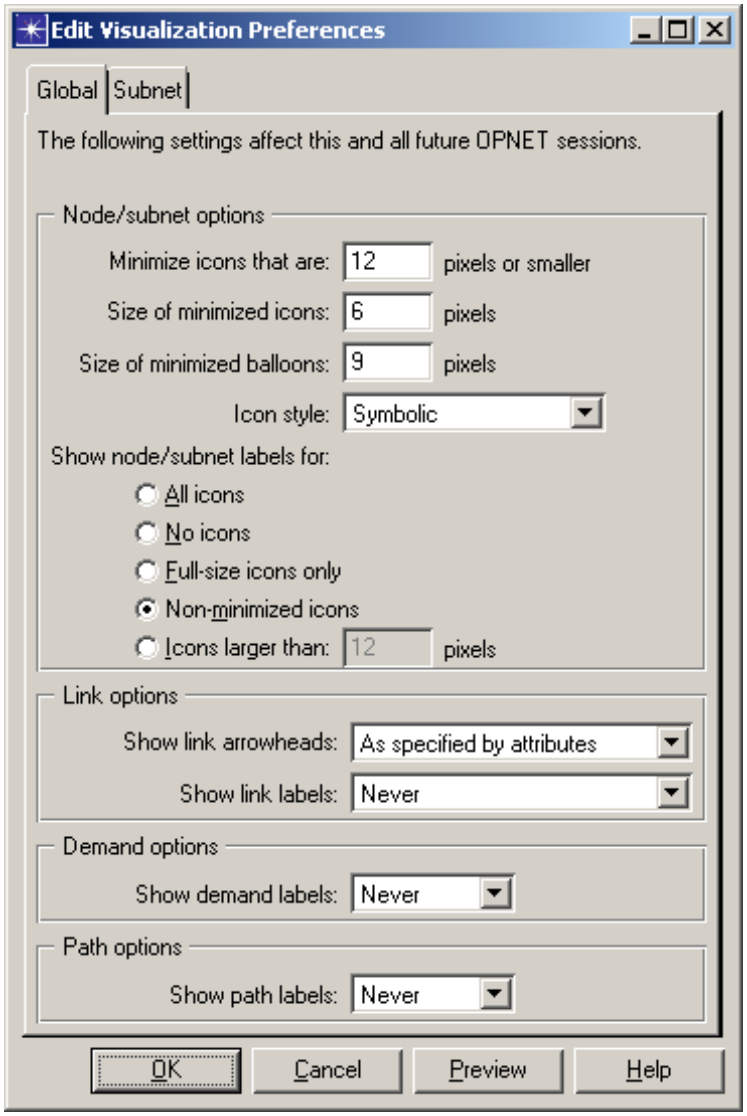


Figure 3-50 Edit Visualization Preferences dialog box - Global tab

Global preferences are available on the Global tab:

- **Minimize icons that are X pixels or smaller**: Determines at what size icons become minimized. If an object appears smaller than the specified number of pixels, it is drawn using its minimized icon, as specified by its minimized icon attribute. This setting also changes the network_visualization.minimized_icon_pixel_threshold environment preference.

- **Size of minimized icons**: Determines the size, in pixels, of all minimized icons in the network. It also changes the network_visualization.minimized_icon_size environment preference.
- **Size of minimized balloons**: Determines the size, in pixels, of all minimized balloons in the network. It also changes the network_visualization.minimized_balloon_size environment preference.
- **Icon style**: Specifies the model icon style to display. Select **Symbolic** or **Chassis**.
- **Show node/subnet labels for**: Specifies which nodes and subnet objects display labels. It also changes the network_visualization.show_node_labels environment preference. The selections have the following effects:
  - **All icons**: Labels appear on all objects.
  - **No icons**: No labels appear on any node or subnet objects.
  - **Full-size icons only**: Labels only appear on objects that have not been scaled below their original full size.
  - **Non-minimized icons**: Labels appear on any icons that have not been minimized.
  - **Icons larger than X pixels**: Labels appear on any icons that are larger than the specified number of pixels. It also changes the network_visualization.show_labels_icon_pixel_threshold environment preference.
- **Show link arrowheads**: Specifies which links have arrowheads. It also changes the network_visualization.show_link_arrowheads environment preference. The selections have the following effects:
  - **Never**: No arrowheads appear on any link objects.
  - **As specified by attributes**: The appearance of arrowheads is determined by each link object's arrowheads attribute.
  - **For simplex links only**: For all simplex links, the appearance of arrowheads is determined by each link object's arrowheads attribute. For all other link types, no arrowheads display.
- **Show link labels**: Specifies which links have labels. It also changes the network_visualization.show_link_labels environment preference. The selections have the following effects:
  - **Never**: No link labels appear on any link objects.
  - **Always**: Link labels appear on all link objects.
  - **For off-page links only**: Link labels appear on links that are drawn out of the active subnet.

- **Show demand labels**: Specifies which demands have labels. It also changes the network_visualization.show_demand_labels environment preference. The selections have the following effects:
  - — **Never**: No demand labels appear on any demand objects.
  - — **Always**: Demand labels appear on all demand objects.
- **Show path labels**: Specifies which paths have labels. It also changes the network_visualization.show_path_labels environment preference. The selections have the following effects:
  - — **Never**: No path labels appear on any path objects.
  - — **Always**: Path labels appear on all path objects.

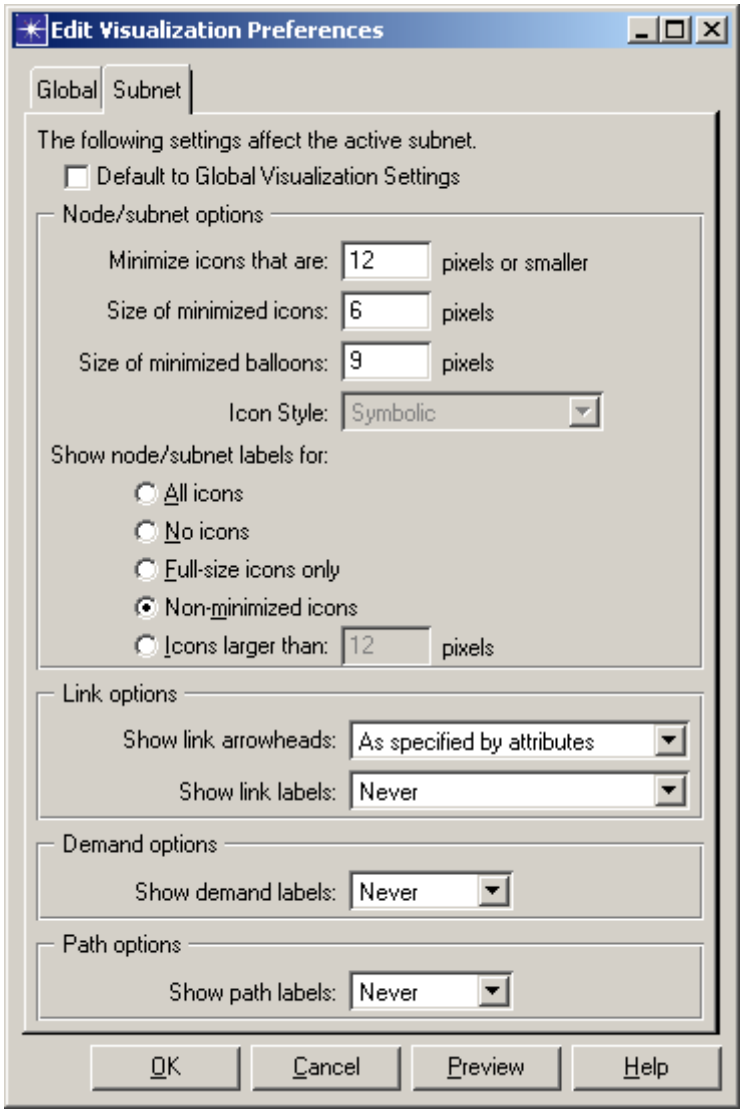


Figure 3-51 Edit Visualization Preferences dialog box -Subnet tab

Subnet preferences are available on the Subnet tab:

- **Default to Global Visualization Settings** checkbox: Determines if the subnet inherits the global preferences, or uses the subnet settings specified in the dialog box. If checked, the subnet-level dialog box controls are disabled. If unchecked, the subnet settings can be specified for the active subnet.
- **Minimize icons that are X pixels or smaller**: Determines at what size icons become minimized in the subnet. If an object appears smaller than the specified number of pixels, it is drawn using its minimized icon, as specified by its minimized icon attribute.
- **Size of minimized icons**: Determines the size, in pixels, of all minimized icons in the active subnet.
- **Size of minimized balloons**: Determines the size, in pixels, of all minimized balloons in the active subnet.
- **Icon style**: Specifies the model icon style to display. Select **Symbolic** or **Chassis**.
- **Show node/subnet labels for**: Specifies which nodes and subnet objects display labels in the active subnet.
  - **All icons**: Labels appear on all objects.
  - **No icons**: No labels appear on any node or subnet objects.
  - **Full-size icons only**: Labels only appear on objects that have not been scaled below their original full size.
  - **Non-minimized icons**: Labels appear on any icons that have not been minimized.
  - **Icons larger than X pixels**: Labels appear on any icons that are larger than the specified number of pixels.
- **Show link arrowheads**: Specifies which links have arrowheads in the active subnet. The selections have the following effects:
  - **Never**: No arrowheads appear on any link objects.
  - **As specified by attributes**: The appearance of arrowheads is determined by each link object's arrowheads attribute.
  - **For simplex links only**: For all simplex links, the appearance of arrowheads is determined by each link object's arrowheads attribute. For all other link types, no arrowheads display.
- **Show link labels**: Specifies which links have labels in the active subnet. The selections have the following effects:
  - **Never**: No link labels appear on any link objects.
  - **Always**: Link labels appear on all link objects.
  - **For off-page links only**: Link labels appear on links that are drawn out of the active subnet.

- **Show demand labels:** Specifies which demands have labels in the active subnet. The selections have the following effects:
  - — **Never**: No demand labels appear on any demand objects.
  - — **Always**: Demand labels appear on all demand objects.
- **Show path labels**: Specifies which paths have labels in the active subnet. The selections have the following effects:
  - — **Never**: No path labels appear on any path objects.
  - — **Always**: Path labels appear on all path objects.
- **OK** button: Closes the dialog box. All settings take effect immediately, including modification of global environment preferences and subnet-level settings.
- **Cancel** button: Closes the dialog box and returns to the original display without making changes to the visualization settings, global environment attributes, or subnet-level settings.
- **Preview** button: All of the settings changed take effect immediately, although the environment preferences and subnet settings are not retained unless you click **OK**.

Refresh Workspace

View > Refresh Workspace: Redraw/refresh the workspace view.

Layout

View > Layout > (*option*): You can customize the appearance of objects in your workspace by scaling the icons to a larger or smaller size, and moving icon labels that overlap. JCSS provides commands that will perform these functions automatically for you, or you can make adjustments manually to selected icons using the Scale Selected Icons dialog box.

Layout > Automatic Icon Scaling

View > Layout> Automatic Icon Scaling: Toggle on and off the ability to adjust icon sizes automatically to minimize overlap.

Layout > Automatic Label Placement

View > Layout> Automatic Label Placement: Toggle on and off the ability to adjust label positions automatically to prevent overlap with adjacent icons.

Layout > Scale Node Icons Interactively

View > Layout > Scale Node Icons Interactively: Scale/change the size of node/subnet icons. For example, if an icon is 12 x 12 pixels at full size, you can scale the icon to 50% so that it appears at half-size (6 x 6 pixels).

If you scale an icon below a certain size, the icon becomes minimized—that is, it appears as a simple shape rather than as a bitmap.

1) Choose **View > Layout > Scale Node Icons Interactively**. The Scale All Icons dialog box displays.

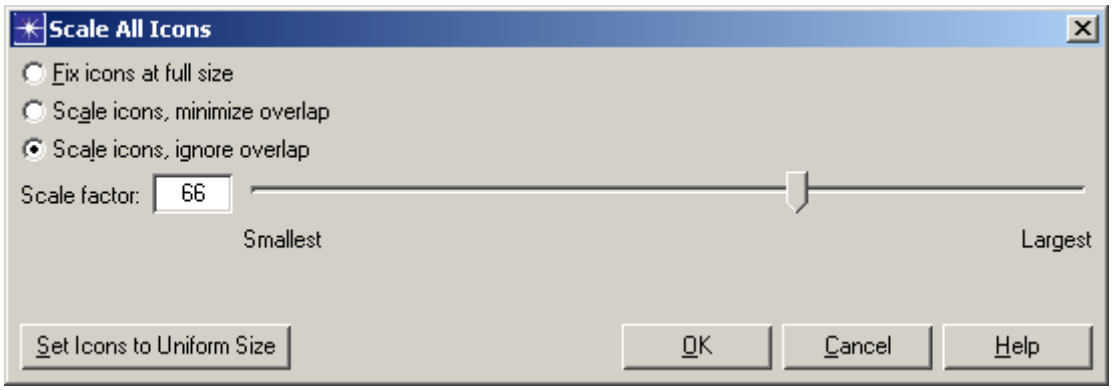


Figure 3-52 Scale All Icons dialog box

2) Scale the selected icons by the desired amount.

   a) Select the **Fix icons at full size** radio button to scale selected icons to their full size (as defined by the icon bitmaps). When zooming in or out in the workspace, the icons remain at full size regardless of the zoom level. This behavior is achieved by setting the "threshold" attribute on the selected objects to a value of 0. Selecting this option disables the icon scaling slider and the uniform size button.

   b) Select the **Scale icons, minimize overlap** radio button to use the Scale Factor slider and field to scale all of the selected icons, except those that would overlap other icons if they were scaled any larger. The size can range from 1% to the percentage at which all of the icons will appear full size. This maximum percentage will vary, depending on the icon size when you open this dialog box.

   c) Select the **Scale icons, ignore overlap** radio button to use the Scale Factor slider and field to scale all of the selected icons equally, even those that overlap. The size can range from 1% to the percentage at which all of the icons will appear full size. This maximum percentage will vary, depending on the icon size when you open this dialog box.

   d) To scale all selected icons to the same size (that is, to the current size of the smallest icon in the selection set), click the **Set Icons to Uniform Size** button and then use the slider. A progress bar appears as the icons are updated.

3) Click **OK** to change the selected icons as specified and close the dialog box. A progress bar appears as the icons are updated. If no changes are necessary, the dialog box will simply close.

4) Click **Cancel** to close the dialog box without making any of the specified changes.

Layout > Lay Out Nodes Interactively

View > Layout > Lay Out Nodes Interactively...: Use this option to change the position of various node/subnet/utility objects.

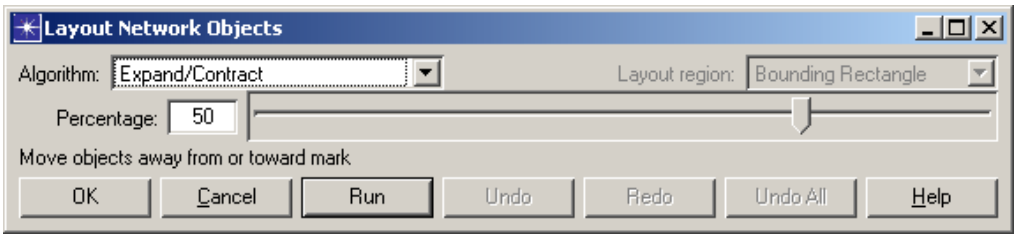


Figure 3-53 Layout Network Objects dialog box

- **Algorithm** pull-down menu: Specifies the currently selected layout algorithm. The layout algorithm determines how the positions of the selected objects will change when the Run button is clicked. The algorithms are:
  - — **Expand/Contract**: Moves the selected network objects away from or toward the Mark icon (the movable green cross icon in the Project Editor workspace).
  - — **Expand/Contract Horizontally**: Moves the selected network objects horizontally away from or toward the Mark icon.
  - — **Expand/Contract Vertically**: Moves the selected network objects vertically away from or toward the Mark icon.
  - — **Rotate**: Moves the selected network objects as if they were rotated about the Mark icon.
  - — **Disperse**: Moves the selected network objects with the specified layout region.
- **Layout region** pull-down menu: Specifies a rectangular region for the algorithm to operate on. This element will be disabled if the selected algorithm does not use a particular layout region.
- **Percentage/Degrees** combo field: Specifies a variable parameter to customize the behavior of the selected layout algorithm. The label changes between Percentage and Degrees, depending on the selected algorithm. You can either drag the slider or enter a number directly to specify the value. This is disabled, if the selected algorithm does not use a parameter value.
- **OK** button: Closes the dialog box. The position changes become permanent and a single undo command is added to the editor's undo stack.
- **Cancel** button: Closes the dialog box after undoing all position changes that were made while this dialog box was open.
- **Run** button: Performs the selected layout algorithm upon the current selection set or, if no objects are selected, all objects in the current subnet.

- **Undo** button: Undoes the effect of the last algorithm run or the last object drag. The dialog box maintains a local undo stack since the time it was opened, so you can undo repeatedly to return to any earlier state. This button is disabled if there are no operations that can be undone.
- **Redo** button: Redoes the effect of the last local undo operation. Until you run another algorithm or directly drag objects, you can redo repeatedly to return to any previously undone state. This button is disabled if there are no operations that can be redone.
- **Undo All** button: Undoes the effect of all algorithm runs and object drags since the dialog box was opened. This has the same effect as repeatedly clicking the **Undo** button until it becomes disabled.

Note—While the Layout Network Objects dialog box is open, the toolbar and the menu bar of the editor window are disabled. However, the editor workspace is still active. You can change the selection set, and you can directly drag selected objects around. Any drags are considered part of the interactive layout operation and will be undone if you cancel out of this dialog box. Also, while this dialog box is open, the Mark icon (a green cross) is present at the center of the current selection set. This icon represents the Mark point used by various algorithms. You can drag this icon around without dragging any network objects and without changing the current selection set.

Layout > Layout Nodes (Balanced)

View > Layout> Layout Nodes (Balanced): Selecting this option automatically balances the layout of nodes in the current view.

Layout > Layout Nodes (Simple)

View > Layout> Layout Nodes (Simple): Selecting this option automatically simplifies the layout of nodes in the current view.

Layout > Layout Nodes (Core Centric)

View > Layout> Layout Nodes (Core Centric)...: Use this option to create a meaningful network layout, based on core nodes. The layout places core nodes in the center. All other nodes are placed around the core nodes based on the quantity of links. Nodes attached to the network with only one link are placed furthest from the core nodes. Nodes with no link attachments to the network are placed on the left side of the workspace.

Layout > Geographic Positioning

View > Layout> Geographic Positioning: Toggle on and off the current view mode to logical/physical.

Zoom > To Rectangle

View > Zoom > To Rectangle or press **<Ctrl>**+**<I>**, or use the **Zoom In** option from the shortcut menu, or click the **Zoom In** toolbar button: Get a close-up view of the object in question in the workspace. Apply this option multiple times to obtain an ever-closer view.

1) Select the **Zoom > To Rectangle** command, and then drag the cursor to define a rectangular shaped region to be magnified.

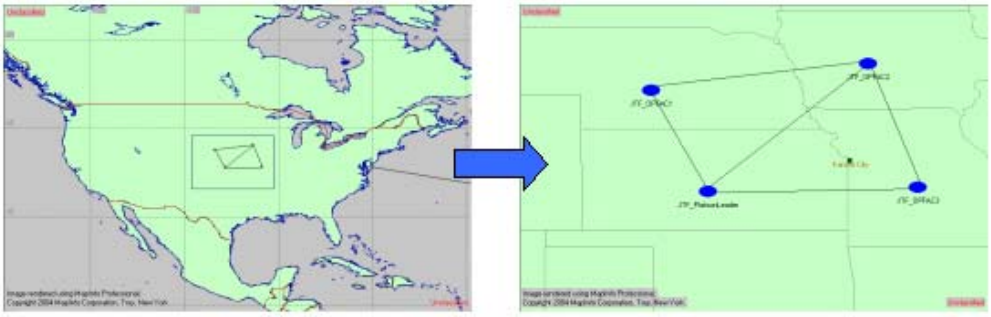


Figure 3-54 Selecting an area to zoom in

Zoom > To Selection

View > Zoom > To Selection or press **<Ctrl>**+**<Shift>**+**<Z>**: Get a close-up view of a selected area or object in the workspace.

1) Drag the cursor to define a rectangular shaped region to be magnified, and then select the **Zoom > To Selection** command.

Zoom > To Window

View > Zoom > To Window: Zoom out on the workspace so it shows completely in the window.

Zoom > Unzoom

View > Zoom > Unzoom or press **<Ctrl>**+**<U>**, or use the **Zoom Out** option from the shortcut menu, or click the **Zoom Out** toolbar button: Restore the workspace view to the previous zoom level.

Zoom > To All

View > Zoom > To All or press **<Ctrl>**+**<Alt>**+**<Z>**: Zoom out on the workspace so it shows active areas in the window.

Filter

View > Filter > (*option*): Apply a number of different filters to the scenario.

Filter > Selected Objects

View > Filter > Selected Objects: Filters can be applied on objects that have been selected on the workspace. The objects could have been selected by manually clicking on them, or by the use of the selection filter.

Figure 3-55 Filter Map dialog box

Using this option, you can hide the selected objects or show only the selected objects. An option is also provided to redisplay all objects.

Filter > Devices

View > Filter > Devices: Filters can be applied to devices based on the type of applications they support. Filtering can be done based on the functional area or the service. For example, if you are interested in viewing just the SIPRNET portion of the scenario, you can apply a device filter and specify SIPRNET as the service of interest. All other devices will be hidden. If an OPFAC has no visible devices, the OPFAC is hidden as well.

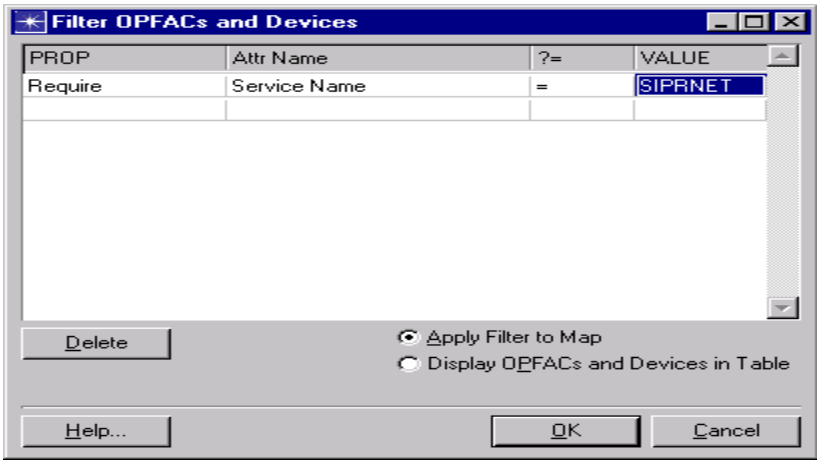


Figure 3-56 Filter OPFACs and devices dialog box

You can display the result of this filtering operation either on the workspace or in a table. The table displays the OPFAC name, device name, application name, functional area and service.

Filter > Infrastructure

View > Filter > Infrastructure: Show or hide the different types of infrastructure like relationships, links, broadcast networks and satellite links. By default, all infrastructures are displayed and all the options are checked. If you want to hide a certain type of infrastructure, uncheck that option and all infrastructure of that type will be hidden.

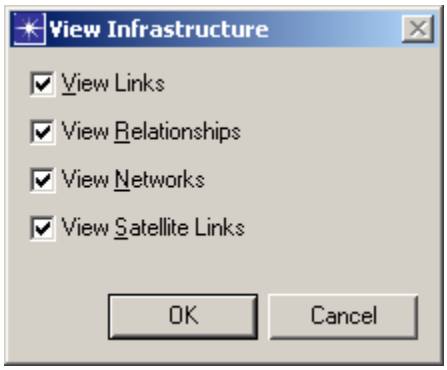


Figure 3-57 View Infrastructure dialog box

For example, if you do not want to view relationships in the scenario, you can go to this interface and uncheck the option **View Relationships**.

Filter > Hide Locked Units

View > Filter > Hide Locked Units: Hide all locked units in the Subordinate Planner query file.

Filter > Restore Full View

View > Filter > Restore Full View: View all locked units. In the Subordinate Planner query file, all locked units are hidden by default.

Network Views

View > Network Views > (*option*): JCSS offers a number of different views of the scenario. Each of these views offers the user a different perspective of the network.

Network Views > Planning

View > Network Views > Planning:This view is useful for someone working in a collaborative planning environment. Using this view, the planner can display only the planning portions of the scenario and hide all others. Organizations, OPFACs and planning links are all part of the planning view. Devices and connectivity links are hidden in this view.

Network Views > Connectivity

View > Network Views > Connectivity: If a planner is not interested in the planning links and just wants to view the connectivity links in the network, he can choose this menu option. Planning links and OPFACs without any devices are hidden in this view.

Network Views > Planning and Connectivity

View > Network Views > Planning and Connectivity:The planner can use this menu option to return to the default view, which includes the planning and connectivity elements in the scenario.

Network Views > Operational

View > Network Views > Operational: Selecting this menu item will switch from a Systems or Logical View back to the default view of the scenario, the same view that loads when you first open the scenario in Scenario Builder.

Network Views > OPFAC Systems

View > Network Views > OPFAC Systems: Selecting this menu item will switch from the current view, the Operational View or some other type of Systems View, to the OPFAC Systems View. The OPFAC Systems View expands the contents of the organizations, collapses the contents of the OPFACs, and redraws the links to connect to their respective devices.

Save Current Unit View

View > Save Current Unit View: Set a default view to which you can return.

Restore Saved Unit View

View > Restore Saved Unit View: Return to the saved view.

Show LOS > LOS Settings

View > Show LOS > LOS Settings: Set the parameters for the editor to control what LOS visualizations are displayed and how they appear.

Note—The LOS that you see takes into account the position as well as the altitude for that device. You can set the altitude for an OPFAC via the OPFAC Attributes dialog box (right-click on the OPFAC and select **Edit NETWARS Attributes**.) By default, altitude is set by terrain and measured in meters. You can enter an altitude value in another unit of measurement (such as kilometers, feet, yards, or miles), and JCSS will automatically convert the measurement to meters the next time you open the OPFAC Attributes dialog box.

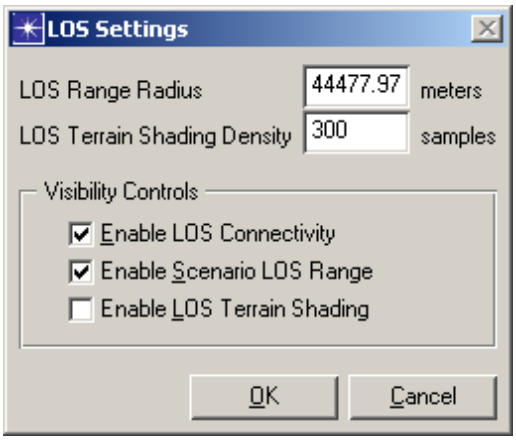


Figure 3-58 LOS Settings

- **LOS Range Radius** – Measured in meters, this specifies the size of the circle that may appear around an organization or OPFAC. It is a nominal range, and this single setting affects all LOS Visualization in the editor. For reference, 10,000 meters is the typical transmission limit when terrain elevation data is not available. Transmitters assisted by elevation could transmit many times this distance.

- **LOS Terrain Shading Density** – Specifies the number of samples to check within the LOS Range Radius for determining LOS Terrain Effects Shading. More samples give a more accurate representation, but can also slow display as more information must be considered to update Terrain Effects Shading.

- **Enable LOS Connectivity** – Allows the editor to display LOS Connectivity links when selected for organizations or OPFACs. Enabled by default.

  LOS Connectivity is a thick, dashed, double-headed arrow. The color of the arrow indicates whether the two objects could communicate. Connectivity is bi-directional and independent of specific device characteristics. Connectivity is based on signal loss between the transmitter and receiver positions as determined by the Longley-Rice signal propagation model.

  LOS Connectivity exists between OPFACs. Organizations that contain OPFACs may display LOS Connectivity links when they are collapsed. Enabling LOS Connectivity on an organization enables LOS Connectivity on all subordinate organizations and OPFACs. An organization that contains no subordinate OPFACs will not display any LOS Connectivity links (it would not be able to join a broadcast network or connect via point-to-point links either).

  LOS Connectivity works through broadcast networks and point-to-point transmission links to determine all of the OPFACs to which a selected Organization or OPFAC could possibly communicate and designates the state of that communication link. LOS Connectivity links are attributes of their end points. The existence of LOS Connectivity links indicates that one or both of the objects at an end of the link has LOS Connectivity enabled. LOS Connectivity links disappear when neither end point has LOS Connectivity enabled, in either the top-level organization or any subordinate organization or OPFAC. LOS Connectivity link coloration is automatically updated when either end point moves for any reason, including mouse dragging, attribute changes, or animation, to reflect the LOS closure state at the new location.

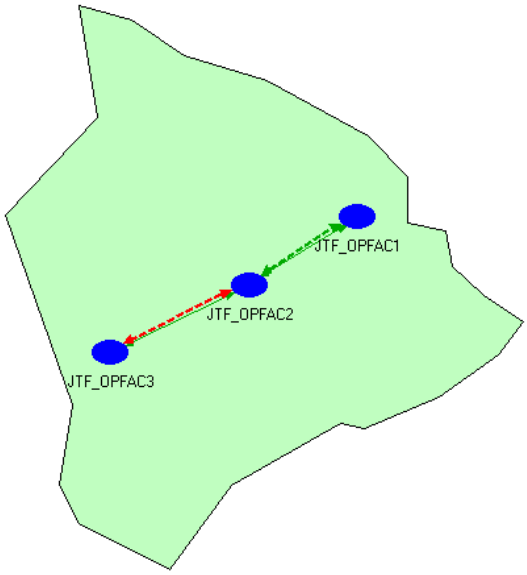


Figure 3-59 LOS Connectivity

- **Enable Scenario LOS Range** – Allows the editor to display LOS Range circles and Terrain Effects Shading for selected organizations or OPFACs. Enabled by default.

  LOS Range is a circle of a fixed radius placed around the organization or OPFAC in question. The circle represents a ground distance and is used as a reference for the distance from a particular object. There is a single setting for the range that pertains to all LOS Ranges displayed in the entire scenario. The LOS Range is a nominal indicator, having no direct connection to a particular device's transmission range. The LOS Range does indicate the limit of the area within which locations may be sampled for determining LOS Terrain Shading.

  Any organization or OPFAC may display an LOS Range without regard for the type of device contained within or even if an OPFAC or device is contained. The LOS range simply indicates a distance.

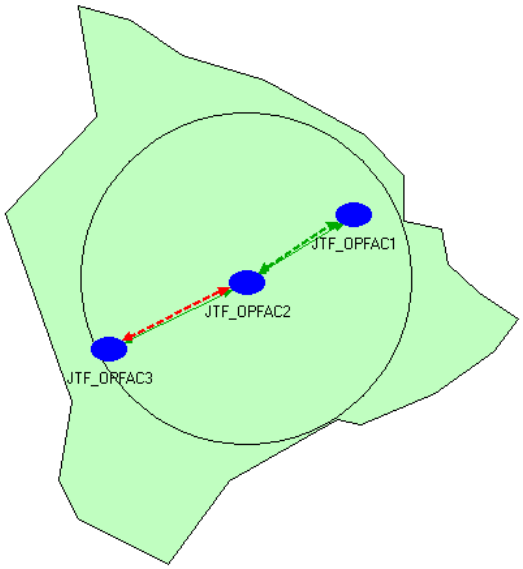


Figure 3-60 LOS Range

- **Enable LOS Terrain Shading** – Allows the editor to display Terrain Effects Shading if LOS Range is also enabled. Disabled by default.

  Terrain Effects Shading indicates portions of the area around an organization or OPFAC where a radio receiver could not receive a hypothetical transmission from the object.

  Since the LOS Range defines the limit of the area where LOS Terrain Shading is tested, LOS Range must be enabled to view LOS Terrain Shading. Disabling LOS Range on an object also disables LOS Terrain Shading.

  Like LOS Range, LOS Terrain Shading does not indicate anything about the specific OPFACs or devices contained within the designated object. It simply indicates that if the object contained a radio transmitter, that radio transmitter would or would not be able to communicate with a generic radio device at a particular location.

LOS Terrain Shading works by testing for LOS closure at a predetermined number of sample locations evenly spaced throughout the circle formed by the LOS Range. You can select the number of samples to take.

LOS Terrain Shading updates when the selected object moves for any reason, whether by mouse dragging, attribute change, or animation.

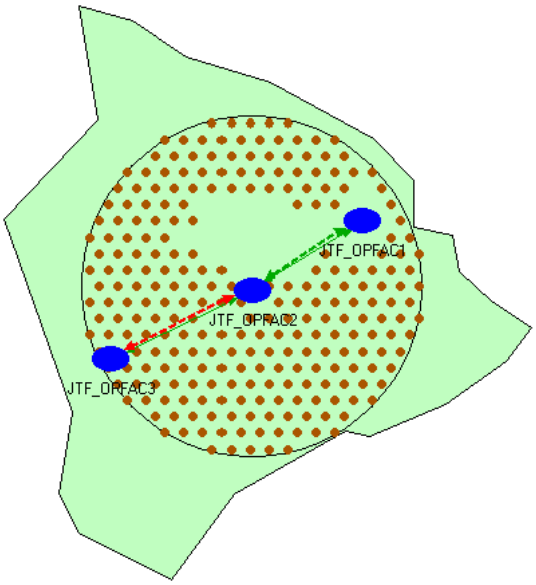


Figure 3-61 Terrain Effects Shading

Show LOS > Clear LOS Ranges

View > Show LOS > Clear LOS Ranges: Clears LOS range circles and terrain effects shading for all units.

Show LOS > Display LOS Legend

View > Show LOS > Display LOS Legend: Display the LOS Legend, which explains the color-coding of LOS Connectivity as well as LOS Ranges and Terrain Shading. The Auto-Display Legend checkbox determines whether the LOS Legend will automatically appear following the activation of LOS visualization. Clearing the checkbox will prevent this automatic display.

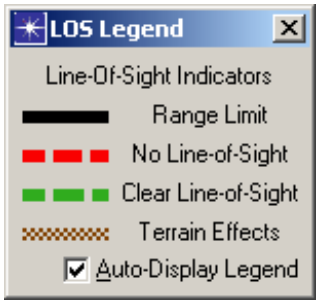


Figure 3-62 LOS Legend

OPFAC Distances > View OPFAC Distances

View > OPFAC Distances > View OPFAC Distances: Create OPFAC Distance Links (LOS Connectivity Links with a label showing the distance between each pair of OPFACs) between all selected OPFACs.

OPFAC Distances > Remove OPFAC Distances

View > OPFAC Distances > Remove OPFAC Distances: Remove any OPFAC Distance Links currently existing in the scenario.

Demands

View > Demands > *(option)*: Sub-menu allows you to select whether or not you want to show or hide all demands from the view.

Circuits

View > Circuits > *(option)*: Sub-menu allows you to select whether or not you want to show or hide all circuits from the view.

Paths

View > Paths > *(option)*: Sub-menu allows you to select whether or not you want to show or hide all paths from the view.

Wireless Domains

View > Wireless Domains > *(option)*: Sub-menu allows you to select whether or not you want to show or hide all wireless domains from the view.

Annotations

View > Annotations > *(option)*: Sub-menu allows you to select whether or not you want to show annotations in the subnet, and show, hide, minimize or clear annotation balloons from the view.

Visualize Protocol Configuration

View > Visualize Protocol Configuration > (*option*): Control the display of protocol configuration using the following options:

- **IP Interface Status** – Shows whether the connected interface is active or shutdown. A green up arrow indicates that interface is active whereas a red down arrow indicates that the interface is shutdown.
- **IP Routing Protocols** – Shows the routing protocols that are configured on the router interfaces.
- **IP Address Types** – Shows the IP address types on the network by color, along with a legend to explain the color-coding.
- **IP QoS Configuration** – Shows the IP QoS configuration on the links.
- **IP Security Configuration** – Shows the packet filtering security configured on the links.
- **IP Tunnel Configuration** – Shows the router tunneling configured on the network.
- **BGP Peers** – Shows the BGP peering (neighbor) relationships in the network. EBGP confederation peers are shown in dark green, EBGP peers are shown in blue, all other colors indicate IBGP peers. You can view the tool tip of a peering visualization for more information about the peers. For networks using route reflection, the visualization also indicates if a router is a reflector or a client.
- **OSPF Area Configuration** – Shows the OSPF areas configured on the router interfaces.

- **ATM Routing Domains** – Shows which ATM routing protocols that are configured on the switch ports.
- **VLAN Configuration** – Shows the VLANs configured in the network.
- **Clear Visualization** – Removes all protocol configuration visualizations from the workspace.

Visualize Link Loads

View > Visualize Link Loads > (*option*): Control the display of link loads using the options discussed below.

Visualize Link Loads > Settings

View > Visualize Link Loads > Settings: Designate your choice of link load link colors and thickness.

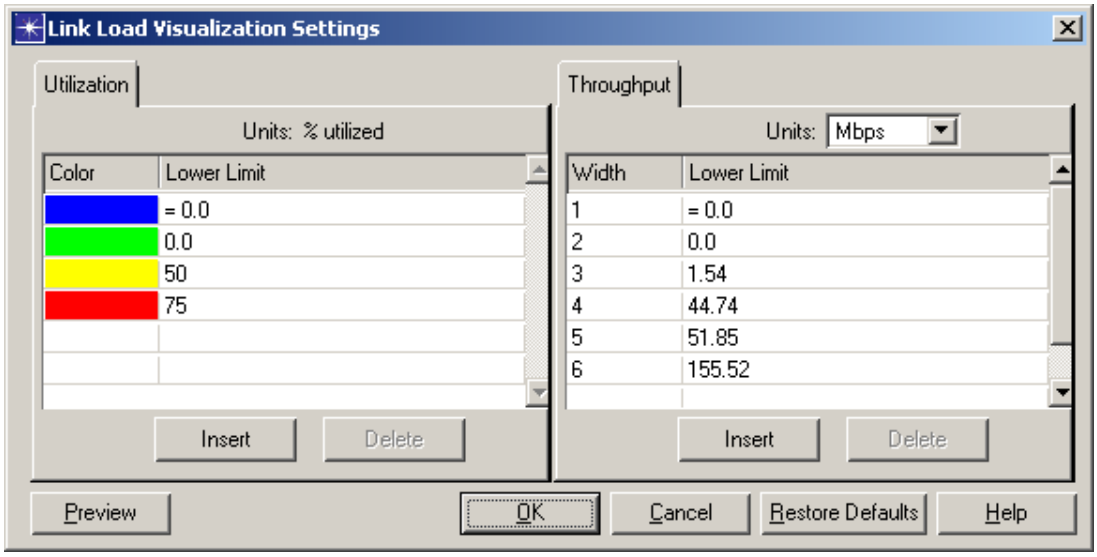


Figure 3-63 Link Load Visualization Settings dialog box

- **Color Settings table** – Lists the colors to use when visualizing link loads, and shows the utilization value that corresponds to each color. The Lower Limit number represents the percentage of the link that is utilized.
- **Width Settings table** – Lists the link thickness to use when visualizing link loads, and shows the throughput value that corresponds to each link thickness.
- **Preview button** – Applies the current settings to the network. This button appears only if link load visualization is enabled.
- **Restore Defaults button** – Resets the dialog box to the default link visualization settings.

Visualize Link Loads > Color by Link Load

View > Visualize Link Loads > Color by Link Load: Set the link colors and thickness based on the link's peak, average, or current utilization (depending upon which one you choose in the **Color links based on data from** and **using the** drop-down list boxes) and throughput statistics respectively.

The recorded statistic values and the peak time are reported in the display that appears when you move your mouse over any link. These statistics were collected from a Capacity Planning Evaluation. The link statistics are computed taking into consideration the entire duration of the Capacity Planning Evaluation.

You must first run a Capacity Planning Evaluation before you can visualize utilization.

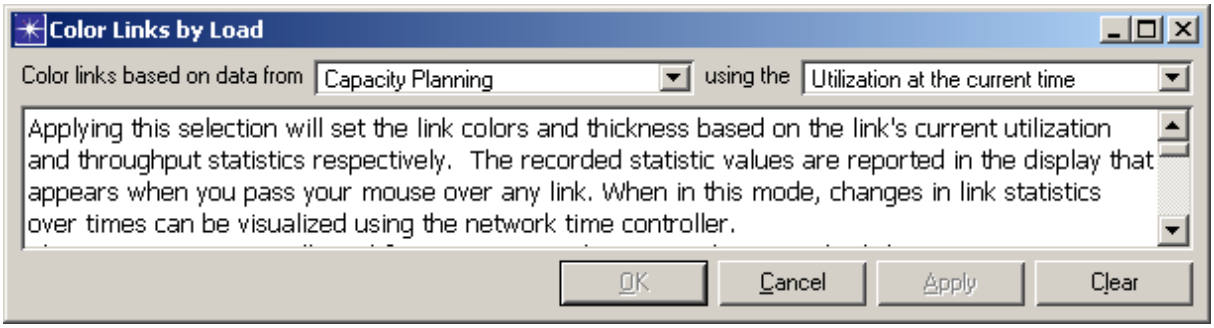


Figure 3-64 Color Links by Load dialog box

Visualize Link Loads > Clear Visualization

View > Visualize Link Loads > Clear Visualization: Removes all link load visualizations from the workspace.

Visualize Link Loads > Show Legend

View > Visualize Link Loads > Show Legend: Displays a legend that explains the current link load visualization settings.

Scenarios Menu

New Scenario

Scenarios > New Scenario or press **<Ctrl>+<Shift>+<N>**: Create a new blank scenario.

1) Enter a name for the new scenario in the New Scenario Name field, and then click **OK**.

Figure 3-65 Creating a New Scenario

The new scenario is blank and contains no data. The Scenario Builder indicates in the title bar the current open scenario.

Duplicate Scenario

Scenarios > Duplicate Scenario or press **<Ctrl>+<Shift>+<D>**: Create an exact duplicate of an existing scenario under a new name.

1) Enter a name for the new scenario in the Duplicate Scenario Name field, and then click **OK**.

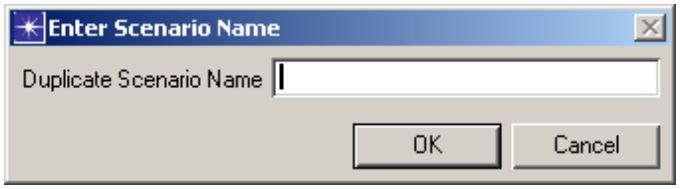


Figure 3-66 Duplicating a Scenario

The new scenario is an exact copy of the current scenario, which can serve as the baseline scenario. The Scenario Builder indicates in the title bar the current open scenario.

Manage Scenarios

Scenarios > Manage Scenarios: View each scenario in the project and its status. Buttons and drop-down menus enable you to do the following:

- Add new scenarios,
- Duplicate scenarios,
- Delete scenarios,
- Run a simulation,
- Change the simulation duration,
- Delete results, and
- Reorder scenarios in the list.

Click in each table cell to edit its contents. Brackets indicate a change to the scenario is requested, and will be applied when you click **OK**.

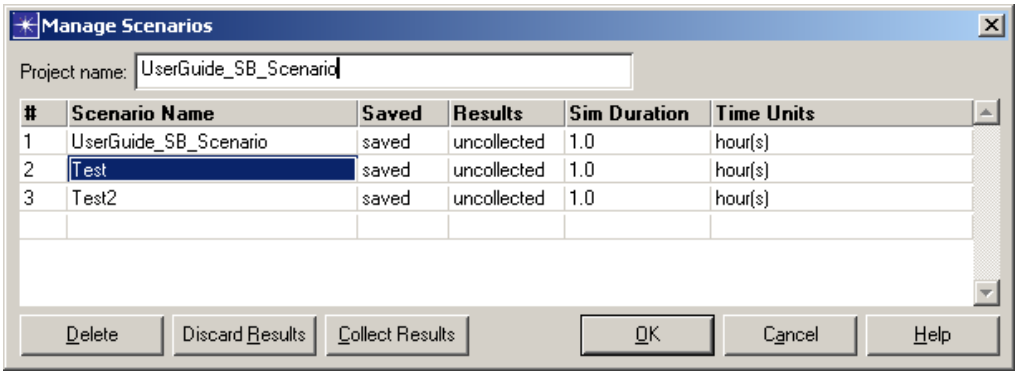


Figure 3-67 Manage Scenarios dialog box

- **Project Name** field—Saves the project under a different name. If you enter a new name and save your project, the project is saved under the new name. This is similar to the **File > Save Project As...** feature.Scenario Table columns:
  - — **#**: Changes the order of the scenarios in the table.
  - — **Scenario Name**: Lists the name of each scenario.
  - — **Saved**: Shows if a scenario is saved or unsaved. Clicking within a cell also lets you delete a scenario.
  - — **Results**: Shows the state of the results of each scenario simulation. “Uncollected” means that a probe file exists, but you have not run a simulation. “Out of date” means that the set of chosen results, simulation configuration, or network model has changed since the results file was created.
  - — **Sim Duration**: Specifies the length of time to run the simulation. Use it with the Time Units column.
  - — **Time Units**: Specifies the time units that apply to the simulation exam: seconds, minutes, hours, days, or weeks. Used with the Sim Duration column.
- **Delete**—Deletes scenarios that you select.
- **Discard Results**—Deletes results from a scenario.
- **Collect Results**— Runs the simulation and collects statistics you specify.
- **OK**—Closes the Manage Scenarios dialog box, and runs any simulations that are waiting.
- **Cancel**—Closes the Manage Scenarios dialog box without making changes.

Previous Scenario

Scenarios > Previous Scenario or press **<Ctrl>+<Up>**: Display the previous scenario in the project.

Next Scenario

Scenarios > Next Scenario or press **<Ctrl>+<Down>**: Display the next scenario in the project.

Switch To Scenario

Scenarios > Switch To Scenario: Switch between the scenarios associated with a project, as a project can have scenarios. A black bullet indicates the current scenario open in the Scenario Builder.

1) Select the desired scenario.

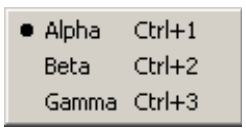


Figure 3-68 Switch between scenarios

When the project is saved, the scenario that is currently open is set as the default scenario.

Set Classification

Scenarios > Set Classification: Set the classification of a scenario. Every project in JCSS has a classification. The default classification is Unclassified.

Select a classification from a drop-down list or specify a user-defined classification. Once the classification is set, the classification string will always appear at the top-left and bottom-right hand corners of the workspace.

Set Scenario Documentation

Scenarios > Set Scenario Documentation: Access a free-form text box to enter scenario purpose, traffic, and topology notes to be stored with the scenario for documentation purposes. The scenario's Operational Scenario In Time (OPSIT) name is also entered here.

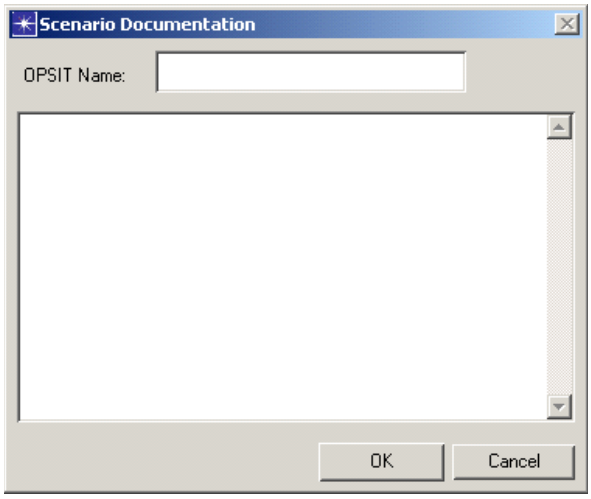


Figure 3-69 Scenario Documentation dialog box

1) Type desired text in the **Scenario Documentation** field.
2) Enter desired value in the **OPSIT Name** field.
3) Click **OK**.

Import Scenario

Scenarios > Import Scenario > (*option*): Import scenarios from the sources discussed below.

Import Scenario > Subordinate Response

Scenarios > Import Scenario > Subordinate Response: The Lead Planner imports subordinate response files from the Subordinate Planners into the original scenario.

1) Open the original project (or the subordinate query file if you are an Intermediate Planner.)

2) Select **Scenarios > Import Scenario > Subordinate Response**, and choose a response file received from a subordinate. The dialog box below displays.

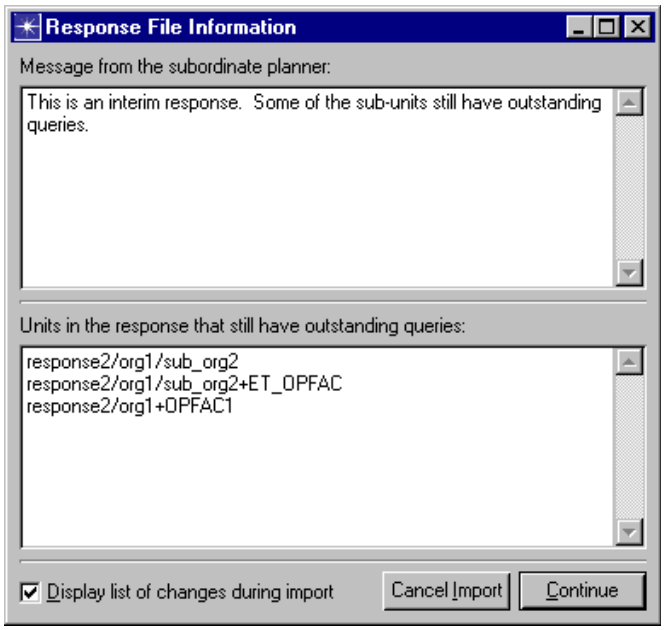


Figure 3-70 Response File Information dialog box

The edit area at the top of the dialog box will display the message that the Subordinate Planner entered when generating the response (this will contain the string “<none>” if the subordinate did not enter a message). The other edit area will display the hierarchical names of the units in the subordinate response that still have outstanding queries (once again this will contain the string “<none>” if there are no such units). These are shown to the Lead Planner because they are an indication that the response is an interim response. The **Display list of changes during import** checkbox controls whether or not the Changes dialog box will appear, as discussed later in this section.

3) Click the **Continue** button to proceed with the import. During the import, if the subordinate has made any changes to external associations (links, satellite links, and broadcast networks that connect an editable unit to a locked unit), the Lead Planner will be given options to import or ignore the changes. This will be accomplished through a treeview dialog box that breaks down the different types of external associations according to the type of change (add, delete, or modify).

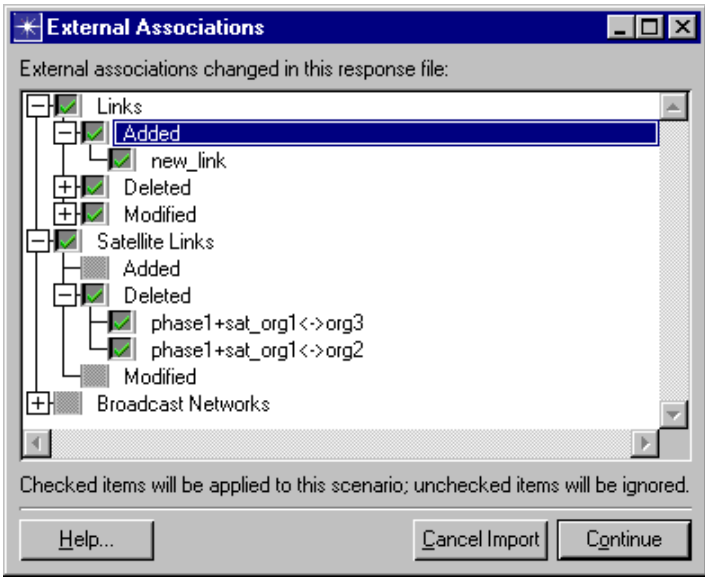


Figure 3-71 External Associations

Items that are checked off will be imported, and unchecked items will be ignored. All items will be checked by default when the dialog box opens. Note that this dialog box will not appear if no changes were made to external associations.

4) Proceed with the import by clicking the **Continue** button. If the **Display list of changes during import** checkbox was checked on the Message dialog box, the next step in the import process will be to display the Changes Made to the Original Scenario dialog box, which is shown below.

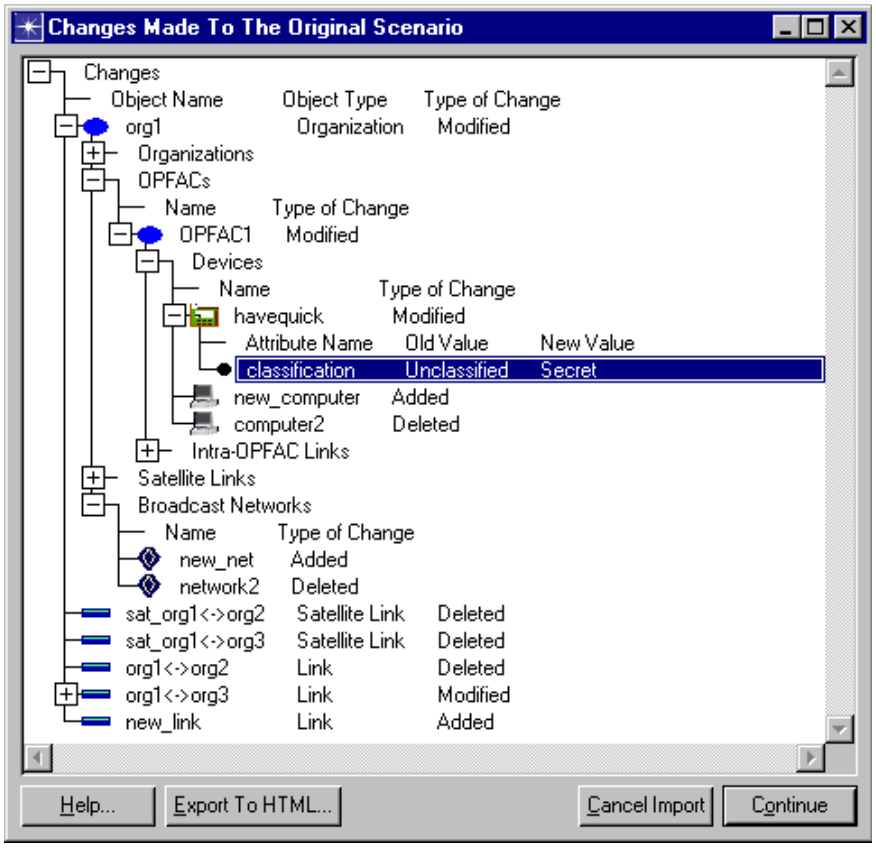


Figure 3-72 Changes Made to the Original Scenario

Note that this dialog box will display directly after the Response File Information dialog box if there were no modifications to external associations. Otherwise, it will display after clicking the **Continue** button on the External Associations dialog box. The main feature of this dialog box is a treeview that displays all of the changes that will be made to the Lead Planner's scenario if the import is completed. For each object that was changed by the subordinate, the treeview lists whether it was added, deleted, or modified. If it was modified, you will be able to expand the entry to see more information about the modifications that were made. For most objects, this will be a list of the attributes that changed, along with the old and new values. However, organizations and OPFACs are more complex since they can have child objects. So, expanding one of those entries will show a listing of the types of child objects that were changed, and inside each of those will be more information about the exact changes that were made to the child objects. The above example shows how the individual changes within an organization can be viewed.

a) The contents of the treeview can be exported to HTML by clicking the **Export To HTML** button. Clicking the **Cancel Import** button will stop the import process.

5) Click the **Continue** button to perform the actual import. After the import is complete, a "Subordinate Response import complete" message will be printed in the message buffer.

After this import process is repeated for each of the subordinate response files, the Lead Planner will have a consolidated view of the entire scenario based on the requirements specified by his subordinates.

Import Scenario > From NETWARS

Scenarios > Import Scenario > From NETWARS: Combine the organization structures of different scenarios automatically, thereby eliminating the need to duplicate another scenario's organization structure manually. The organization structure of the imported scenario is added to the top level of the current scenario.

1) In the Import Scenario dialog box, select the scenario for which the organization structure is to be imported.

2) Click **Open**. The newly added organization structure displays both in the treeview and in the workspace.

Import Scenario > From OPNET Modeler

Scenarios > Import Scenario > From OPNET Modeler: Import scenarios already developed in the OPNET Modeler and create the JCSS specific objects/hierarchy. The location information is intact in the newly created OPFACs and organizations. The new structure is below the newly created scenario to which the modeler scenario is imported.

1) Create a new Scenario.

2) Select **Scenarios > Import Scenario > From OPNET Modeler**.

3) In the subsequent Import Scenario from OPNET Modeler dialog box, select the scenario for which the organization structure is to be imported.

4) When prompted to save OPFACs, click **OK**. Saving the OPFACs is necessary to reuse the OPFACs in other scenarios and to run simulation. The newly added organization structure displays both in the treeview and in the workspace.

Import Scenario > From XML

Scenarios > Import Scenario > From XML: Import a scenario in XML format into JCSS.

1) Create a new project (or start with an already existing project).

2) Choose **Scenarios > Import Scenario > From XML**.

3) Select the XML file to be imported and then click **Open** to start the import process. At the end of this process, the scenario represented by the XML file has been imported into the current project.

Import Scenario > TNAPS to XML

Scenarios > Import Scenario > TNAPS to XML: Convert a TNAPS file into an equivalent JCSS XML representation of the network.

The converter reads transmission nodes, data network, circuit switch network, links, circuits, transmission node layout, site mux plan, site equipment inventory, mode and link properties/configurations, and ports. It does not read message switch network, local long circuits, unrouted circuits, circuits with sub-groups and sub-channels, and TNAPS annotations.

1) Select **Scenarios > Import Scenario > TNAPS to XML** to run the TNAPS converter. The TNAPS to NETWARS Converter dialog box displays.

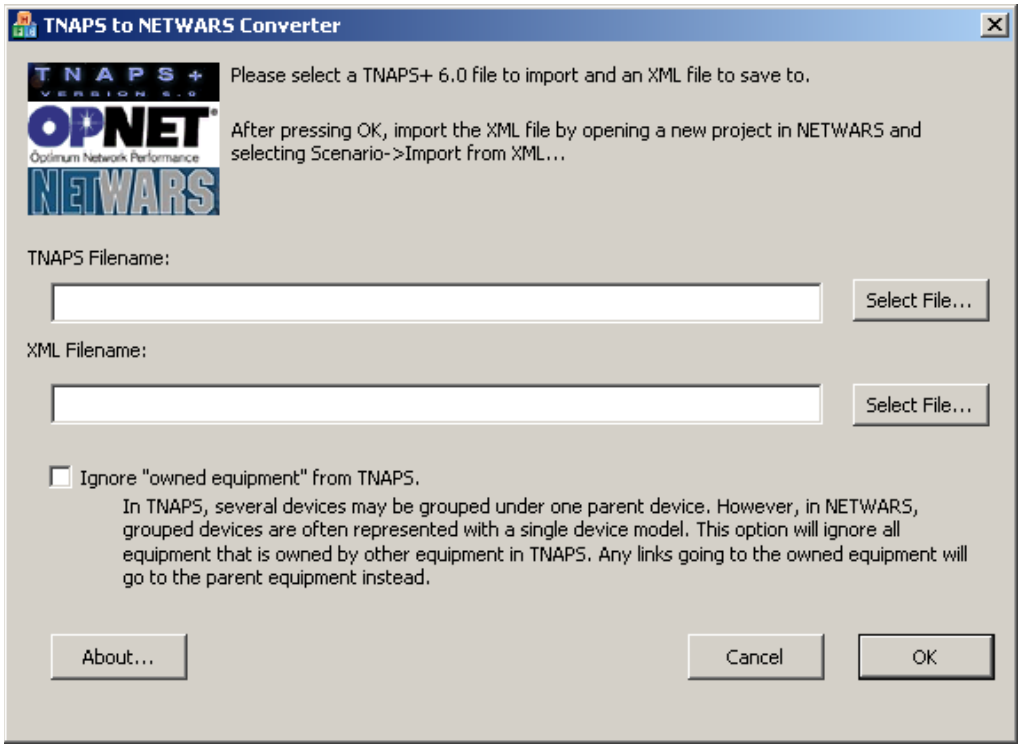


Figure 3-73 TNAPS to NETWARS Converter dialog box

2) Select a TNAPS 6.0 file to open and an XML filename to which you want to save, and then click **OK**.

   The TNAPS file converts into XML, and a message box displays to inform you of its success. Click **OK** to close the message box.

3) Now you can create a new project in the Scenario Builder and import the new XML file into it (select **Scenarios > Import Scenario > From XML**.)

Export Scenario

Scenarios > Export Scenario > (*option*): Export scenarios to the formats discussed below.

Export Scenario > Subordinate Query

Scenarios > Export Scenario > Subordinate Query: Export a subordinate query file and send it to Subordinate Planners to get the subscriber requirements. Various Subordinate Planners fill different portions of the scenario. The Lead Planner marks portions of the scenario to be editable by a Subordinate Planner by setting owner information on units. Units will be editable for the subordinate if the organization of the Subordinate Planner matches the organization field in the owner information. Otherwise, units will be locked; therefore, the Lead Planner must remember to set the correct owner information on units before exporting the subordinate query file.

1) If the scenario contains any units that do not have owner information set, the planner will be warned and given a chance to set the owner information before the subordinate query file is created. The dialog box shown below contains un-owned units in the current scenario in a treeview format.

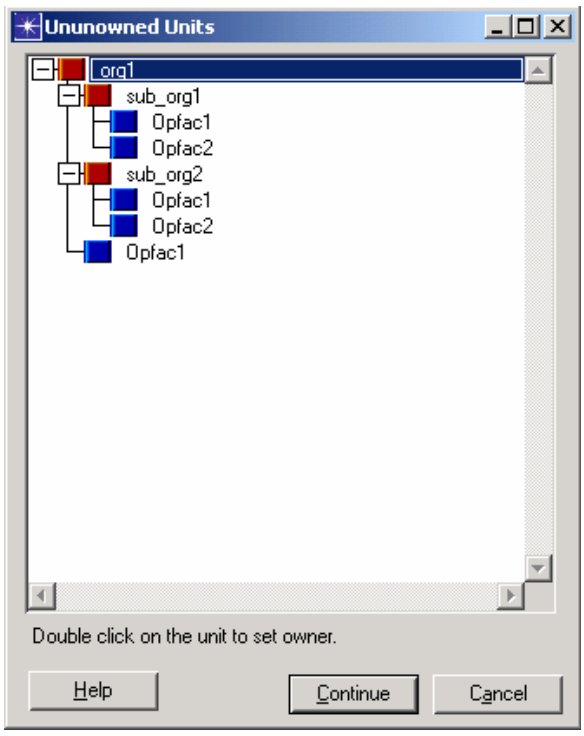


Figure 3-74 Unowned Units dialog box

Double-click on a unit to set owner information, which will display the Set Owner dialog box. Expand the treeview on an organization and set owner on individual sub-units. Note that you can continue the export process with un-owned units in the scenario, however, those un-owned units will become locked to the Subordinate Planners.

2) Specify a name for the subordinate query file, option to automatically send the subordinate query file via email, and the functional areas that will be available to the Subordinate Planner in the following dialog box.

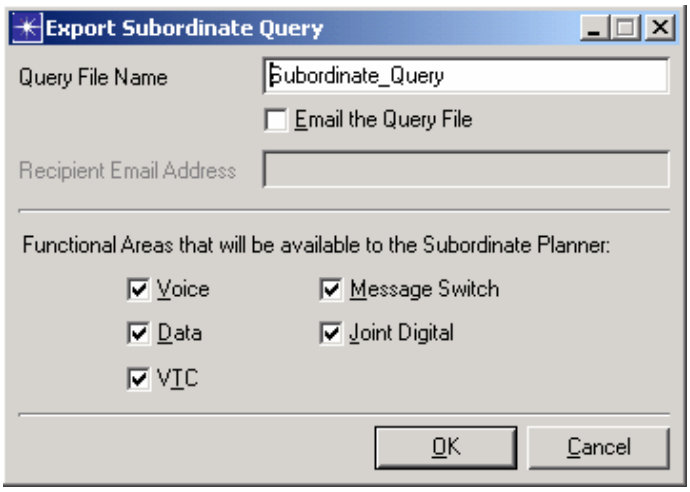


Figure 3-75 Export Subordinate Query File

The Lead Planner has the option in the dialog box to email the exported subordinate query to a recipient. If the email option is selected and the recipient's email address specified, not only is the file saved to the disk, but it is sent to the Lead Planner's default email application using MAPI. If the email application is configured for MAPI server support, an email message is created addressed to the recipient's email address with the subordinate query file attached to the email. Preview the message and click the **Send** button. If the email application is configured for MAPI to automatically send the email, the email is sent without the preview option. Much of the behavior of MAPI is dependent on the particular email application and its MAPI configuration. If the auto-email option is not selected, send it to the Subordinate Planners by whatever means available, such as FTP or use a location on a shared network drive.

3) Click **OK**; the subordinate query file is written to the disk and is located in `<NETWARS path>\User_Data\Projects\<current project>\<current phase>\Subordinate_Queries\`.

4) Save the project directly by clicking the **Yes** button.

Once the subordinate query file has been generated, all units with yellow marked icons will change to an orange folder icon indicating that there's an outstanding query.

Figure 3-76 Outstanding Query in a Scenario

Export Scenario > Subordinate Response

Scenarios > Export Scenario > Subordinate Response: Send the scenario back to the Lead Planner after the Subordinate Planner has elaborated the network and specified the subscriber requirements.

1) A check will be performed to see if there are any empty Requirements Matrix entries on any editable units. Any Requirements Matrix entries that have zero requested bandwidth and an empty answer field are considered to be empty. If there are empty entries, an alert dialog box as shown below will display.

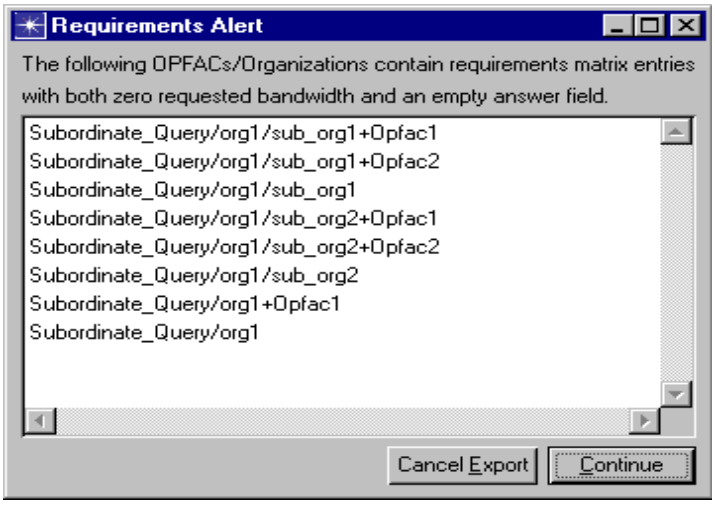


Figure 3-77 Alert dialog box when Exporting Response

This dialog box alerts the Subordinate Planner with a list of organizations and OPFACs that have at least one empty Requirements Matrix entry. Continue the export process or cancel the export.

2) If the Subordinate Planner clicks the **Continue** button, or if there were no empty fields, the Export Subordinate Response dialog box displays.

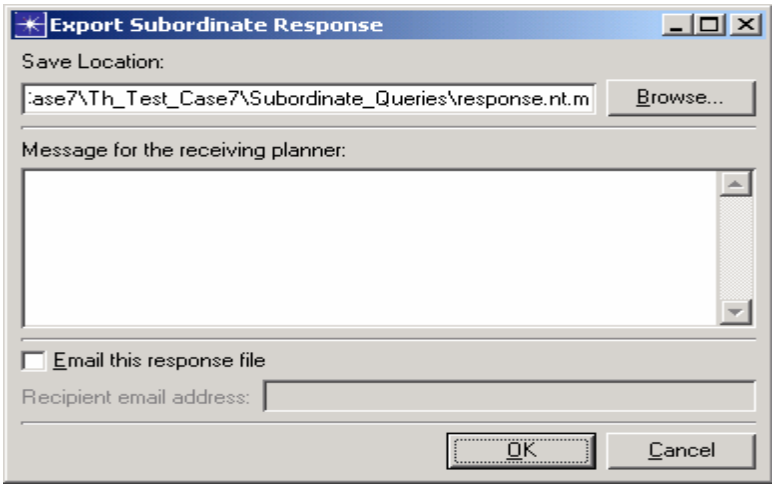


Figure 3-78 Export Subordinate Response dialog box

This dialog box serves three purposes: specify the save location for the response file, enter a message for the receiving planner, and email the response file.

3) To specify the save location, either type the full path in the **Save Location** field, or click the **Browse** button and choose the path from the resulting file chooser.

4) To enter a message for the receiving planner, type the message in the **Message for the receiving planner** field. The message will be shown to the receiving planner during the response import process.

5) To automatically email the response file that is generated, check the **Email this response file** checkbox and enter the destination email address in the **Recipient email address** field. If the recipient's email address is specified, MAPI is used to create a new email message in the Subordinate Planner's default email application. The subordinate response file is attached to this email. Depending on the MAPI configuration of the default email application, the email may or may not be previewed by the Subordinate Planner. Much of the behavior of MAPI is dependent on the particular email application and its MAPI configuration. If the auto-email option is not selected, send it to the Subordinate Planners by whatever means available, such as FTP or use a location on a shared network drive.

6) Click **OK**.

Export Scenario > To XML

Scenarios > Export Scenario > To XML: Export the contents of a scenario to XML. This allows for easy transfer of projects and scenarios between computers. The XML file is written to the folder that contains the scenario file. The exported XML file contains all the information in the scenario, except for some traffic information.

Export Scenario > To JNMS

Scenarios > Export Scenario > To JNMS: Export the scenario to JNMS.

Export Scenario > To Visio

Scenarios > Export Scenario > To Visio: Export the scenario to Visio XML. A <scenario name>.xml file and <scenario name>.vdx file are created in the scenario folder.

Reports

Scenarios > Reports > (*option*): Access the reports to view summary information in a table format. These options, as well as steps for working with these features, are discussed below.

Reports > CCSD Summary

Scenarios > Reports > CCSD Summary: View a CCSD Summary report.

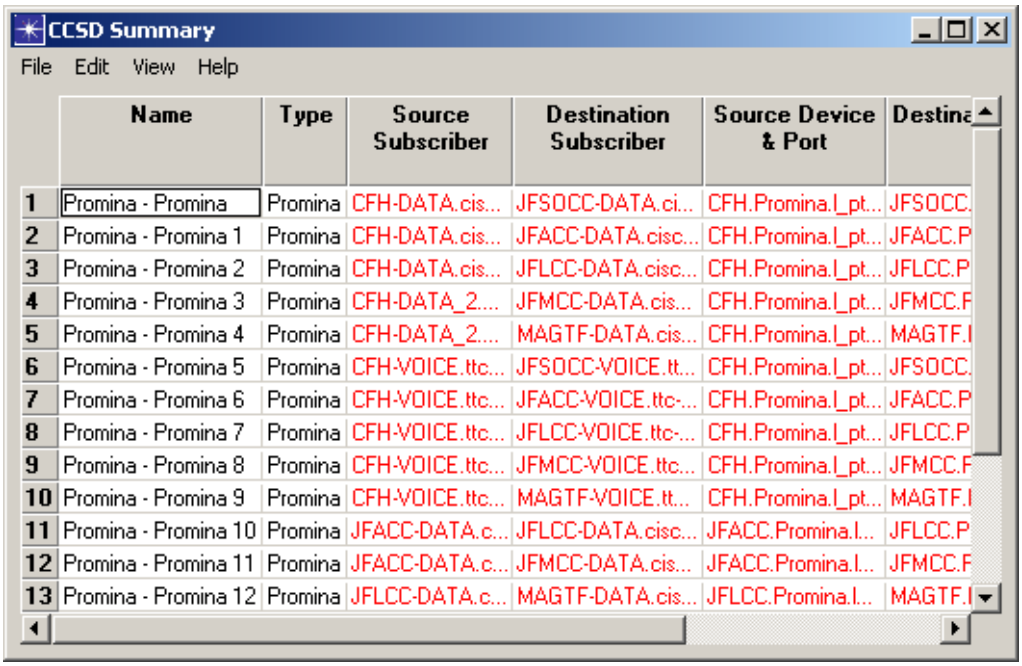


Figure 3-79 CCSD Summary Report

Reports > Free-Text Questions

Scenarios > Reports > Free-Text Questions: View a summary report of free-text questions attached to organizations, OPFACs, and devices. This provides a mechanism for the planners to communicate with each other using free text. This is especially useful when a Subordinate Planner wants to communicate with a Lead Planner since the mission analysis question interface cannot import back new questions added by the Subordinate Planner.

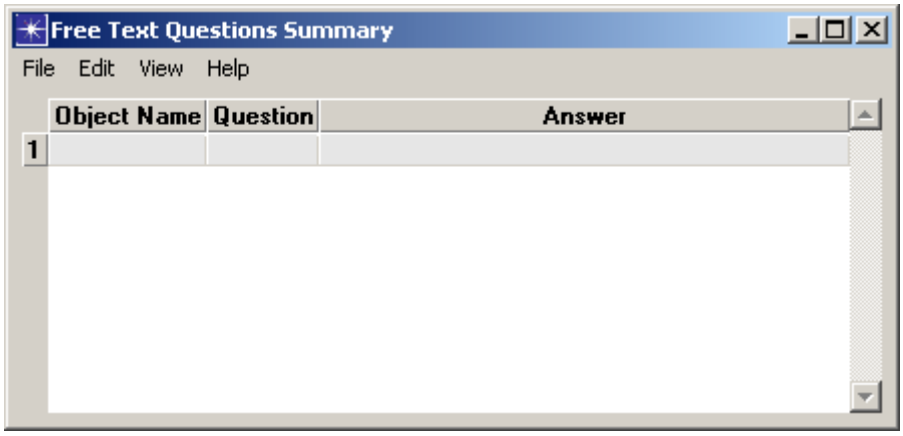


Figure 3-80 Free-Text Questions Report

These questions and answers are stored as attributes on the objects and can be accessed by right-clicking on the object and selecting **Edit Attributes** and clicking the **Value** column next to the **Free-Text Questions** attribute.

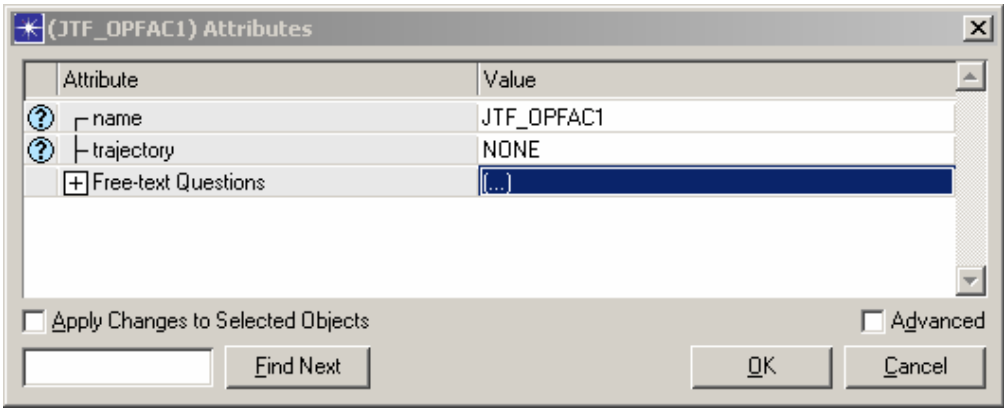


Figure 3-81 Free-text Questions Stored as Object Attributes

Reports > IP Addresses

Scenarios > Reports > IP Addresses: Examine the IP addresses for all connected interfaces on the devices. The IP Address Summary report contains the device name, interface name, IP address and subnet mask information. Buttons at the bottom of the table export the contents to an Excel file or to a HTML file, and print the contents to the default printer, respectively. You cannot edit any of the fields in this table.

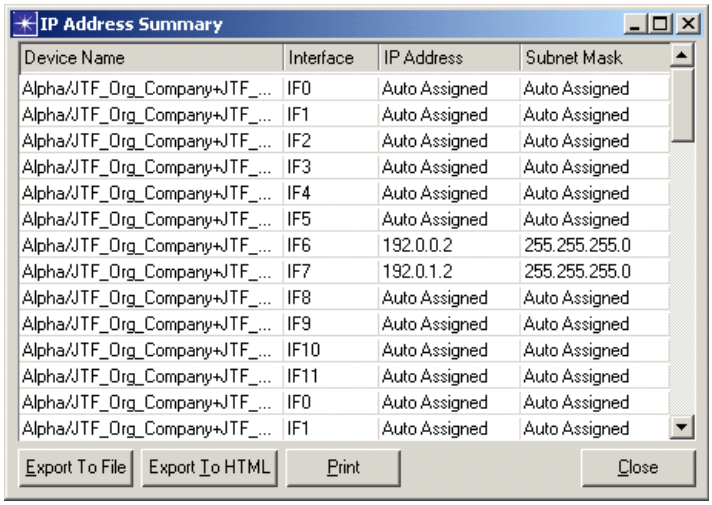


Figure 3-82 IP Address Summary Report

Reports > Links

Scenarios > Reports > Links: View a list of all links in the scenario along with some of the key attributes like name, bandwidth, type of link, etc. Buttons at the bottom of the table export the contents to an Excel file or to a HTML file, and print the contents to the default printer, respectively.

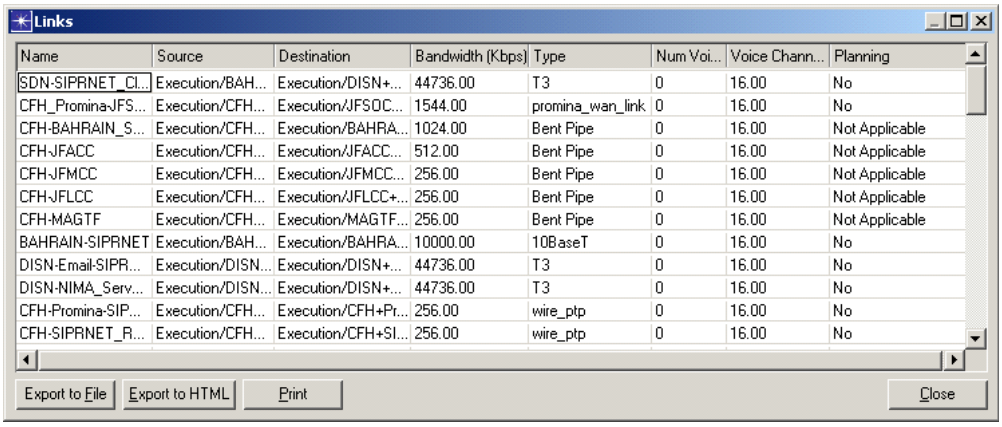


Figure 3-83 Links Summary Report

Reports > Mission Analysis Questions

Scenarios > Reports > Mission Analysis Questions: Every project has a default set of mission analysis questions that is automatically included in the project when the project is created. Any planner can answer these questions using the table interface shown below. Modifications made to the mission analysis questions and answers get exported to the Subordinate Planner; however, modifications made by the Subordinate Planner will not get imported back to the Upper-level Planner during the import response process.

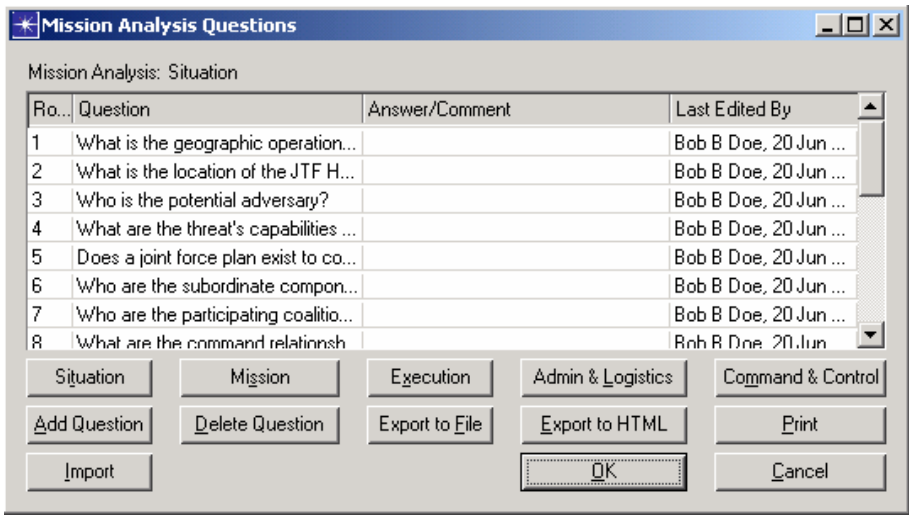


Figure 3-84 Mission Analysis Questions Report

The mission analysis questions are stored in five different sections: Situation, Mission, Execution, Admin & Logistics and Command & Control. When the Mission Analysis Questions table is launched, questions under the Situation section are displayed. The planner can view questions in the other sections by clicking on the appropriate button below the table. The label on top of the table represents the current section.

1) Click on either the **Question** or **Answer/Comment** cell to enter the desired text.

2) Click the **OK** button to populate the question/answer back into the table.

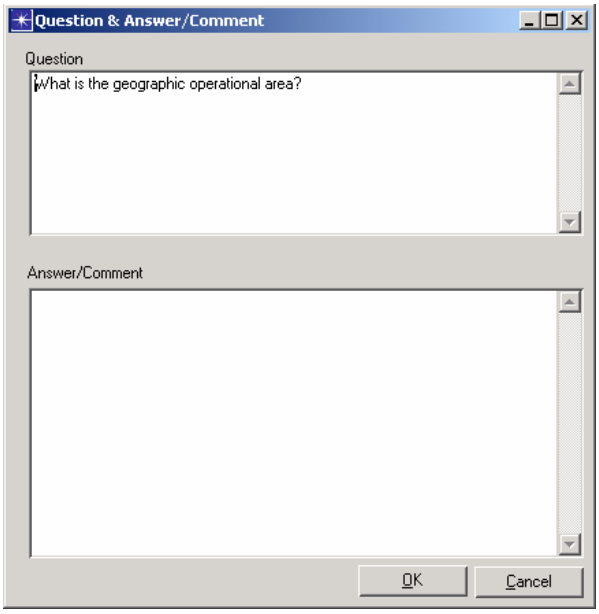


Figure 3-85 Answering a Mission Analysis Question

a) Add or delete a question from the Mission Analysis Questions table by selecting the **Add Question** or **Delete Question** buttons.

b) Print the content of the Mission Analysis Questions table to the default printer by clicking on the **Print** button. The printout contains all mission analysis questions and answers including the last edited by information divided by each section.

c) The content of the Mission Analysis Questions table can be exported to an Excel file or to an HTML file by clicking the **Export to File** or **Export to HTML** buttons, respectively. Specify the target location on disk to store the exported file.

d) In addition to manually typing in mission analysis questions for each project, you may also import additional mission analysis questions in to the Mission Analysis table. Import this information by selecting the **Import** button. A file chooser dialog box launches; select a desired text file and select the **Open** button.

Note—The text file must be tab-delimited and must follow the same format as the Mission Analysis defaults file, i.e., MissionAnalysis.gdf.

Once you have selected the file from which the additional Mission Analysis questions are to be loaded, the table will be updated to display this newly appended information. Duplicate mission analysis questions will not be appended to the dialog box.

Reports > Requirements

Scenarios > Reports > Requirements: View a summary report of the bandwidth requirements for all the organizations in the scenario. When this table is launched, there is one row for every top-level organization in the scenario. Each row has the name of the organization followed by the required bandwidth for each service type. The default functional area is Voice.

1) View the requirements summary for a different functional area by clicking on the appropriate button.

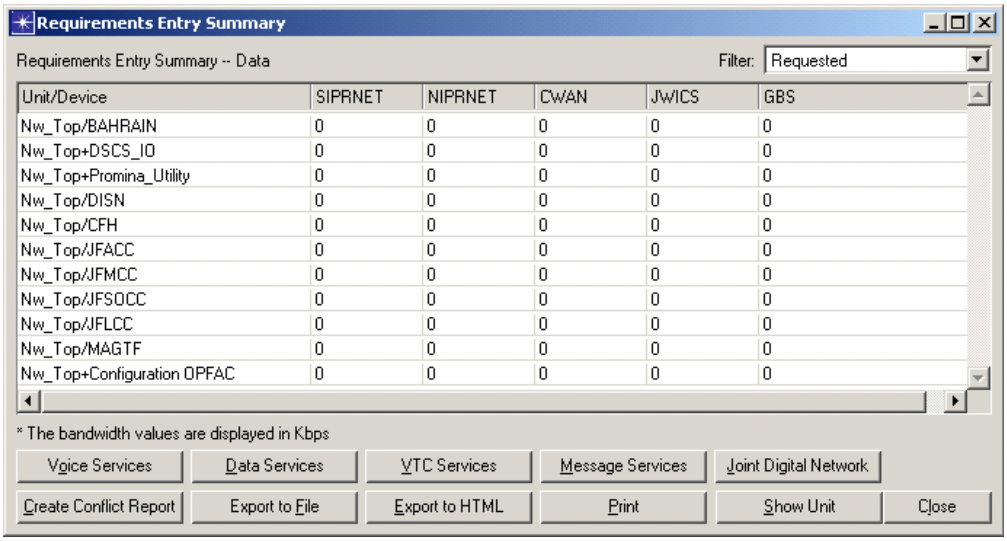


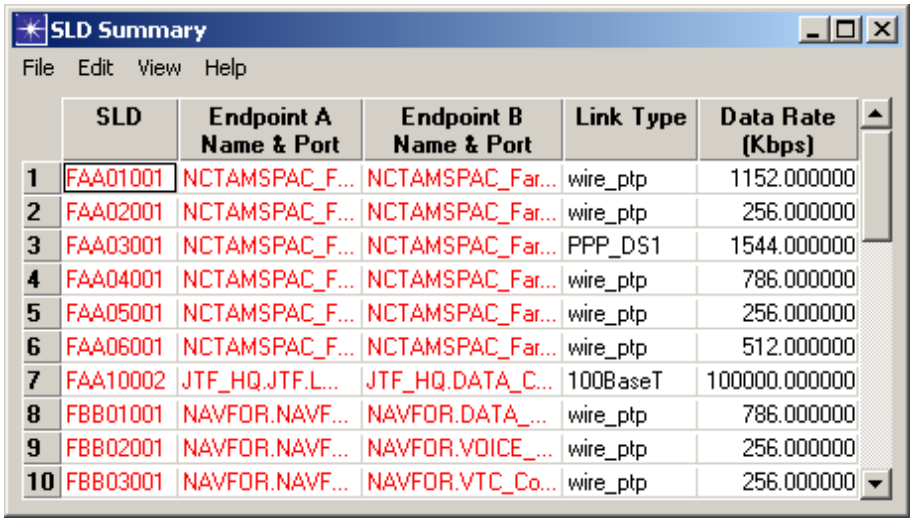
Figure 3-86 Requirements Summary Report

a) Click on the organization name to view its sub-organizations examine a breakdown of the bandwidth requirement for each sub-organization.

b) Export the contents of the table to an Excel spreadsheet, to HTML, or send it directly to a printer.

Reports > SLD Summary

Scenarios > Reports > SLD Summary: View a list of all SLDs in the scenario along with some of the key attributes like name, location, equipment, etc. Buttons at the bottom of the table export the contents to an Excel file or to a HTML file, and print the contents to the default printer, respectively.



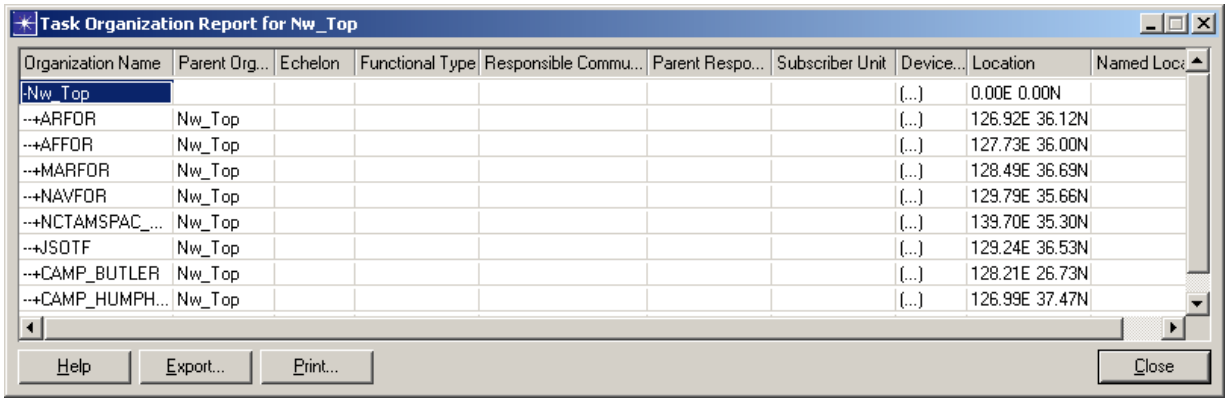
	SLD	Endpoint A Name & Port	Endpoint B Name & Port	Link Type	Data Rate (Kbps)
1	FAA01001	NCTAMSPAC_F...	NCTAMSPAC_Far...	wire_ptp	1152.000000
2	FAA02001	NCTAMSPAC_F...	NCTAMSPAC_Far...	wire_ptp	256.000000
3	FAA03001	NCTAMSPAC_F...	NCTAMSPAC_Far...	PPP_DS1	1544.000000
4	FAA04001	NCTAMSPAC_F...	NCTAMSPAC_Far...	wire_ptp	786.000000
5	FAA05001	NCTAMSPAC_F...	NCTAMSPAC_Far...	wire_ptp	256.000000
6	FAA06001	NCTAMSPAC_F...	NCTAMSPAC_Far...	wire_ptp	512.000000
7	FAA10002	JTF_HQ.JTF.L...	JTF_HQ.DATA_C...	100BaseT	100000.000000
8	FBB01001	NAVFOR.NAVF...	NAVFOR.DATA_...	wire_ptp	786.000000
9	FBB02001	NAVFOR.NAVF...	NAVFOR.VOICE_...	wire_ptp	256.000000
10	FBB03001	NAVFOR.NAVF...	NAVFOR.VTC_Co...	wire_ptp	256.000000

Figure 3-87 SLD Summary Report

Reports > Task Organization

Scenarios > Reports> Task Organization: View the Task Organization Report, which provides basic information about organizations within the mission plan, such as organization name, parent organization, and location. The report displays various JCSS and JNMS attributes of the organization; the values in the columns are not editable.

1) To view rolled-up device information for an organization, click its cell in the **Device Information** column.



Organization Name	Parent Org...	Echelon	Functional Type	Responsible Commu...	Parent Respo...	Subscriber Unit	Device...	Location	Named Loc
-Nw_Top							(...)	0.00E 0.00N	
--+ARFOR	Nw_Top						(...)	126.92E 36.12N	
--+AFFOR	Nw_Top						(...)	127.73E 36.00N	
--+MARFOR	Nw_Top						(...)	128.49E 36.69N	
--+NAVFOR	Nw_Top						(...)	129.79E 35.66N	
--+NCTAMSPAC_...	Nw_Top						(...)	139.70E 35.30N	
--+JSOTF	Nw_Top						(...)	129.24E 36.53N	
--+CAMP_BUTLER	Nw_Top						(...)	128.21E 26.73N	
--+CAMP_HUMPH...	Nw_Top						(...)	126.99E 37.47N	

Figure 3-88 Task Organization Report

The Device Information Report for the organization displays.

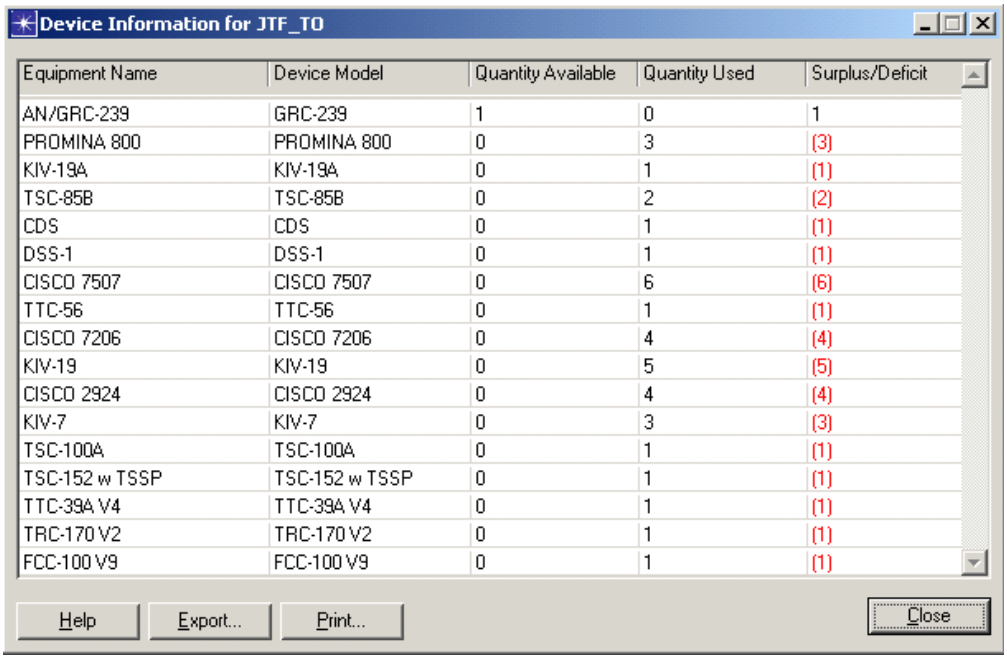


Figure 3-89 Device Information Report

Reports > User-Defined Reports

Scenarios > Reports> User-Defined Reports > *(option)*: Select from these options to open user-defined reports in live mode, and specify the content you want to include in your reports.

Reports > User-Defined Reports > Open Live Report Table...

Scenarios > Reports> User-Defined Reports > Open Live Report Table...: Open one or more user-defined report tables in live mode. This means the tables will not only show the corresponding values given the current selection set, but will also automatically repopulate themselves whenever the selection set changes. This provides a quick way of surveying the relevant attributes of a number of network objects without generating a full report and searching for the rows of interest. It can also sometimes be more handy than, for example, opening the Edit Attributes dialog box for each of the objects of interest.

Note—This functionality is supported independently of the **Generate Report from Template...** functionality, so it will neither create a new report nor discard any existing report information.

---

1) Select **Scenarios > Reports> User-Defined Reports > Open Live Report Table...** The Live User-Defined Report Tables dialog box displays.

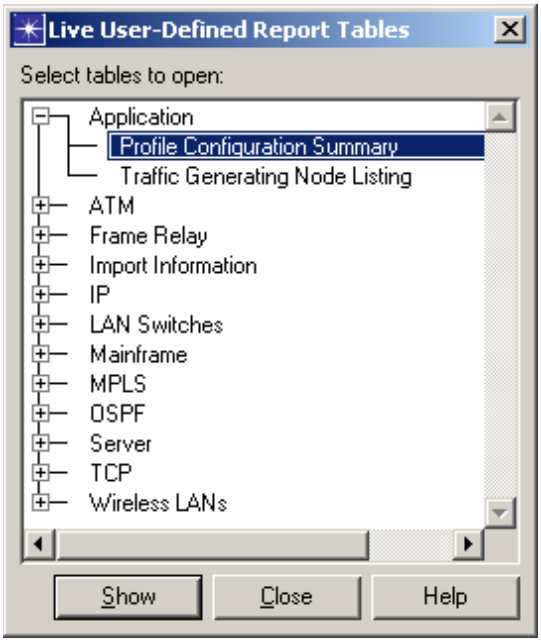


Figure 3-90 Live User-Defined Report Tables dialog box

2) Expand the top-level category subtree and left-click on the desired table entry to select it for opening. You can select multiple tables by holding down the **<Ctrl>** key while left-clicking or by clicking-and-dragging a rectangular selection. Selecting a top-level category tree row itself does not imply selecting all or any of its tables; it is simply ignored.

3) Click **Show** to build and open individual user-defined report table dialog boxes for each of the tables you have selected. Click **Close** to close this dialog box but not any of the user-defined report tables you might have opened.

Reports > User-Defined Reports > Generate Report from Template...

Scenarios > Reports> User-Defined Reports > Generate Report from Template...: Specify the content you want to include in your report.

1) Select **Scenarios > Reports> User-Defined Reports > Generate Report from Template...** The Generate User-Defined Report dialog box displays.

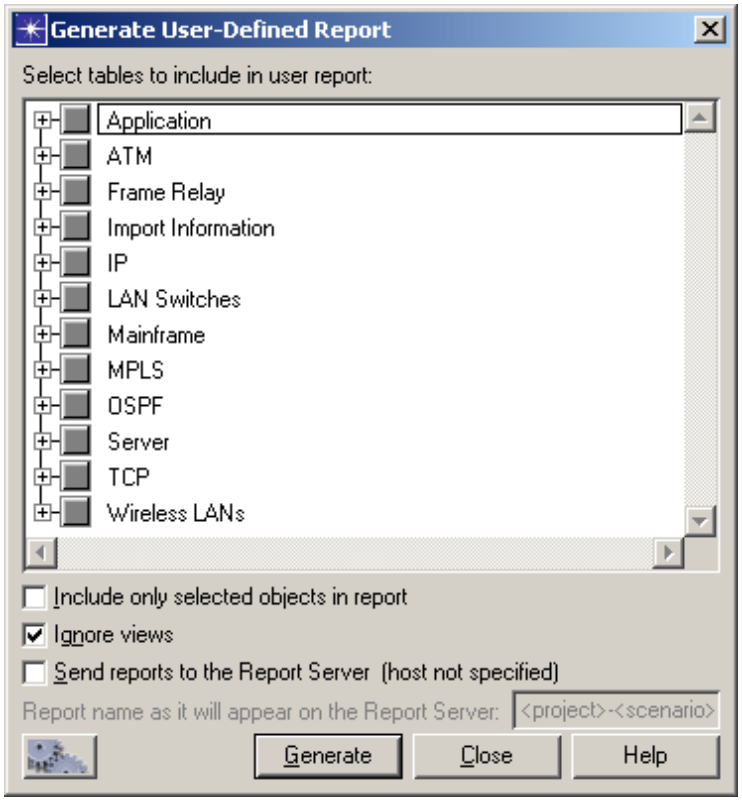


Figure 3-91 Generate User-Defined Report dialog box

2) Specify which tables of which categories should be included in the report by selecting them in the treeview. To select an entire category of tables, left-click on the top-level category entries. To select an individual table, expand the top-level category subtree and left-click on the table entry. Selected tables and categories show a green check mark. Categories with some but not all of its tables selected will display a green dot instead.

3) Click the **Include only selected objects in report** checkbox if you want table data to be collected only for objects that are currently selected in the network.

4) Click the **Ignore views** checkbox if you want table data to be collected even for objects that are not in the current view or views of the network.

5) Click the **Send reports to the Report Server (<host>)** checkbox if you want to send reports to the Report Server. This option is active only when a Report Server license is available.

   a) Specify a name for the report in the Report Server in the **Report name as it will appear on the Report Server** text field.

6) Click **Generate** to generate the table data for all of the selected tables. Your content settings are saved and will be reloaded the next time you generate a report. Click **Close** to close this dialog box but retain any table selections.

Generate Scenario Web Report

Scenarios > Generate Scenario Web Report: Choose the location where you want to save the scenario web report, and then generate the report.

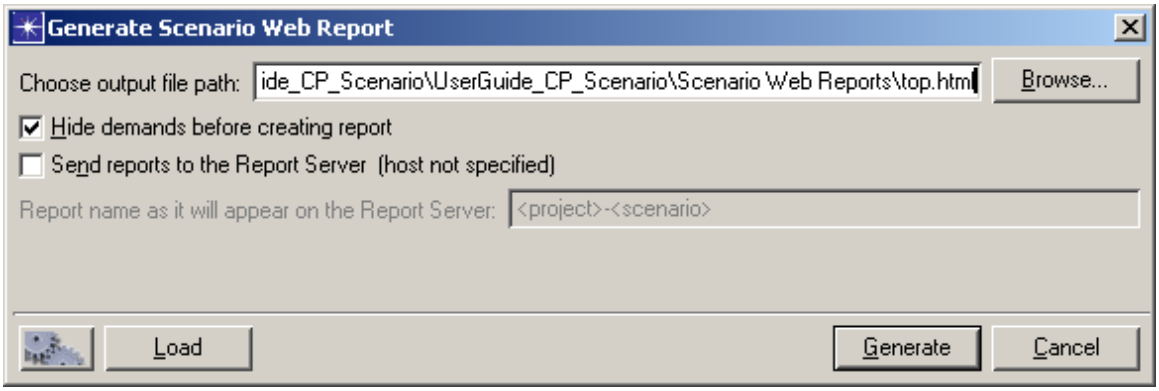


Figure 3-92 Generate Scenario Web Report dialog box

Click **Generate** to create the scenario web report. Navigate to the folder in which you saved the web report, and open it in your chosen web browser.

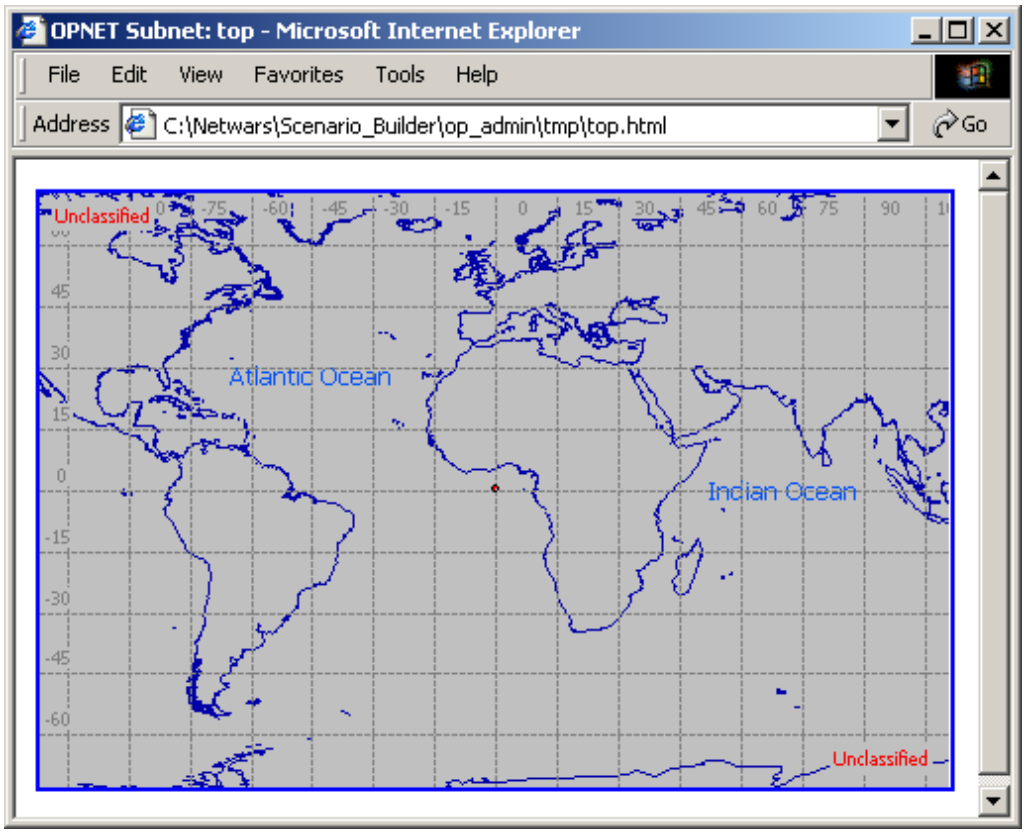


Figure 3-93 Displaying Scenario Web Report in Web Browser

Generate Scenario Bitmap

Scenarios > Generate Scenario Bitmap: Export a bitmap image of the model in the active window and set parameters of the export file.

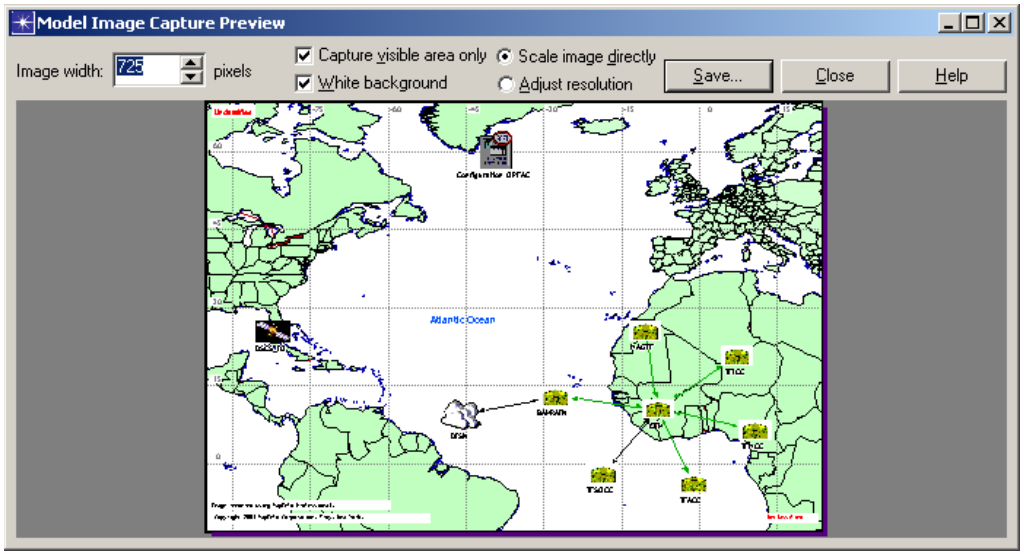


Figure 3-94 Model Image Capture Preview dialog box

Use the bitmap image controls to set the elements of the bitmap:

- **Image Width**: Lets you set the bitmap width using the up and down arrows. The height is adjusted accordingly.
- **Capture Visible Area Only**: Captures only the area visible in the editor window. When unselected, the entire subnetwork or model is captured.
- **White background**: Sets the bitmap background color to white, which improves contrast and visibility.
- **Scale Image Directly**: Sets the bitmap to the same level of detail as seen in the editor, regardless of the bitmap size.
- **Adjust Resolution**: Adjusts the level of detail based on the target bitmap size, that is, reduces the level of detail in smaller bitmaps and increases detail in larger ones.

Click **Save** to open the Select Captured Image Location dialog box, and select the directory to which the bitmap file is saved and name the file. Click **Close** to close the Model Image Capture Preview dialog box.

Topology Menu

Open Object Palette

Topology > Open Object Palette or click the **Open Object Palette** toolbar button: Access object palettes. JCSS provides a variety of object palettes which contain nodes, devices, links, and paths (circuits.) Once a new project is created, start building the network topology using the models provided in the object palettes. All of these objects are easily added to a scenario using the drag-and-drop technique.

Displaying Object Palettes

The object palettes display in either a Tree view style or an Icon view style, depending upon how your environment preferences are set.

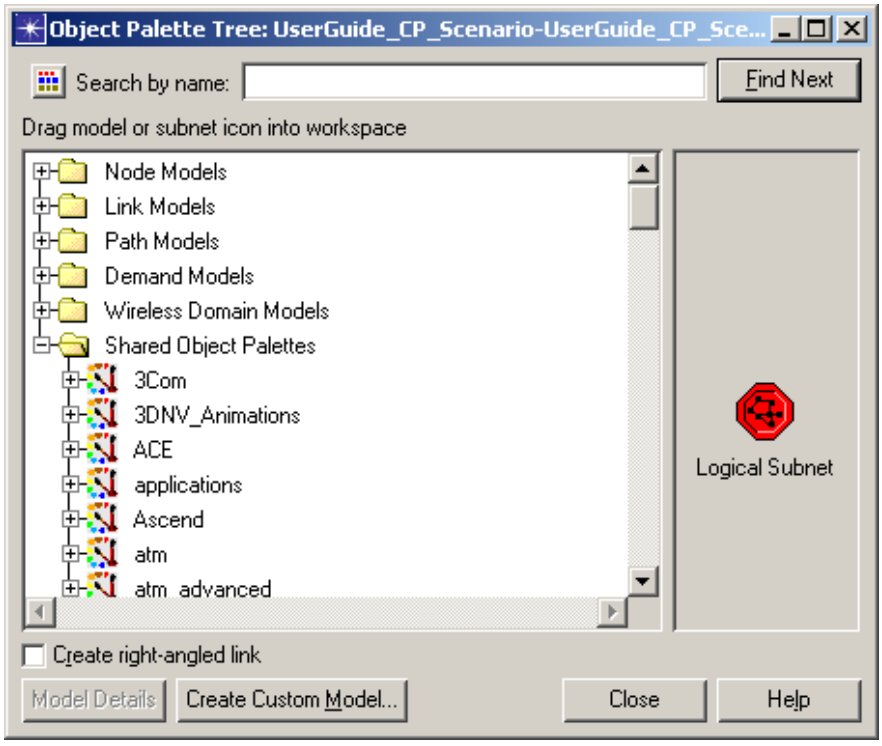


Figure 3-95 Object Palette by Tree View

In the Object Palette Tree view, click the **Open Palette In Icon View** icon button (upper left corner) to replace this dialog box with the Object Palette Icon view that displays only a single palette file at a time. The palette file initially displayed will be either the default palette of the scenario or, if the tree selection is within a particular palette file tree element, the palette that contains the current tree selection.

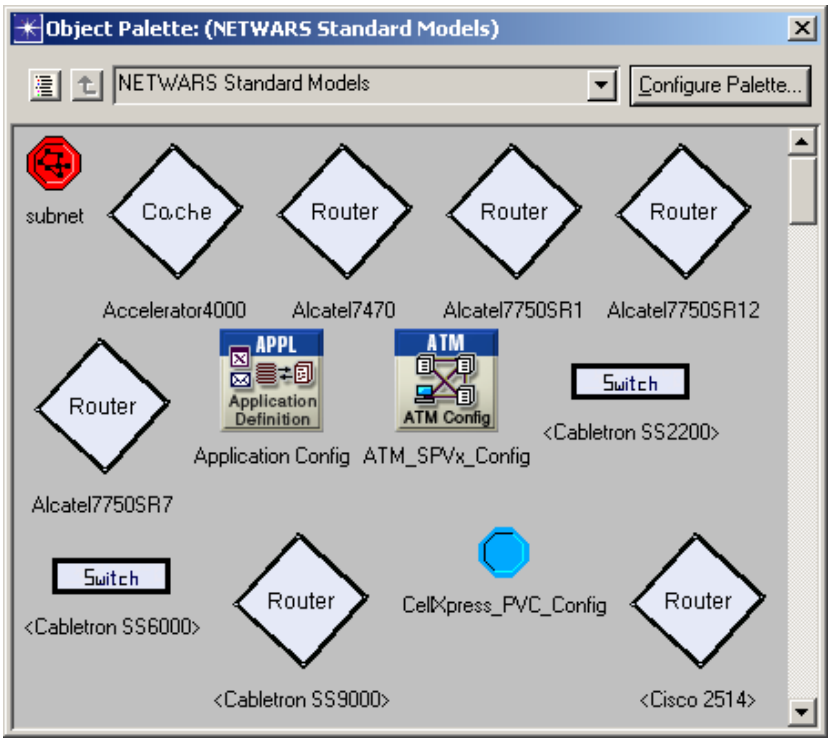


Figure 3-96 Object Palette by Icon View

Note—The view style in which the object palettes display changes based on the value specified for the **network_palette.style** environment preference. You may set the object palette view style preference via **Edit > Preferences > Advanced**.

Adding Objects to a Scenario

Adding an instance of a device model to a network (from either style object palette) is as simple as dragging an icon to the workspace.

1) Select and drag an icon from the palette to the workspace to create a new object. The newly created object in the network is an instance of the corresponding model.

   Whenever possible, planners are encouraged to use the NETWARS standard models as opposed to the more generic models contained in the other model palettes. The NETWARS models are designed to gather specific Measures of Performance (MOPs), which will be useful when modeling the network at a later point.

Note—JCSS device models must be used as **endpoints** for sending and receiving traffic. If generic device models are used as endpoints, JCSS will not be able to produce the traffic.

2) Once you drag an icon from the palette to the workspace, you may continue creating objects of the same type and model by clicking again in the workspace as needed.

3) Right-click in the workspace to stop creating new objects.

   Some object types, like paths and demands, use a right-click menu during the creation process. In those cases, you should pick the appropriate menu operation to complete or cancel the object creation.

4) Right-click a device icon in the workspace, and open a shortcut menu to set the name of the device or set different model attributes that influence the behavior of the model. Device attributes can be changed by clicking on a cell and typing or selecting a new value.

Using Dynamic Listing Option for Object Palettes

The software provides you with the option of displaying the object palettes in two modes: alphabetical or dynamic. In alphabetical mode, the palettes display in alphabetical order; in dynamic mode, the five most recently used object palettes display at the top followed by the alphabetical listing.

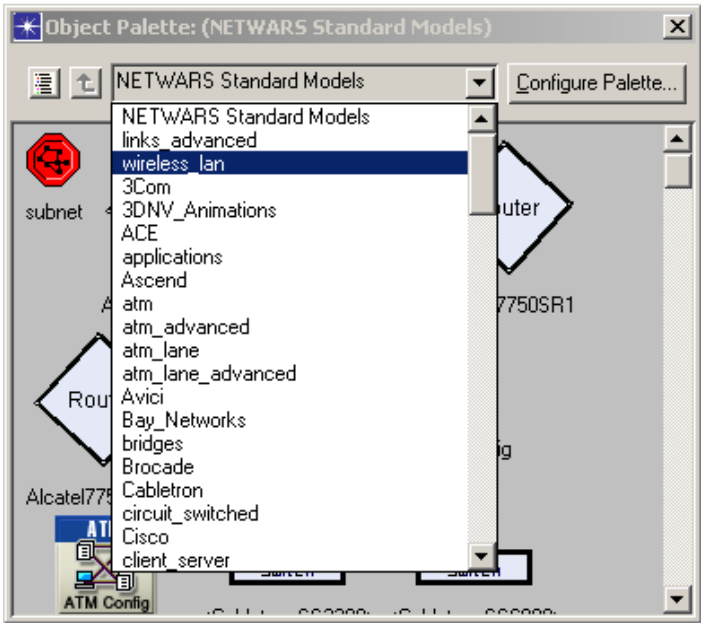


Figure 3-97 Dynamic Palette Listings Mode

The order in which the palettes display changes based on the value specified for the **palette_listing** environment preference. You may set this preference via **Edit > Preferences > Advanced**.

Configuring Custom Palettes

1) Click the **Configure Palette** button located at the upper-right corner of an Object Palette Icon view dialog box. The Configure Palette dialog box displays.

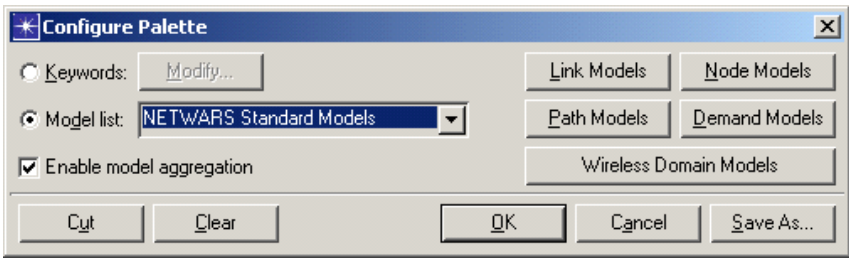


Figure 3-98 Configure Palette dialog box

2) Click the appropriate Models button to add or remove specific models or units from the palette.

3) Toggle the **Status** column to either included or not included to add or remove models units from the palette. To conclude, click **OK**, specify the name of the palette in the Save As dialog box, and then click **OK** to save the new configuration.

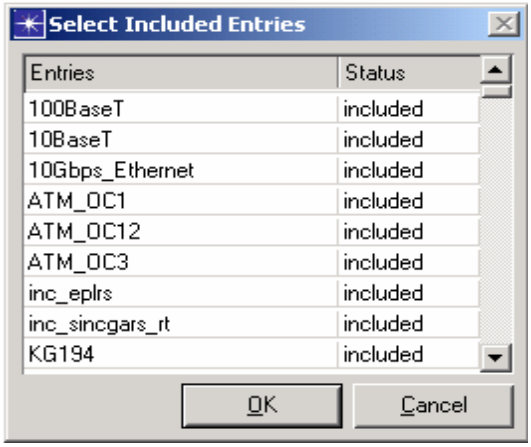


Figure 3-99 Select Included Entries dialog box

Open Library Treeview

Topology > Open Library Treeview: Add Organizations and OPFACs to scenarios by dragging and dropping them from the Library Treeview onto the Scenario Builder window.

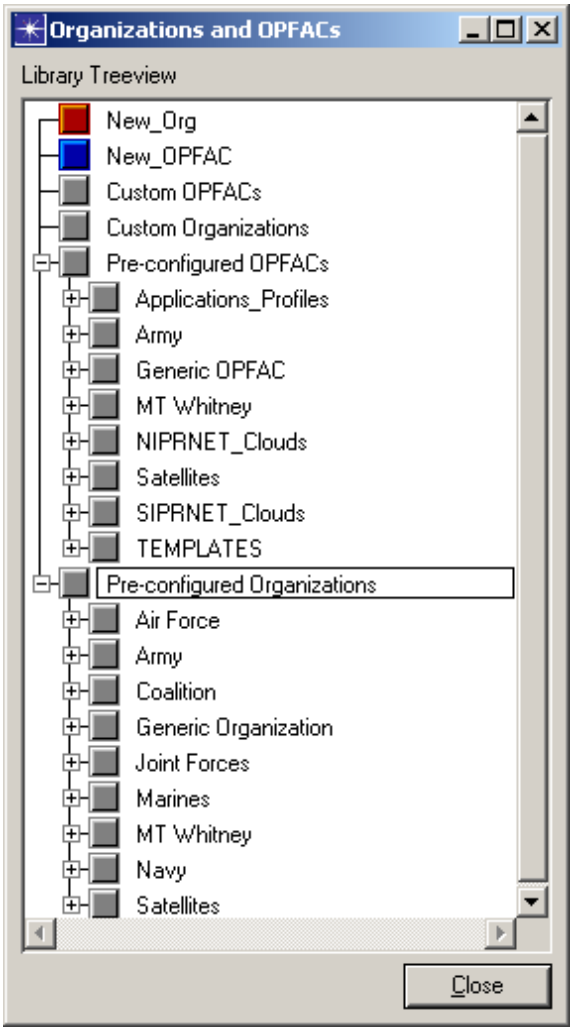


Figure 3-100 Library Treeview

Open Annotation Palette

Topology > Open Annotation Palette: Add graphic elements to a scenario to help organize it visually or call attention to key elements. The annotation palette includes five types of objects: Box, Ellipse, Line, Text, and Icon.

1) Click on the desired annotation object in the annotation palette.

Figure 3-101 Annotation palette

2) Create:

   a) A box or an ellipse by dragging an outline of the shape in the workspace.

b) A line by clicking in the workspace for the start of the line and each vertex (direction change) and then double-clicking to end.

c) Text by clicking on the workspace where the text is to be inserted, and then entering text in the text edit pad that displays.

d) An icon annotation by clicking in the workspace where you want the annotation to appear. To specify the icon to use, right-click on the annotation object and choose **Edit Attributes**; then set the "icon name" attribute. You can specify an icon from any icon database included in your model directories.

3) When you have finished creating an annotation object, you can:

a) Click to define another box, ellipse, line, or icon.

b) Right-click to stop creating boxes, ellipses, lines, or icons.

c) Click in the annotation palette to create a different type of annotation.

4) Modify the annotation object's attributes to customize its appearance. Select the object, and then right-click and select **Edit Attributes** to display the Attributes dialog box and make your changes.

Adding an annotation object does not affect a scenario's functionality in any way.

Import Topology

Topology > Import Topology > (*option*): Import various files relating to topology. The following import options are discussed in the sections below:

- From Device Configurations
- Convert DCI to NETWARS Scenario
- Link Specification
- From Configlets

Import Topology > From Device Configurations

Topology > Import Topology > From Device Configurations: Device Configuration Imports (DCI) lets you generate a network model by importing configuration data for Juniper routers, Cisco routers, and Cisco Catalyst switches. DCI uses the data in the configuration files to set the OPNET attributes that control routing and switching behavior.

OPFACs that are created as part of the import process will house a single device; the icon of that OPFAC will be same as the icon of the device inside.

Note—This section does not discuss in detail how the device configurations are converted to the network topology from the OPNET core software point of view. The discussion is limited to general workflow from the JCSS point of view. Consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**, for complete details.

---

1) The device configuration files can be imported in a new scenario or on top of an existing scenario. To perform a new import, create a new project. If you want to import these files on top of an existing network and then selectively add the various imported components to this network, open the relevant project.

2) Select **Topology > Import Topology > From Device Configurations**. The Import Device Configurations dialog box displays.

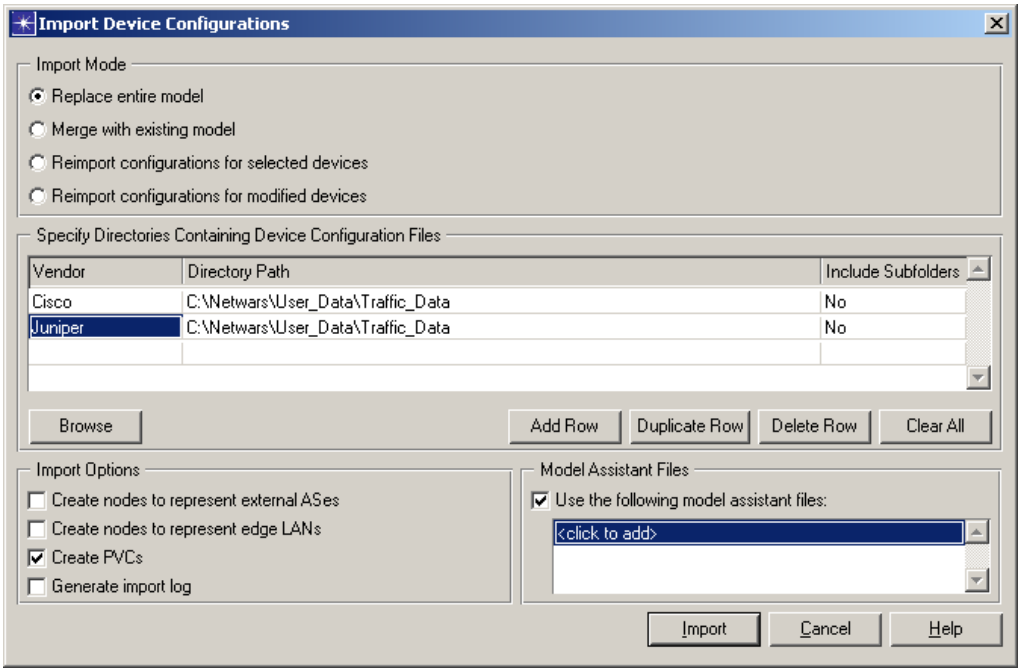


Figure 3-102 Import Device Configurations dialog box

3) Select how this import affects the existing topology (if one exists):

   a) **Replace entire model** radio button – Use this option when creating a new project or scenario, or when you want to overwrite the existing topology.

   b) **Merge with existing model** radio button – Use this option to add devices to the existing topology. If some of the devices are already in the topology, these devices are replaced (that is, re-imported) using the configuration files of this import. The Mapping Editor displays upon import so that users can map devices to the current network.

   c) **Reimport configurations for selected devices** radio button – Select this option to reimport the device configuration files of the nodes that are currently selected in the workspace. DCI imports the files from the same location specified in the original import.

   d) **Reimport configurations for modified devices** radio button – Select this option to reimport any device configuration files in the current topology that were modified since the last import. DCI imports the files from the same location specified in the original import.

4) Specify where the configuration files are located by vendor(s), directory path and whether to include subfolders in this directory path table. If you previously imported device configuration files, your previous selection is configured; you can keep the selection or change it. Multiple directories are supported.

5) Select the desired import options:

   a) **Create nodes to represent external ASes** – Creates dummy objects to represent missing external BGP (EBGP) neighbors.

   b) **Create nodes to represent edge LANs** – Creates LAN objects for active router interfaces that are not connected. By connecting these interfaces to LAN objects, no active router interfaces are left unconnected after the import.

   c) **Create PVCs** – Creates PVCs based on network data, if available; if this is not available, PVCs are created for all possible pairs of devices in the network.

   d) **Generate import log** – Creates a log of actions during the import. You can access the log through the object or workspace pop-up menus after the import.

6) Specify one or more model assistant files. These files can be model assistant files, which have a .ma extension, or import assistance files, which have an .ia extension. Consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**, for information about model assistant files.

7) Click **Import**. During the import process, various nodes can be created by the core DCI process for the end devices. The following table lists the corresponding NETWARS models that will be created:.

Table 3-2 DCI And NETWARS End Station Models

DCI End Station Models	NETWARS End Station Models
ppp_wkstn_adv	nw_ppp_wkstn_adv
10BaseT_LAN	nw_ethernet_wkstn_adv
100BaseT_LAN	nw_ethernet_wkstn_adv
1000BaseX_LAN	nw_ethernet_wkstn_adv
fr_wkstn_adv	No current support.
atm_wkstn_adv	No current support.
16TR_LAN	No current support.
FDDI_LAN	No current support.

8) If the **Merge with existing model** import option was selected, the Node Map Editor displays.

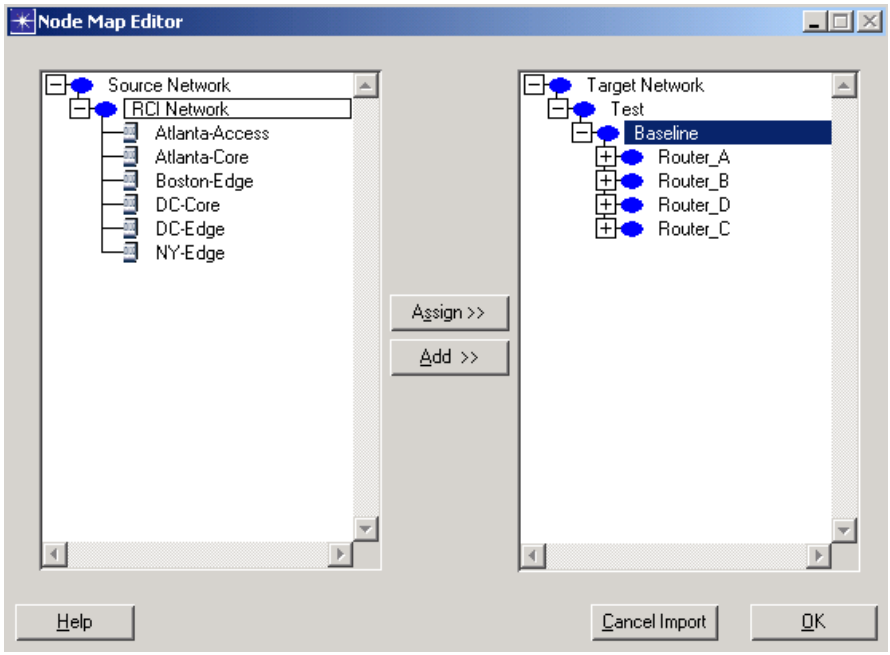


Figure 3-103 Node Map Editor dialog box

a) **Add** – Adds a node (router) from the imported network to the original network. Successful addition of the node requires that you select a node in the Source Network treeview and an OPFAC/organization in the target network before the addition can be performed.

b) **Assign** – Assigns or map a node (router) from the imported network to the original network. Successful assignment of the node requires that you select a node in the source network treeview and another node in the target network before the addition can be performed.

c) **OK** – Closes the mapping editor dialog box and performs any link creation as necessary.

d) **Cancel Import** – Cancels any changes made to the network due to mapping or assignment and brings you back to the original scenario.

e) **Auto-Mapping Feature** – Performs the mapping if router configuration files have been previously imported into this scenario. The Perform Auto-Mapping dialog box displays to confirm the mapping of the nodes based on the mapping performed in the previous imports.

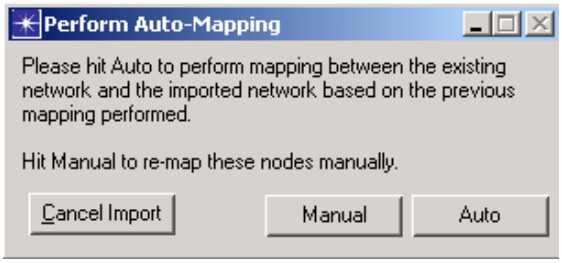


Figure 3-104 Perform Auto-Mapping

f) **Manual** or **Auto** – Continues with the auto mapping or manual mapping. The main dialog box for the node mapping will still display allowing you to map any nodes that were not mapped as part of the auto-mapping operation.

g) **Cancel Import** – Cancels the import procedure.

Import Topology > Link Specification

Topology > Import Topology > Link Specification: Import link information from text files. The link specification is separated into two sections—the JCSS attributes and the standard OPNET attributes. If a link with the specified hierarchical name already exists in the scenario, the changes are applied to the existing link. If there are no links with the specified hierarchical name, a new link is created with the specifications provided in the text file.

Import Topology > From Configlet

Topology > Import Topology > From Configlet: Import topology from configlets (configuration files.) Only those devices which have configuration files associated with them can be re-imported via this feature.

Export

Topology > Export > (*option*): Export various file types relating to topology. The following export options are discussed in the sections below:

- Attributes for Selected Objects
- Attributes for All Objects
- Selected Area to Bitmap
- Visible Area to Bitmap

Export > Attributes for Selected Objects

Topology > Export > Attributes for Selected Objects: Export the selected object attributes to text files.

1) Select the objects of interest (e.g., nodes and links).

2) Choose the **Topology > Export > Attributes for Selected Objects** option. The Export Attributes for Selected Objects dialog box displays.

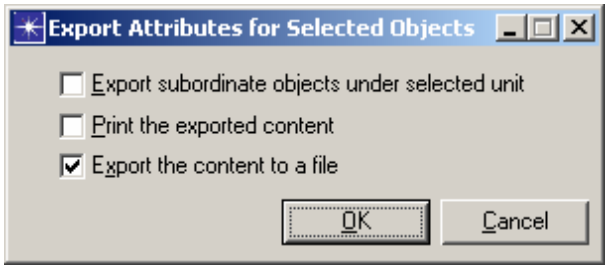


Figure 3-105 Export Attributes for Selected Objects dialog box

3) After making the appropriate selection select **OK**. The Save As dialog box displays prompting you to select the desired location and specify the file name to which to save the attribute text file on the hard drive.

Export > Attributes for All Objects

Topology > Export > Attributes for All Objects: Export all object attributes to text files.

1) The Export Attributes for All Objects dialog box displays.

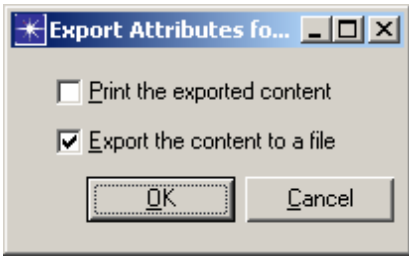


Figure 3-106 Export Attributes for All Objects dialog box

2) After making the appropriate selection select **OK**. The Save As dialog box displays prompting you to select the desired location and specify the file name to which to save the attribute text file on the hard drive.

Export > Selected Area to Bitmap

Topology > Export > Selected Area to Bitmap: Specify an arbitrary area on the workspace and export the contents to an image file.

1) Drag a rectangle around the area of interest for the screen capture.

2) In the Model Image Capture Preview window, set desired image options and click **Save**.

3) In the Select Captured Image Location dialog box, choose a file format, file name, and location, and then click **Save**.

Export > Visible Area to Bitmap

Topology > Export > Visible Area to Bitmap: Capture the visible area of a subnet as a screen shot and save it as a GIF or TIF file.

1) In the Model Image Capture Preview window, set desired image options and click **Save**.

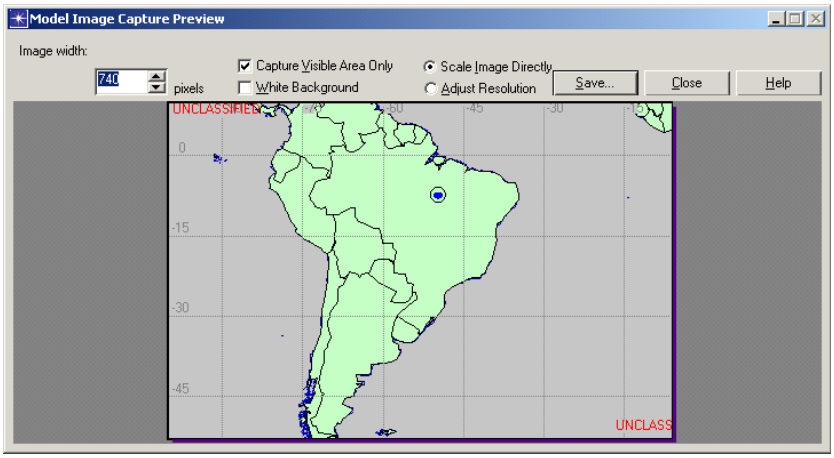


Figure 3-107 Model Image Capture Preview window

2) In the Select Captured Image Location dialog box, choose a file format, file name, and location, and then click **Save**.

3) Click **OK**.

Model Assistant

Topology > Model Assistant > (*option*): Edit model assistant files, apply the data to your network, and save the data to reuse later. A model assistant file is an ASCII text file that specifies a set of changes to a scenario. You can specify multiple changes in one file, then apply the file to make multiple changes in one operation. For example, you can create a model assistant file to:

- Create subnetworks, nodes, and links,
- Set object attributes,
- Import node aliases, and
- Specify object locations.

Model Assistant > Edit File

Topology > Model Assistant > Edit File: Edit model assistant files.

1) The Edit Model Assistant File dialog box displays. This dialog box contains multiple tabbed pages; each page has fields for entering the data you want to apply. If you want to edit an existing file, click **Load**, choose the file to load, and click **OK**. The dialog box updates each tabbed page to reflect the data in the loaded file.

2) Specify the information you want to apply on each page (click the tabs to view specific pages).

   a) If you do not want to apply a certain type of data, leave the page empty.

b) If you want to reset all settings on all pages and reenter your data, click **Clear**.

c) If you want to save your data to a model assistant file so you can reuse it later, click **Save**.

3) Click **Apply** to apply your data to the network, or **Close** to close the dialog box without applying your data.

Model Assistant > Apply File

Topology > Model Assistant > Apply File: Apply the selected model assistant file.

Model Assistant > Open Error Log

Topology > Model Assistant > Open Error Log: View errors that occur when the model assistant file is applied to your scenario.

Model Assistant > Save Current Topology to File

Topology > Model Assistant > Save Current Topology to File: Create a model assistant file based on your current topology.

Create Custom Device Model

Topology > Create Custom Device Model...: Create your own new device models, in addition to using the built-in model objects provided by JCSS and creating derived models. Create several different types of network components, including routers, bridges, hubs, workstations, switches, LANs, and vendor devices. For complete instructions, consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**.

Link Operations

Topology > Link Operations > (*option*): Use the sub-menu options to perform link consistency checks and define valid links between devices using a wizard-like feature.

Link Operations > Deploy Link

Topology > Link Operations > Deploy Link: Quickly define valid links between devices using a wizard-like feature.

1) Select two devices (through GUI or Network Browser) and then select **Topology > Link Operations > Deploy Link** (or press **<Ctrl>** + **L**.)

2) If more than one device is present in an OPFAC, the Select Devices dialog box displays. Select the desired devices to connect and then click **Next**.

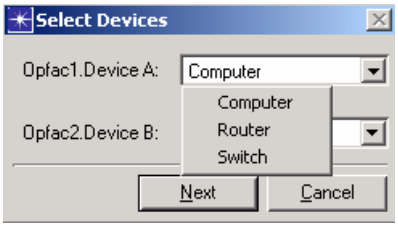


Figure 3-108 Sample Select Devices dialog box

If a selected OPFAC has only one device, then the device will be selected by default and the Select Devices dialog box will be skipped. If the OPFAC doesn't have a device, an error will be flagged.

This feature will be only applicable to the fixed nodes (only wired ports will be considered for selection.)

3) The best default link available and a list of the other applicable link types displays. A series of the most common default bandwidths for the selected link type is also provided as well as the first available ports applicable to the chosen link as default ports. Use the default values or select the desired values, and then click **OK**.

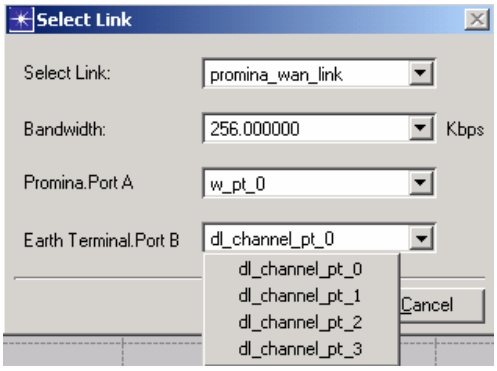


Figure 3-109 Sample Select Link dialog box

An inter-OPAFC link or intra-OPFAC link will be created with the user-defined attributes between the two devices, depending on whether the selected devices are within one OPFAC or different OPFACs.

Deploying Satellite Links

Defining satellite links can be performed using a procedure similar to defining other links.

1) Select two OPFACs or devices and then select **Topology > Link Operations > Deploy Link** (or press **<Ctrl>** + **L**.)

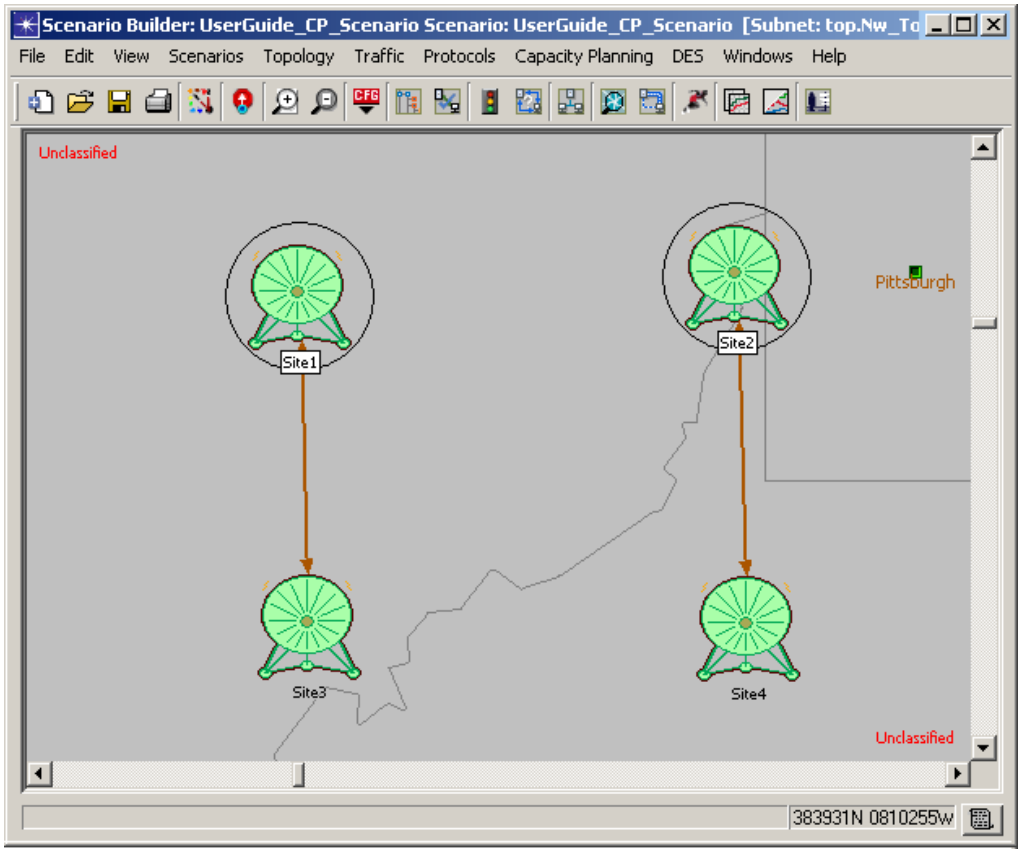


Figure 3-110 Selecting Satellites for Link Deployment

2) If more than one device is present in an OPFAC, the Select Devices dialog box displays. Select the desired devices to connect and then click **Next**.

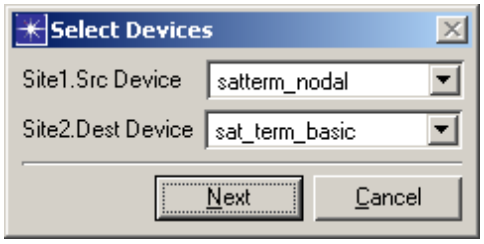


Figure 3-111 Sample Select Devices dialog box

If a selected OPFAC has only one device, then the device will be selected by default and the Select Devices dialog box will be skipped. If the OPFAC doesn't have a device, an error will be flagged.

3) The Satellite Bent Pipe Link Deployment dialog box displays.

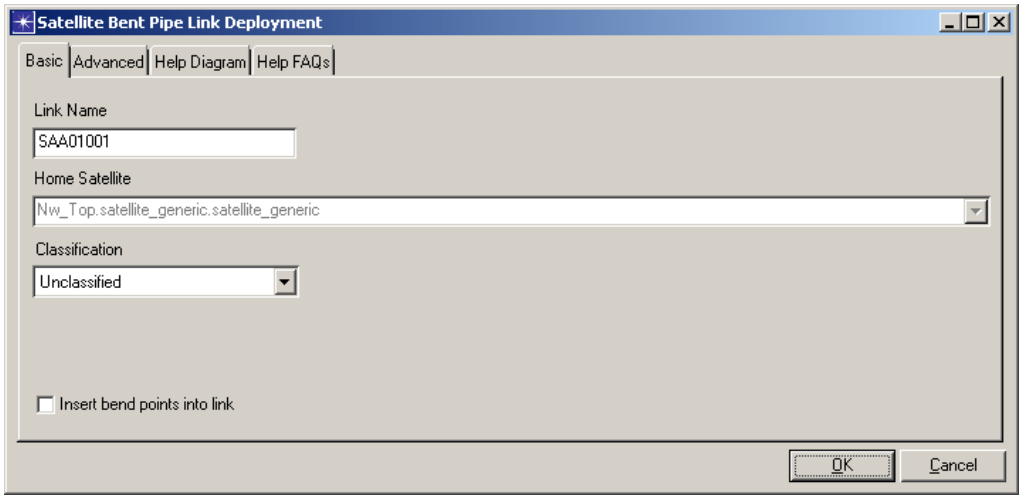


Figure 3-112 Satellite Bent Pipe Link Deployment dialog box ~ Basic

The Basic tab displays the default Link Name, and allows you to select the Home Satellite which provides the available frequencies.

a) Click the Advanced Tab. The following screen displays.

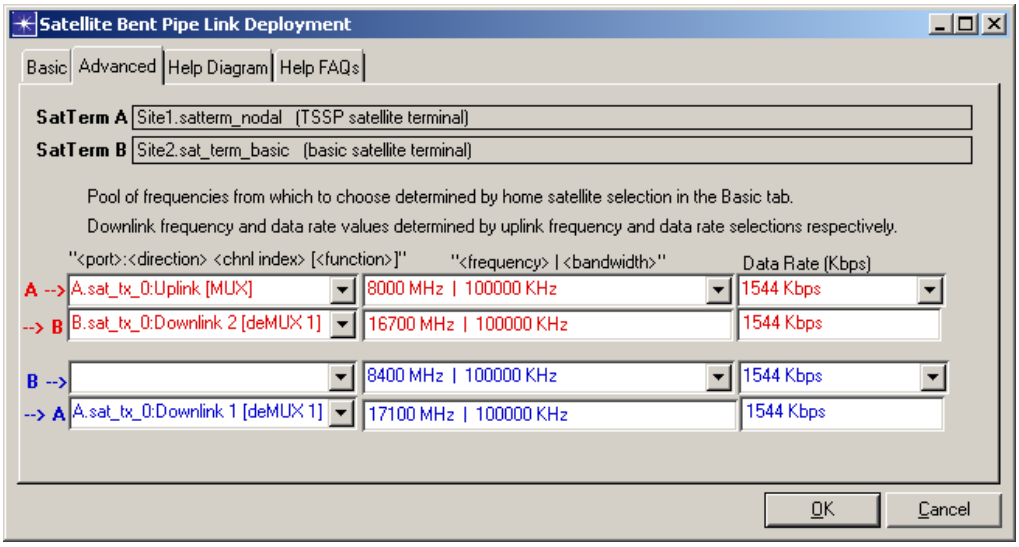


Figure 3-113 Satellite Bent Pipe Link Deployment dialog box ~ Advanced

Based on the frequencies provided by the chosen Home Satellite, this screen allows you to configure the frequencies uplink and downlink (in each direction) to establish a satellite link between the two selected satellite terminals.

The fields of this screen are pre-populated for you with the best available default link.

b) Click the Help Diagram tab. The following screen displays a diagram which explains the function of this dialog box .

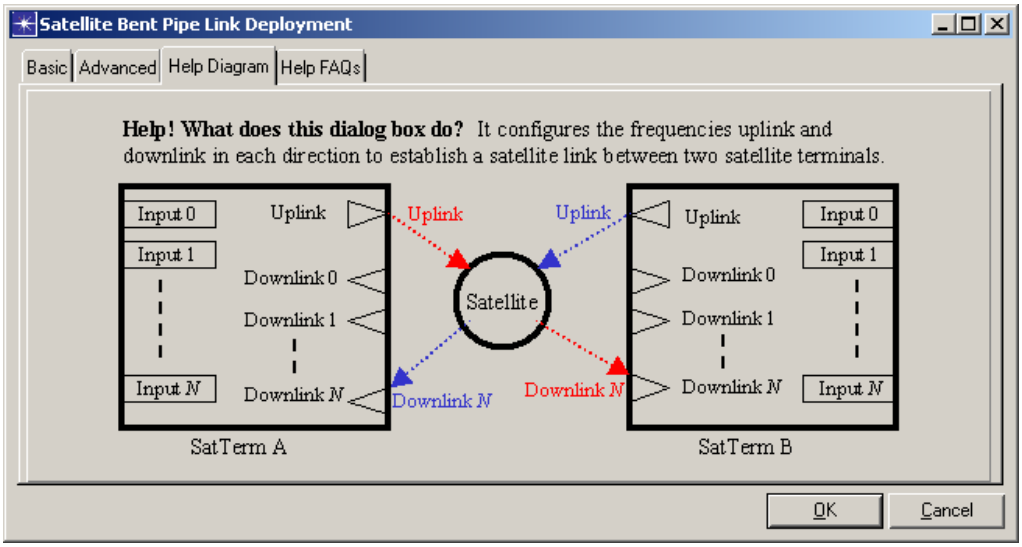


Figure 3-114 Satellite Bent Pipe Link Deployment dialog box ~ Help Diagram

c) Click the Help FAQs tab. The following screen displays a scrollable list of Frequently Asked Questions and answers regarding satellite link deployment.

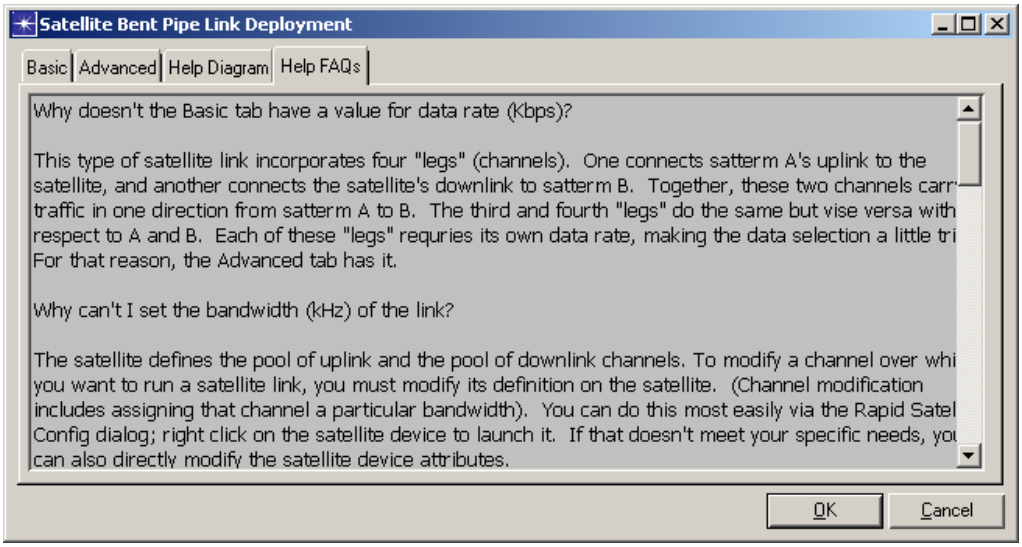


Figure 3-115 Satellite Bent Pipe Link Deployment dialog box ~ Help FAQs

4) Use the default link values provided in the Satellite Bent Pipe Link Deployment dialog box (or select the desired values), and then click **OK**. The newly deployed satellite link displays in the workspace.

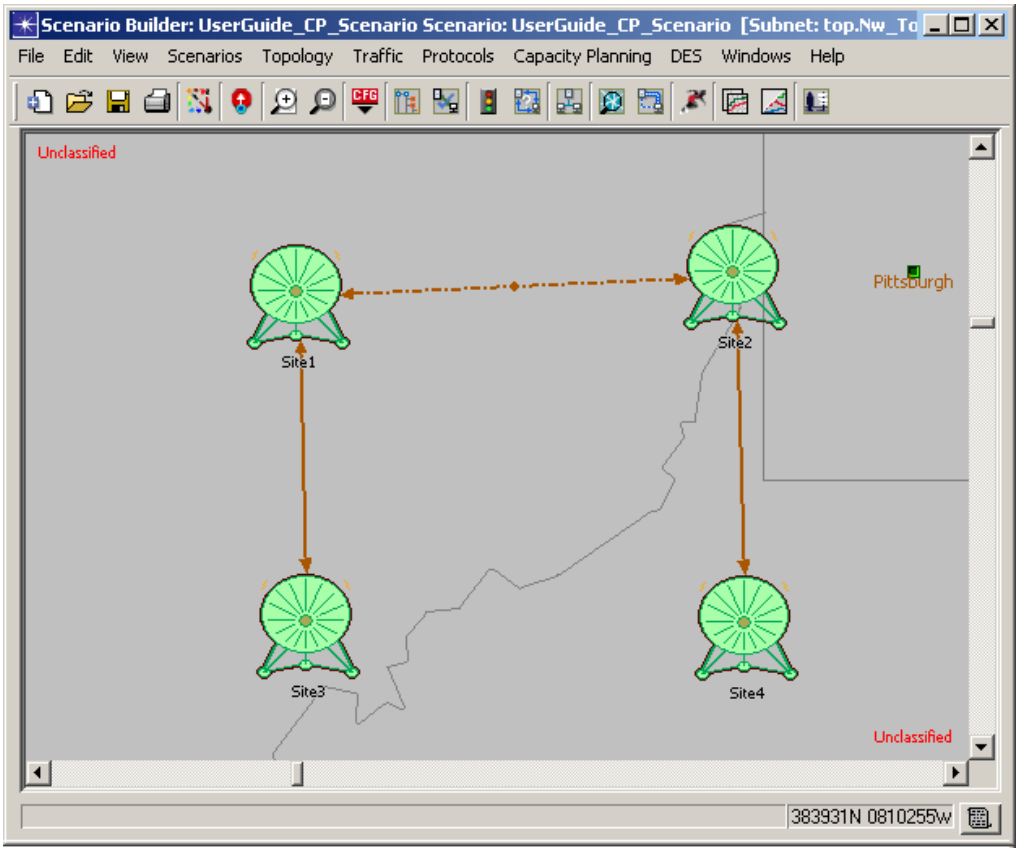


Figure 3-116 Deploying Satellite Links

Link Operations > Verify Links

Topology > Link Operations > Verify Links: Perform inter and intra OPFAC link consistency checks. Inconsistent links (e.g., links not connected to the right ports on the end points of the link) will be marked with a red X in the workspace. This option can also be launched using the **Verify Link Consistency** toolbar button.

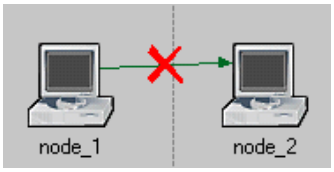


Figure 3-117 Inconsistent Links

In addition, the results of the link consistency check will be displayed in the Scenario Link Consistency dialog box. Troubleshooting solutions are provided, where possible, in the text of the failure reason.

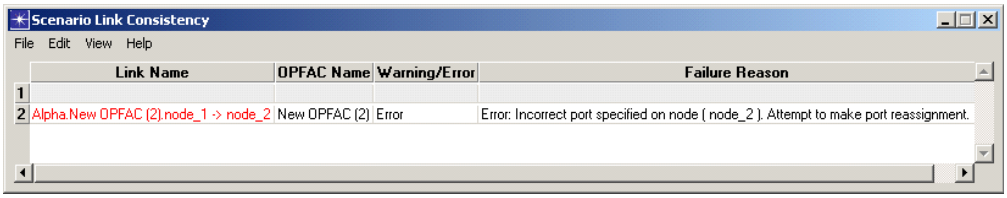


Figure 3-118 Scenario Link Consistency dialog box

If all links are properly connected, a message will be displayed in the bottom left corner of the scenario builder editor after the consistency check is performed.

Link Operations > Clear Links

Topology > Link Operations > Clear Links: Remove red Xs marking inconsistent links in the workspace.

Link Operations > Import Links from File

Topology > Link Operations > Import Links from File: Imports links from designated link import file.

Save As Link Type

To save inter-OPFAC links to a new link, right-click on a link in the workspace and select **Save As Link Type**. The status bar at the bottom left corner of the Scenario Builder window displays the following message, “New link type ‘<selected link name>’ saved.”

Deploy Circuit

Topology > Deploy Circuit: Follow the steps in the Circuit Wizard to configure the desired circuit. All circuit devices (i.e., SCREAM, SHOUTip, CTP, and Promina) are deployed using the same procedure. A sample generic Circuit Wizard workflow – using Prominas as an example – is provided below.

1) Select **Topology > Deploy Circuit** (or press **Ctrl + D**) to start the wizard. You must select at least two compatible circuit devices or OPFACs that contain circuit devices; otherwise, an error will be prompted and the wizard is terminated.

   The Circuit Deployment Wizard displays.

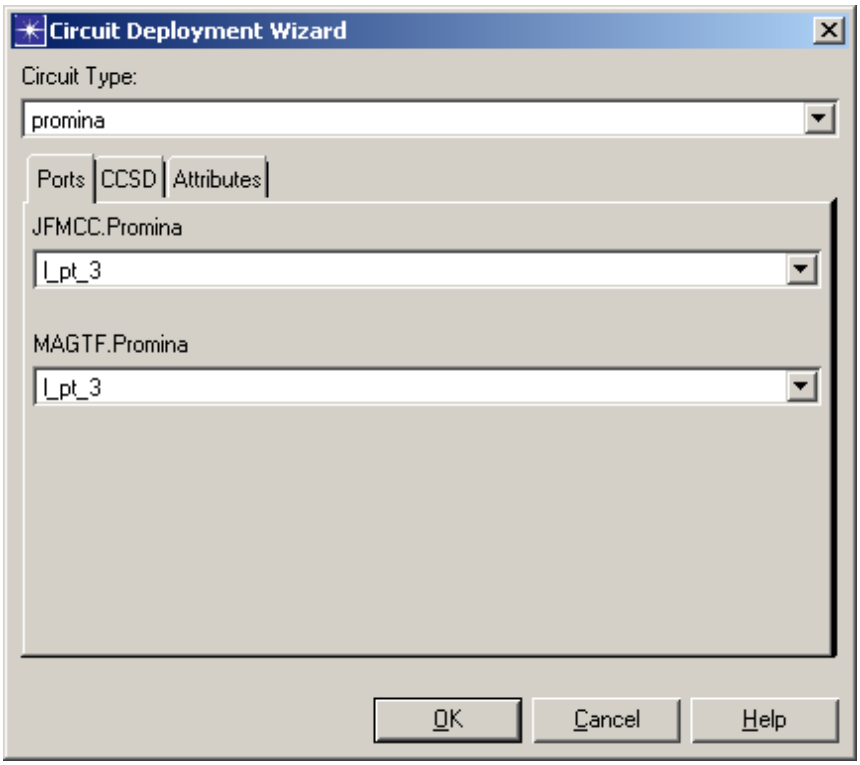


Figure 3-119 Circuit Deployment Wizard Ports Tab

2) On the Ports tab, specify the ports for the selected source and destination Promina devices in the scenario.

3) On the CCSD tab, configure the CCSD codes for the Promina circuit.

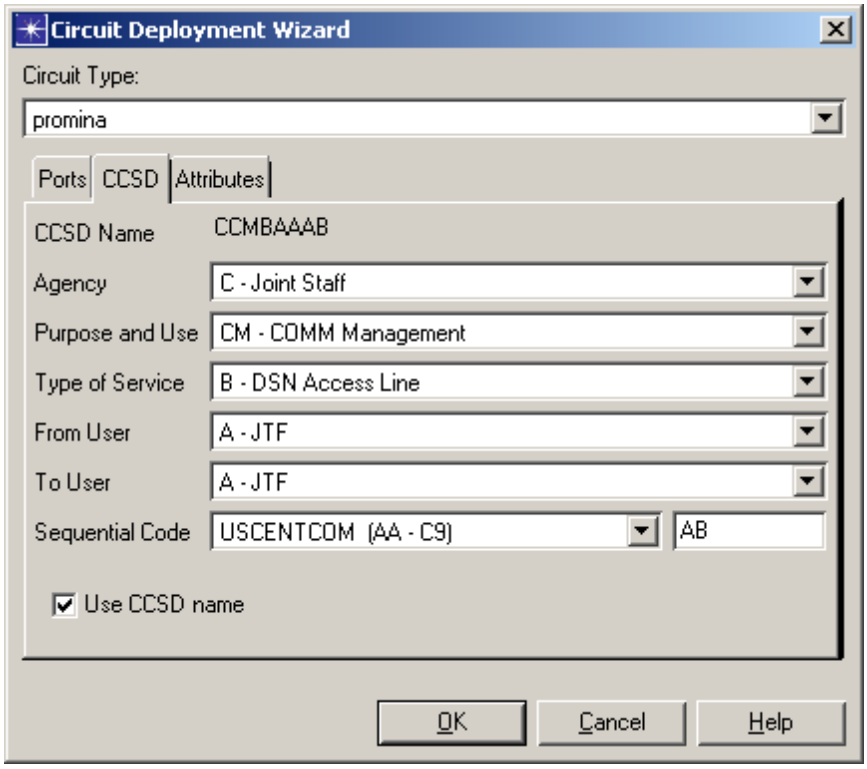


Figure 3-120 Circuit Deployment Wizard CCSD Tab

4) On the Attributes tab, configure the circuit's attribute values.

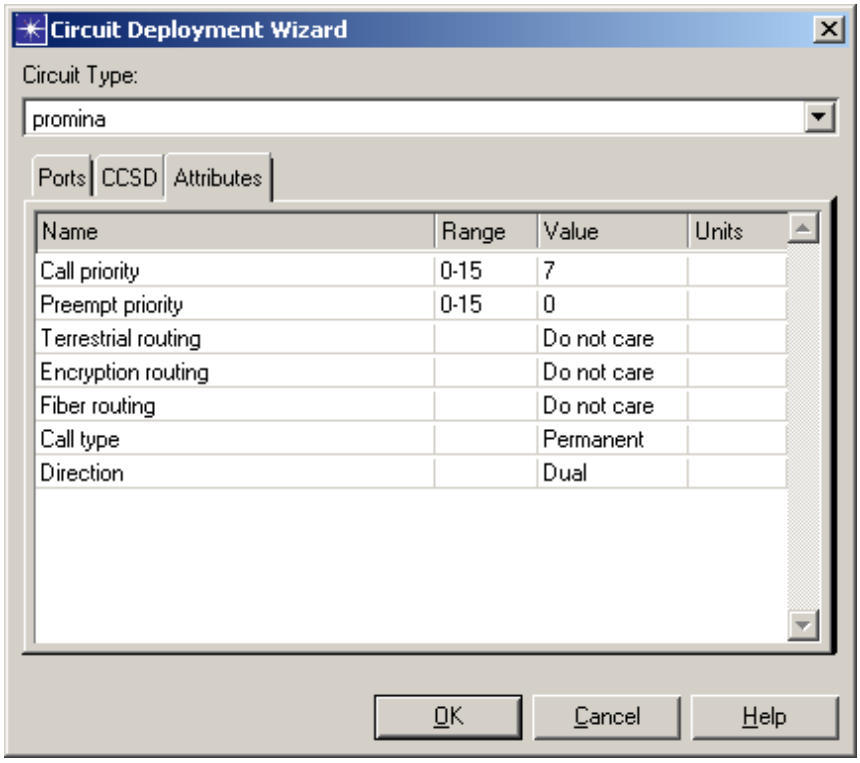


Figure 3-121 Circuit Deployment Wizard Attributes Tab

5) Click the **OK** button to complete the Promina circuit configuration and deploy the circuit.

Deploy Broadcast Network

Topology > Deploy Broadcast Network: Define broadcast networks by selecting units in the workspace, and then selecting **Topology > Deploy Broadcast Network**.

1) Select the source and destination OPFACs or organizations in the workspace, and then select **Topology > Deploy Broadcast Network** (or press **Ctrl + Shift + B**). The Network Attributes dialog box displays.

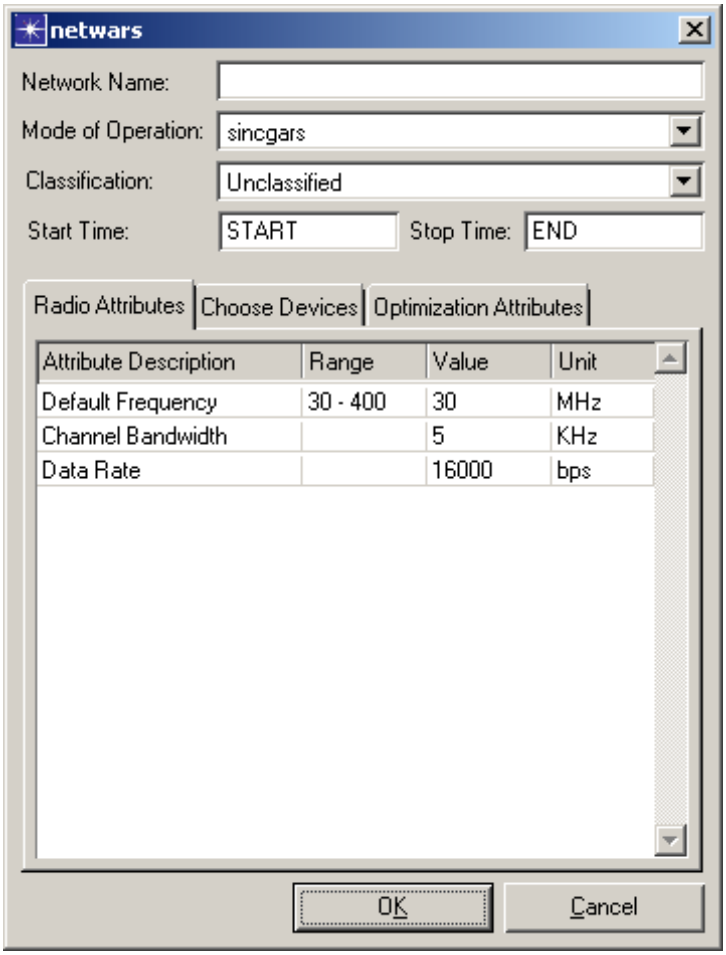


Figure 3-122 Defining Broadcast Network Radio Attributes

2) Type a name for the broadcast network in the Network Name field.

3) On the Radio Attributes tab, enter the desired attributes for default frequency, channel bandwidth, and data rate.

4) On the Choose Devices tab, select devices for the corresponding OPFACs or organizations.

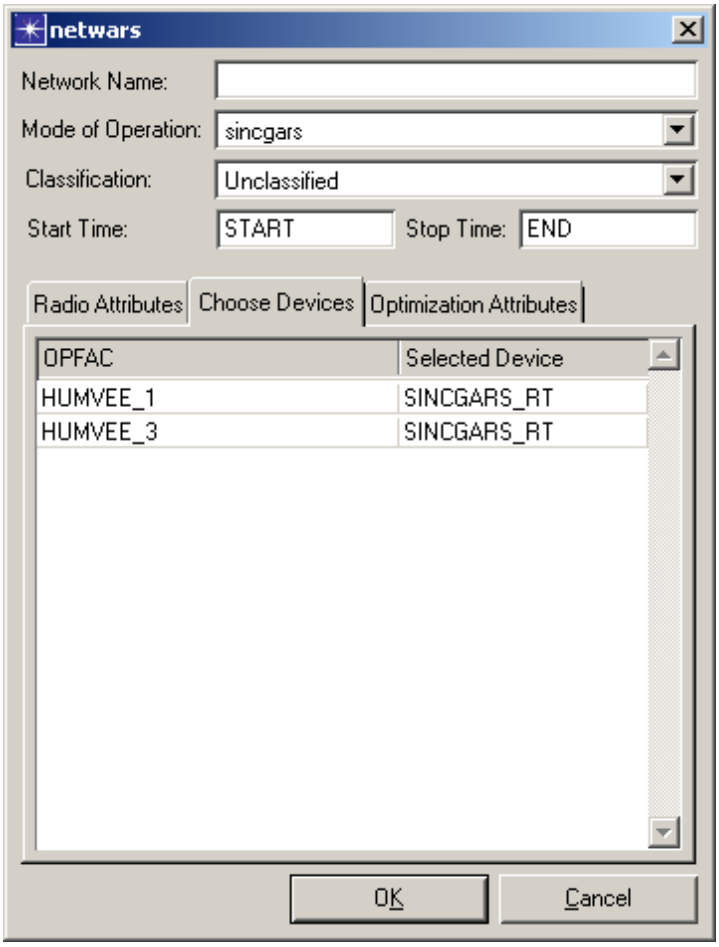


Figure 3-123 Defining Broadcast Network Devices

5) On the Optimization Attributes tab, enter the desired attributes for possible capacities, and target and maximum utilization.

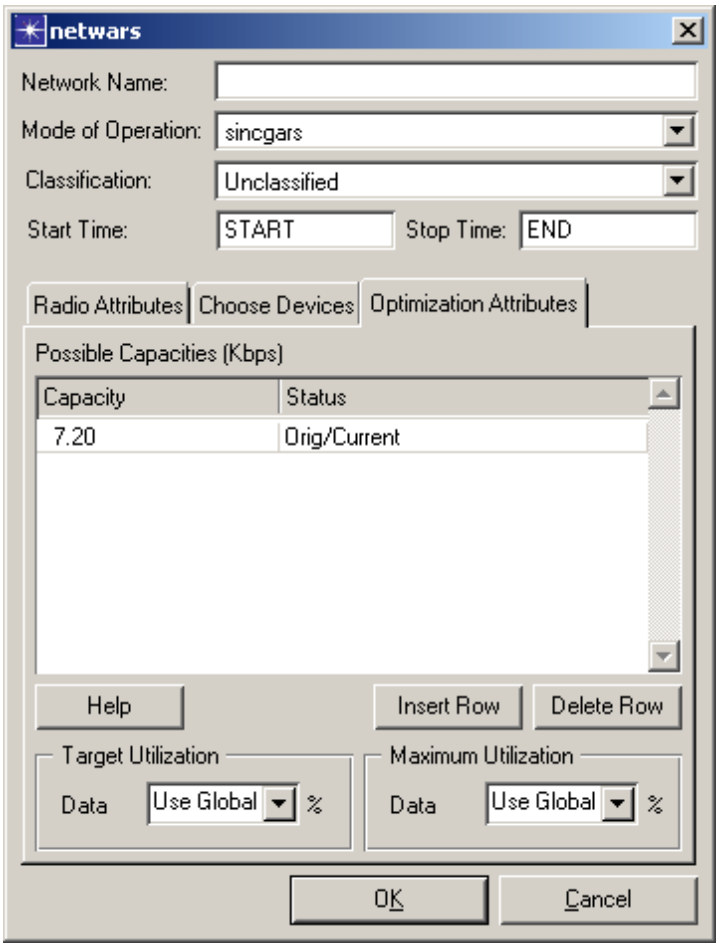


Figure 3-124 Defining Broadcast Network Optimization Attributes

a) Possible Capacities table: Specifies all of the capacities that the optimization engine can use when mutating solutions. For example, if there are two capacities specified, then the engine will be able to set either of them as the suggested capacity for this network. The Capacity column is where the capacity is specified. The Status column is used to mark the “original” capacity and the “current” capacity (or “Orig/Current” if the current capacity is also the original capacity.)

b) **Insert Row** button: To add a new capacity, click this button and type the new capacity into the Capacity column of the new row.

c) **Delete Row** button: To remove a capacity, select a cell in the row and click this button.

d) Target Utilization: Specifies the target utilization to be achieved on this network. The default value is to use the global values specified in the Capacity Optimization Settings dialog. You can override this just on this network by specifying a new value.

e) Maximum Utilization: Specifies the maximum utilization that is allowed on this network. The default maximum utilization is set to use the global values specified in the Capacity Optimization Settings dialog. You can override this just on this network by specifying a new value.

6) Click **OK** to create the newly defined broadcast network.

Deploy Relationship

Topology > Deploy Relationship: Define relationships by selecting units in the workspace, and then selecting **Topology > Deploy Relationship**.

1) Select the OPFACs that you want to associate in the workspace, and then select **Topology > Deploy Relationship** (or press **Ctrl + Alt + R**). The Relationship Attributes dialog box displays.

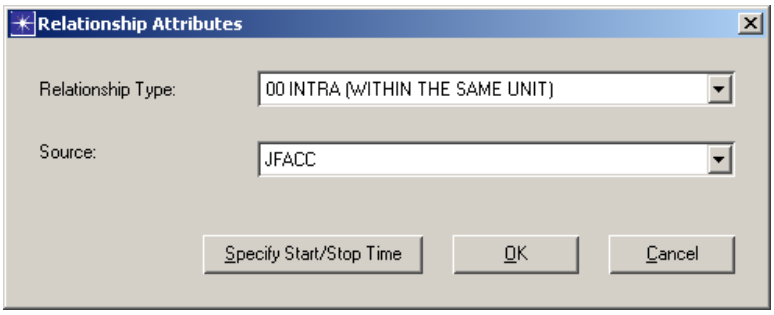


Figure 3-125 Creating a Relationship

2) Select the type of relationship from the Relationship Type drop-down menu.

3) Select the source OPFAC from the Source drop-down menu.

4) Click **OK**. The relationship between the selected OPFACs is created.

Configuration Utilities > UHF DAMA SATCOM

Topology > Configuration Utilities > UHF DAMA SATCOM: Follow the steps in the Configuration Wizard to configure the desired UHF DAMA SATCOM utility node. You must select at least one NCS device and two terminals in order to start the wizard.

1) Select **Topology > Configuration Utilities > UHF DAMA SATCOM** to start the wizard.

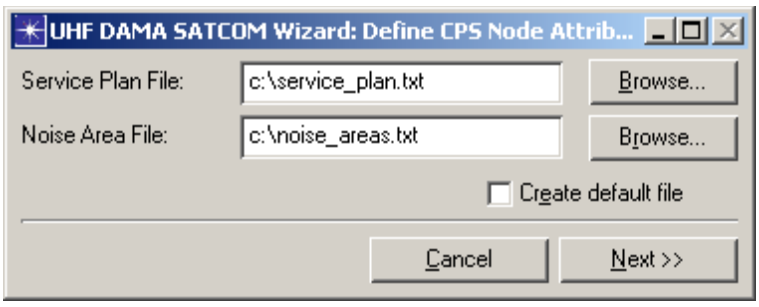


Figure 3-126 UHF DAMA SATCOM Wizard: Define CPS Node Attributes (Step 1 of 3) dialog box

2) Browse and select a service plan file, and browse and select a noise area file (or check **Create default file** to create a default file.) Click **Next>>** to continue.

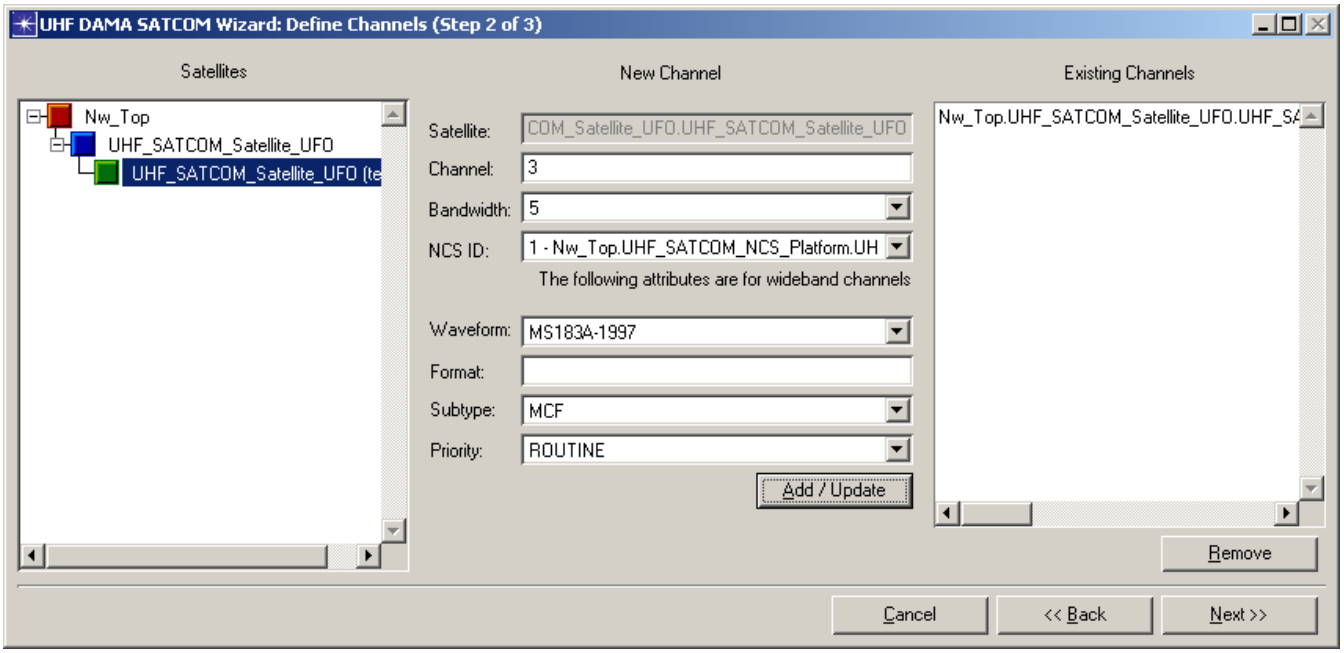


Figure 3-127 UHF DAMA SATCOM Wizard: Define Channels (Step 2 of 3) dialog box

3) Specify the settings for the channel that you want to define, and then click **Next>>** to continue.

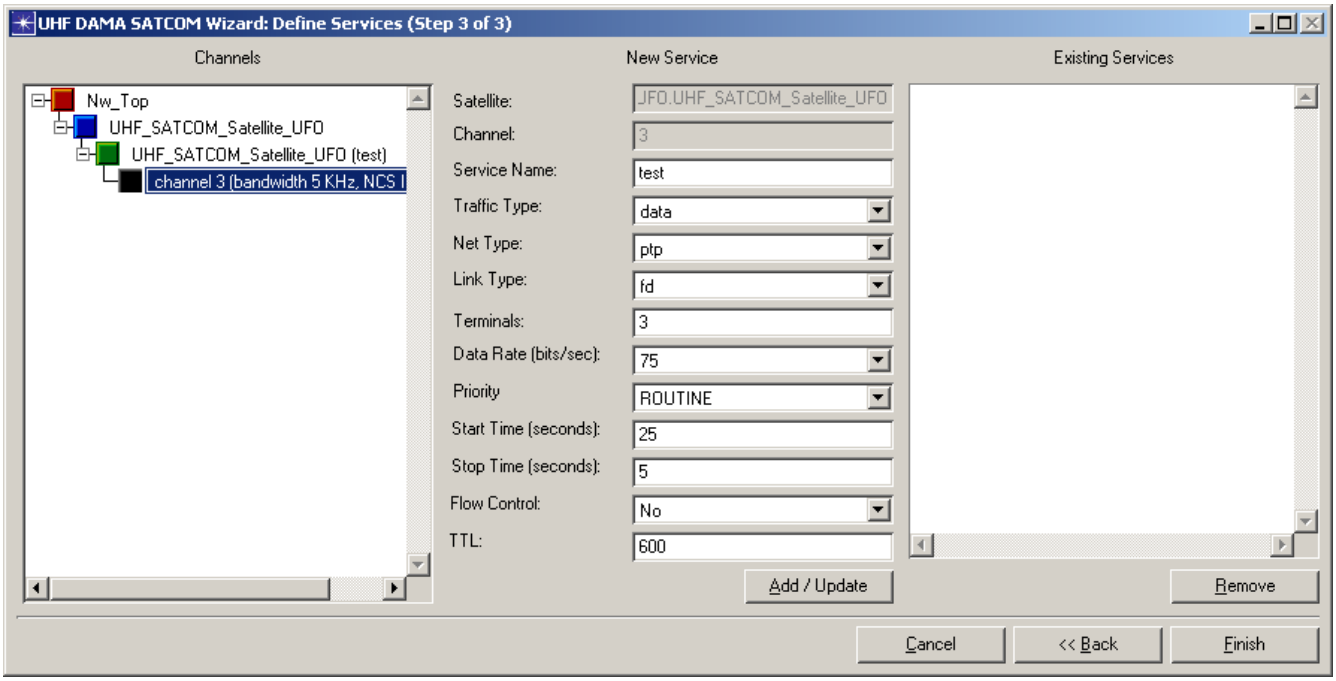


Figure 3-128 UHF DAMA SATCOM Wizard: Define Services (Step 3 of 3) dialog box

4) Specify the settings for the service that you want to define, and then click **Finish** to end and close the wizard.

Configuration Utilities > TSSP > Configure TSSP Groups (Circuits)

Topology > Configuration Utilities > TSSP > Configure TSSP Groups (Circuits): After you have deployed a satellite link, you can create a TSSP circuit by selecting exactly two objects (devices or OPFACs) and configuring the circuit via the TSSP Group Config (Circuit Deployment) dialog box.

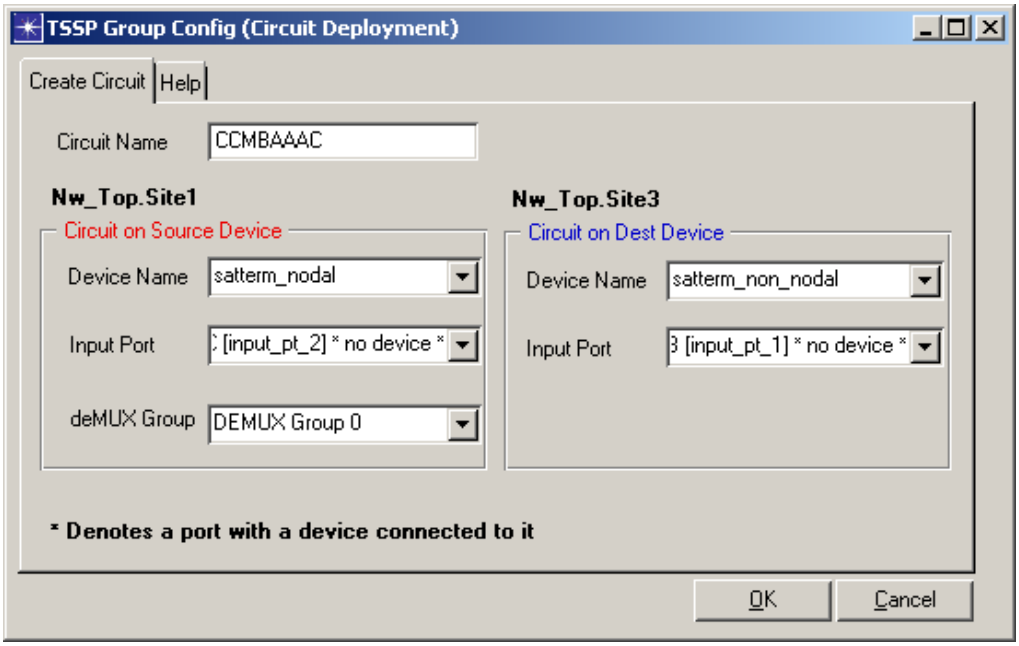


Figure 3-129 TSSP Group Config (Circuit Deployment) dialog box ~ Create Circuit

1) Click the Help tab. The following screen displays a diagram which explains the function of this dialog box .

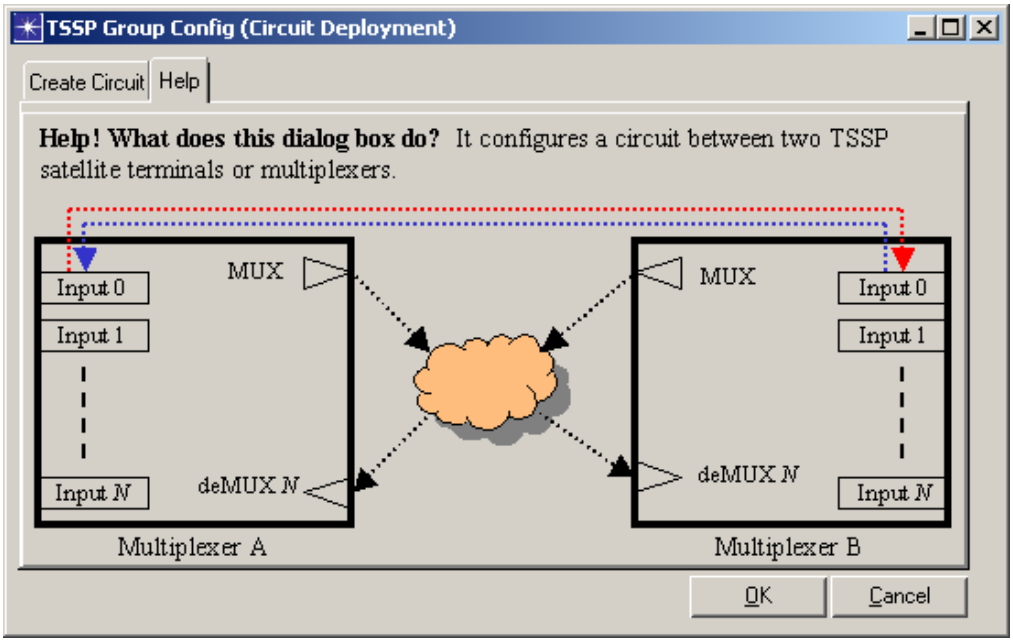


Figure 3-130 TSSP Group Config (Circuit Deployment) dialog box ~ Help

Note—Shortcut keys are provided for launching the Circuit Deployment dialog box for TSSP circuits. SImply press **Ctrl + Alt + T**.

2) Use the default values provided on the Create Circuit tab in the TSSP Group Config (Circuit Deployment) dialog box (or select the desired values), and then click **OK**.

   A message box displays to tell you that the circuit was successfully created.

3) Click **Create Another** to re-display the TSSP Group Config (Circuit Deployment) dialog box, or click **Done** to close the dialog box.

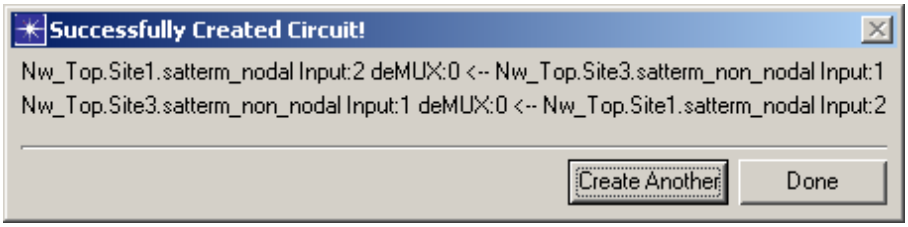


Figure 3-131 Successfully Created Circuit message box

Configuration Utilities > TSSP > View TSSP Group Configs (Circuits)

Topology > Configuration Utilities > TSSP > View TSSP Group Configs (Circuits): View configured TSSP circuits in a table format, and delete selected circuits. The table includes the following circuit configuration information: Name, Device A Name, Device A Input Port, Device A Demultiplexer (deMUX), Device B Name, Device B Input Port, and Device B Demultiplexer (deMUX).

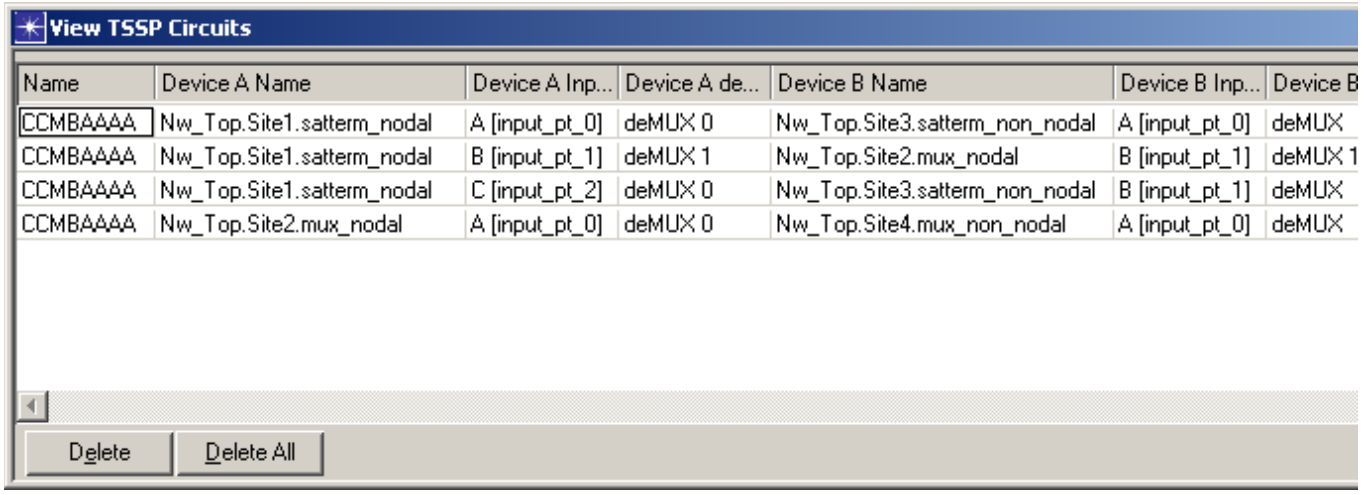


Figure 3-132 View TSSP Circuits dialog box

Note—Shortcut keys are provided for launching the View TSSP Circuits dialog box. SImply press **Ctrl + Alt + U**.

Configuration Utilities > Multiplexer > Configure Multiplexer Circuits

Topology > Configuration Utilities > Multiplexer > Configure Multiplexer Circuits: The process for configuring multiplexer circuits is the same as the process for configuring TSSP circuits (see the previous section in this manual.) You can create a multiplexer circuit by selecting exactly two devices and configuring the circuit via the Multiplexer Circuit Deployment dialog box.

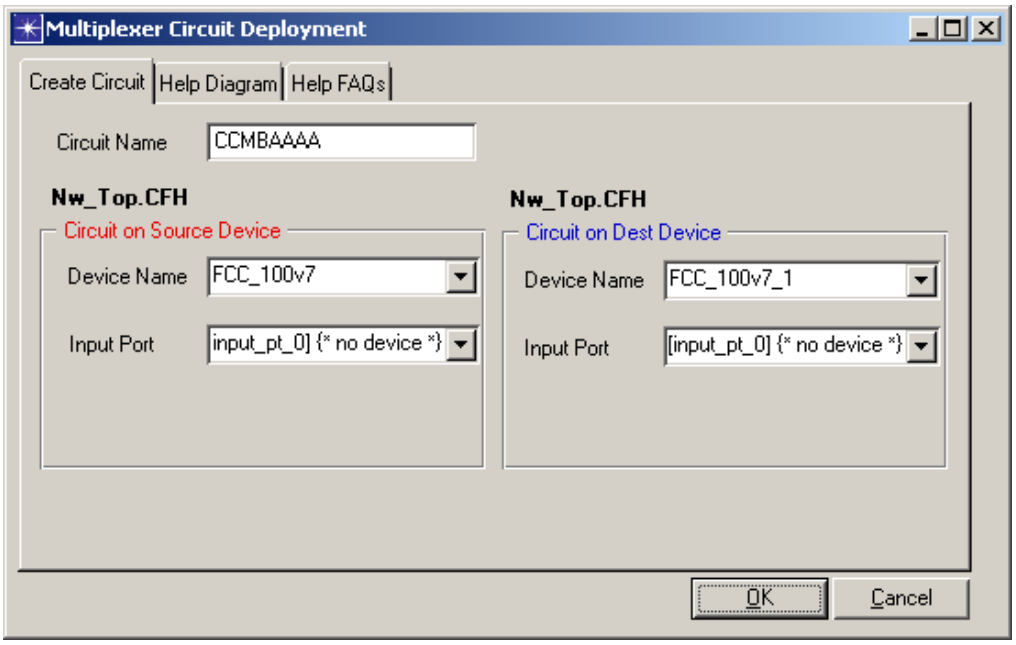


Figure 3-133 Multiplexer Circuit Deployment dialog box ~ Create Circuit

---

Note—Shortcut keys are provided for launching the Circuit Deployment dialog box for multiplexer circuits. SImply press **Ctrl + Alt + M**.

---

1) Click the Help Diagram tab. The following screen displays a diagram which explains the function of this dialog box.

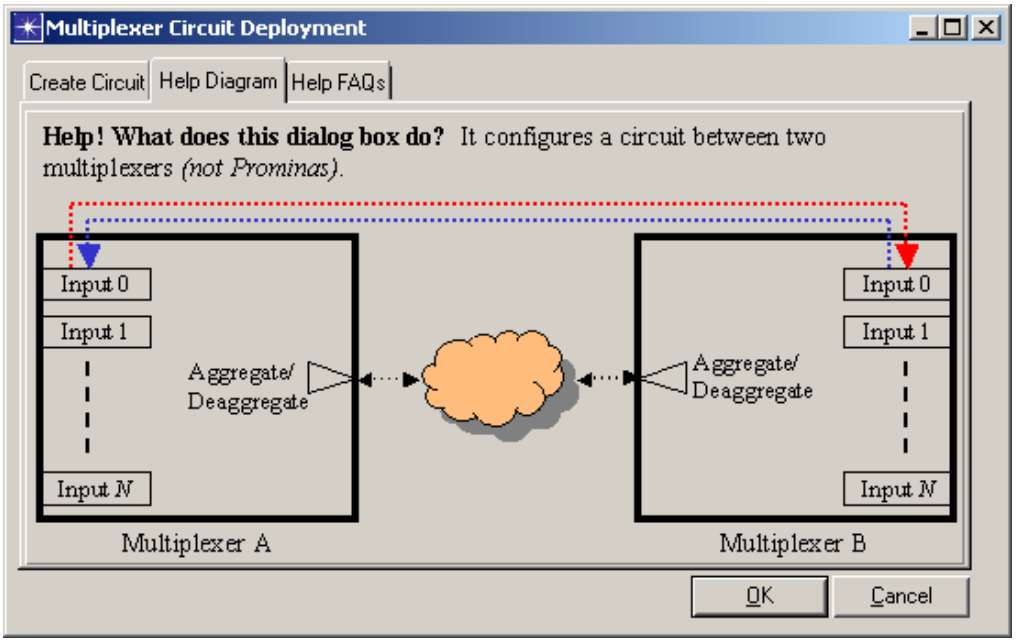


Figure 3-134 Multiplexer Circuit Deployment dialog box ~ Help Diagram

2) Click the Help FAQs tab. The following screen displays answers to frequently asked questions about multiplexer circuit deployment.

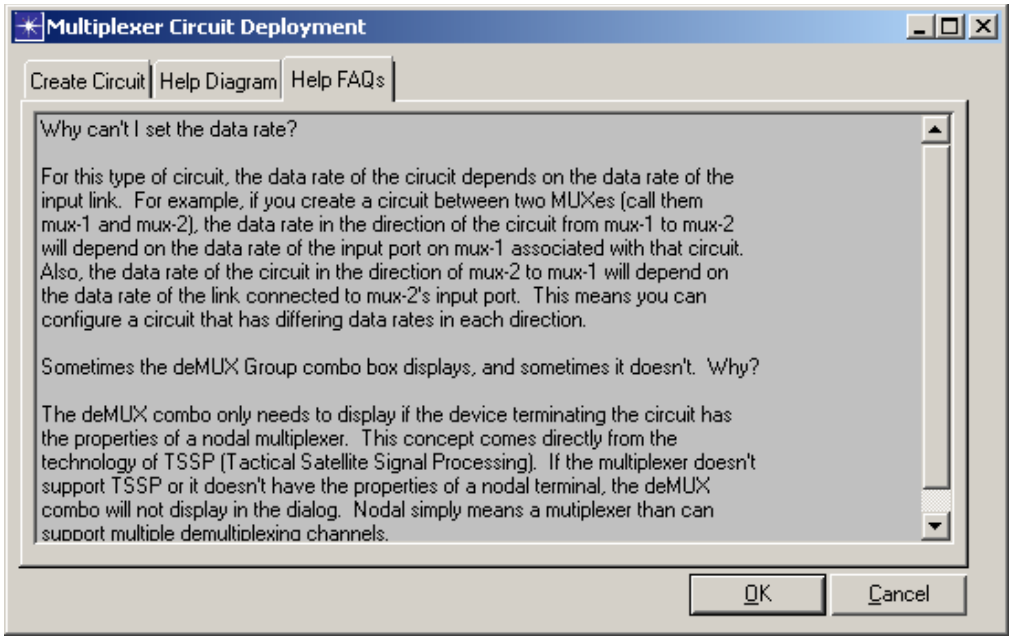


Figure 3-135 Multiplexer Circuit Deployment dialog box ~ Help FAQs

3) Use the default values provided on the Create Circuit tab in the Multiplexer Circuit Deployment dialog box (or select the desired values), and then click **OK**.

   A message box displays to tell you that the circuit was successfully created.

4) Click **Create Another** to re-display the Multiplexer Circuit Deployment dialog box, or click **Done** to close the dialog box.

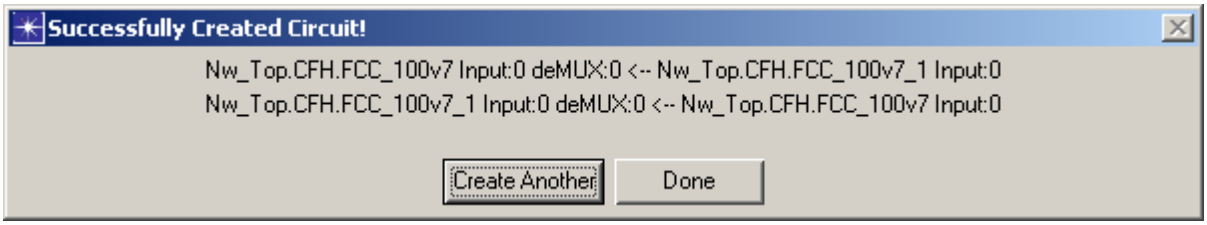


Figure 3-136 Successfully Created Circuit message box

Configuration Utilities > Multiplexer > View Multiplexer Circuits

Topology > Configuration Utilities > Multiplexer > View Multiplexer Circuits: View configured multiplexer circuits in a table format, and delete selected (or all) circuits. The table includes the following circuit configuration information: Name, Device A Name, Device A Input Port, Device A Demultiplexer (deMUX), Device B Name, Device B Input Port, and Device B Demultiplexer (deMUX).

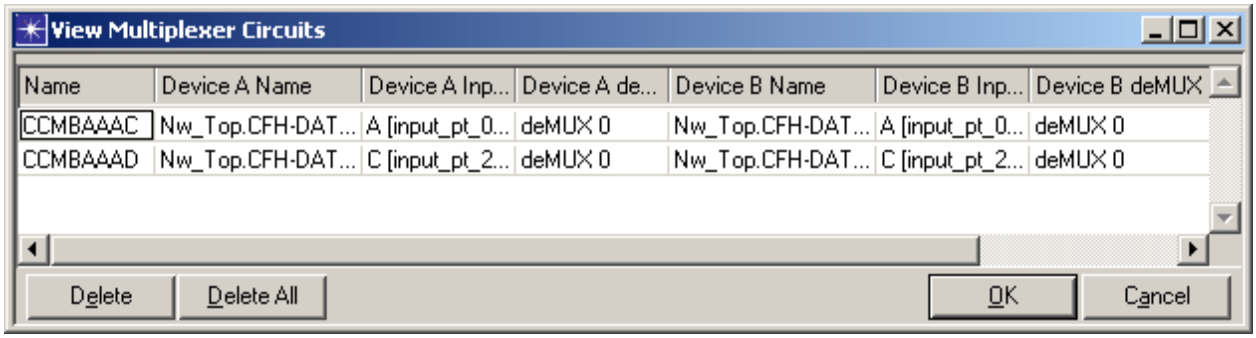


Figure 3-137 View Multiplexer Circuits dialog box

Note—Shortcut keys are provided for launching the View Multiplexer Circuits dialog box. SImply press **Ctrl + Alt + O**.

Configuration Utilities > Cut-through > Enable

Topology > Configuration Utilities > Cut-through > Enable Cut-through on Selected Devices (All Ports): Select the two devices that you want to be cut-through enabled, then select this command (or press **Ctrl + Alt + E**). The selected devices will be cut-through enabled on all ports. (Note the message that displays in the status bar at the bottom of the Scenario Builder window.)

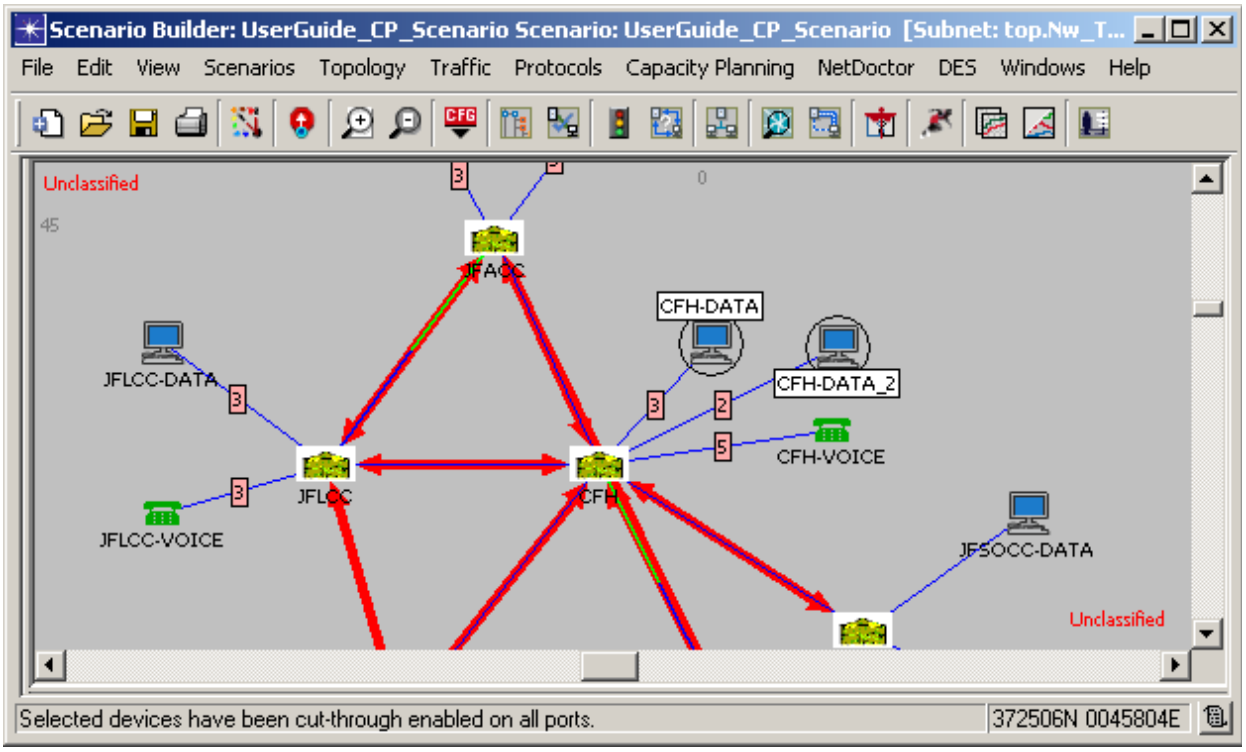


Figure 3-138 Enable Cut-Through on Selected Devices

Configuration Utilities > Cut-through > Disable

Topology > Configuration Utilities > Cut-through > Disable Cut-through on Selected Devices (All Ports): Select two cut-through enabled devices that you want to be cut-through disabled, then select this command (or press **Ctrl + Alt + D**). The selected devices will be cut-through disabled on all ports. (Note the message that displays in the status bar at the bottom of the Scenario Builder window.)

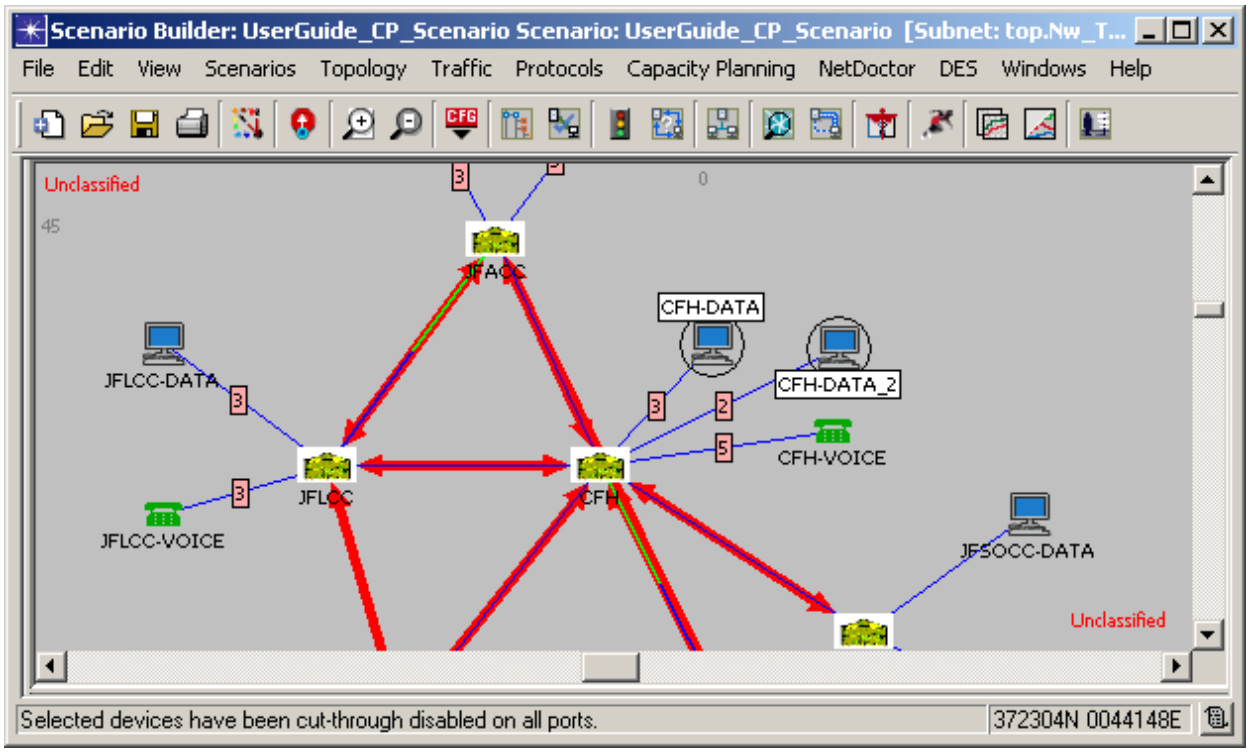


Figure 3-139 Disable Cut-Through on Selected Devices

Set Owner

Topology > Set Owner: Determine which units are needed for subordinate's input (performed by Lead Planner), and then mark them for subordinate input by setting ownership information.

Alternative method to access this option: right-click on a unit and select **Edit NETWARS Attributes**, and then click the **Set Owner** button.

1) Select units to be marked for subordinate before accessing this option.

2) After this option is accessed, click on the **Last Name** drop-down menu to select an owner.

   a) An owner can be set on multiple units through one operation by enabling the **Applying owner information on all selected units** option.

b) If the desired owner is not listed in the drop-down menu, add the desired owner to the list by clicking on the **New Owner** button.

Figure 3-140 Set Owner dialog box

Once the owner information is applied to the desired units, those units will display a yellow icon indicating the owner has been set and those units have been marked. Units are unmarked by default; therefore, the Lead Planner must remember to set ownership information on the units before exporting the subordinate query files to Subordinate Planners.

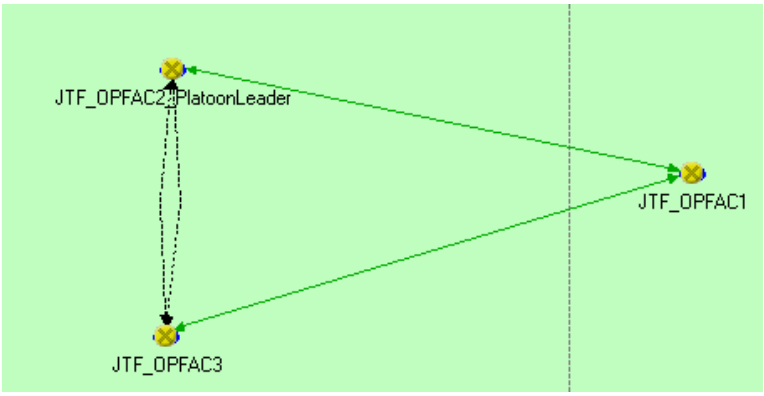


Figure 3-141 Owner Information Set on Desired Units

Remove Owner

Topology > Remove Owner: Remove ownership from selected units.

Alternative method to access this option: right-click on the unit, select **Edit NETWARS Attributes**, click the **Set Owner** button, and then enable the **Unset Owner Information** checkbox.

1) Select units to be unmarked before accessing this option. Once the remove owner operation is applied, the yellow icons are removed.

Terrain

Topology > Terrain *<options>*: This menu is available only if a Terrain Modeling Module GUI (TMMGUI) license is obtained. Use this feature to import terrain data, view the terrain profile and study the effects of the atmosphere on the communication.

JCSS supports the following terrain data formats:

- DTED Level 0
- DTED Level 1
- 15-minute DEM data
- 2-arc-second DEM data
- 1-degree-DEM data

Terrain > View Terrain Profile

Topology > Terrain > View Terrain Profile: View the terrain profile between two points.

1) In View Profile mode, click on one point and then click on the second point. This will display the terrain profile between the two selected points.

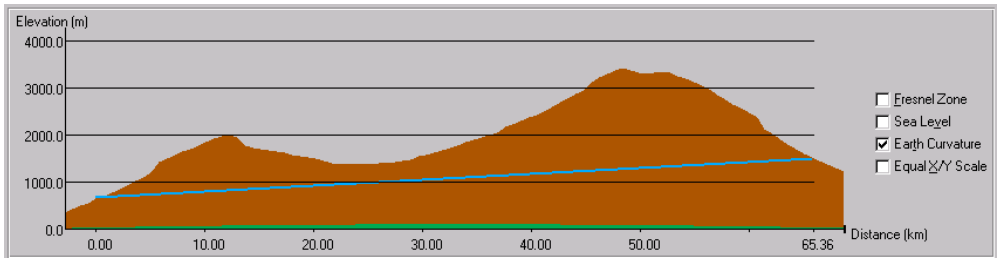


Figure 3-142 Terrain profile

The brown area indicates terrain. Using this interface, you can turn on or off the Fresnel Zone and also the Earth's curvature (which is represented by the green arc). The blue line in the profile can be used to determine whether line-of-sight exists between the two points. If the blue line crosses over the terrain or the Earth's curvature, there is no line-of-sight.

Terrain > Set Propagation Model

Topology > Terrain > Set Propagation Model: Set the model used by the TMMGUI module for representing the atmospheric conditions. Example propagation models include the Free-Space model and the Longley-Rice model You can also create user-defined propagation models.

Terrain > Edit Parameter Sets

Topology > Terrain > Edit Parameter Sets: Access the tabular interface for making changes to the propagation models.

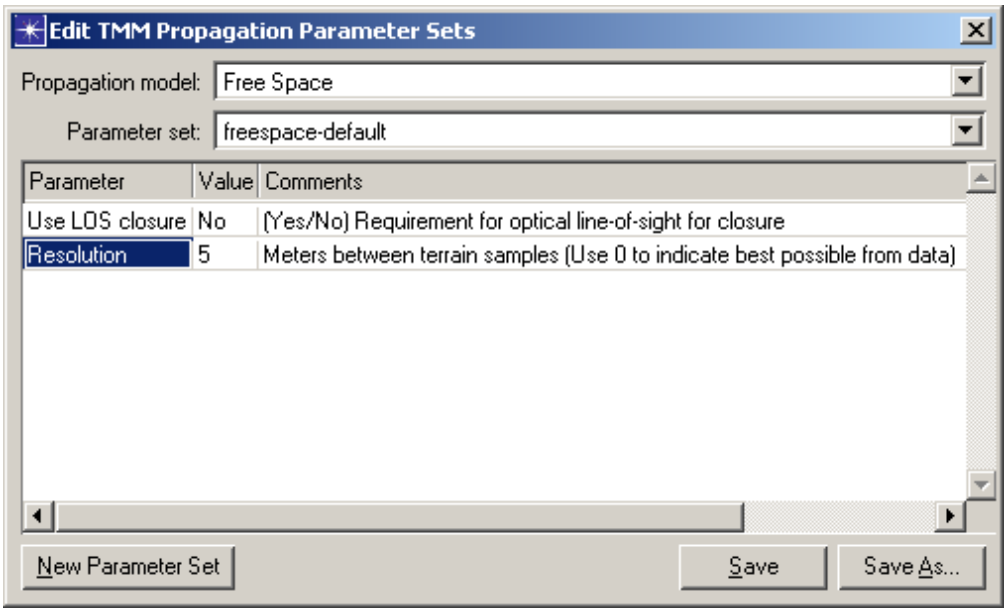


Figure 3-143 Edit TMM Propagation Parameter Sets dialog box

1) From the drop-down menu, select the propagation model to modify.

2) Once the desired changes to the propagation model are made, save it to the same model or save it to a new model (sing the **Save** or **Save As** buttons) and close this table.

Terrain > Specify Terrain Data Directory

Topology > Terrain > Specify Terrain Data Directory: Set the directory where JCSS will access terrain data.

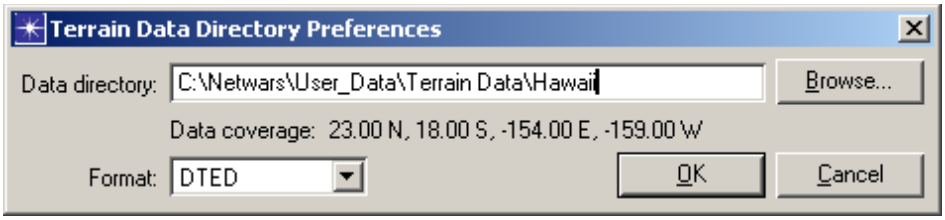


Figure 3-144 Terrain Data Directory Preferences dialog box

1) Type in the folder name where the files are stored or use the **Browse** button to browse to that location using the file browser.

   JCSS supports three types of terrain data—DTED, USGS DEM, and OpenFlight. The specified folder must contain the files of the specified type.

Terrain > Rebuild Terrain Data Catalog

Topology > Terrain > Rebuild Terrain Data Catalog: Refresh terrain data before import if files are added to the terrain data directory or changes are made to the location of the terrain data directory. JCSS does not automatically detect the changes.

Terrain > Set Elevation Maps

Topology > Terrain > Set Elevation Maps: Set elevation or contour lines on the map. If terrain data has already been imported, it can be saved as elevation lines and can be reused in future sessions of JCSS. This saves the time taken to import the data and create the elevation lines.

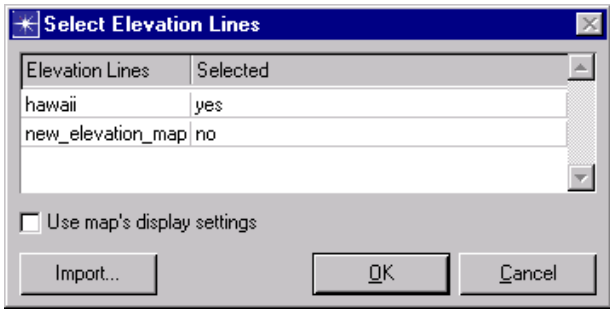


Figure 3-145 Select Elevation Lines dialog box

1) If there are no elevation lines for the area of interest, import terrain data and use the new elevation lines. Click the **Import** button to launch the Import Elevation Lines dialog box.

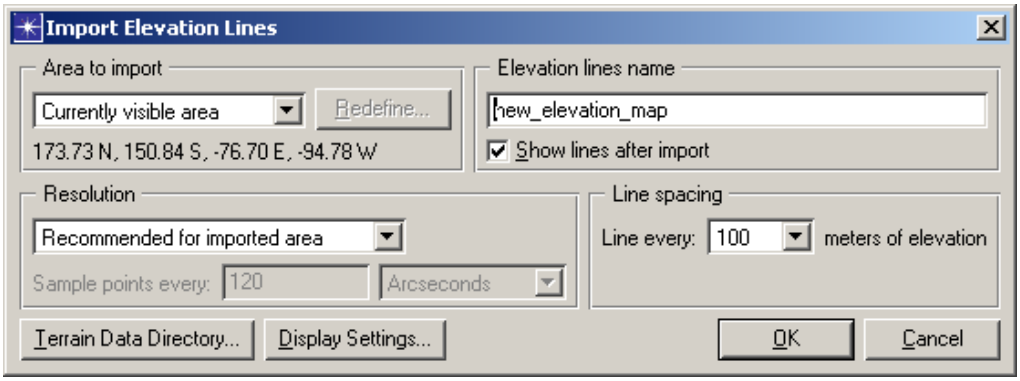


Figure 3-146 Import Elevation Lines dialog box

2) When importing the data, specify the following information:

   a) **Area to Import**: Specifies the portion of the map corresponding to which one you want to import the terrain information. You have three choices: **Currently Visible Area**, **Entire Subnet**, or **Define Area**.

   b) **Elevation Lines Name**: Identifies a name for the elevation lines being imported. By saving the elevation lines, you can avoid going through the import process again.

   c) **Resolution**: Indicates the number of sample points per unit.

   d) **Line Spacing**: Specifies how far away the elevation lines are spaced when imported into the scenario.

   e) **Terrain Data Directory**: Click this button to select the folder that contains the terrain data. This launches the same interface as the **Terrain > Specify Terrain Data Directory** menu option.

f) **Display Settings**: Click this button to control the display preferences. This option launches the same interface as the **Terrain > Edit Map Display Settings** option.

3) Click **OK**.

Terrain > Edit Map Display Settings

Topology > Terrain > Edit Map Display Settings: Specify map display settings, such as the colors and thresholds to be used for the elevation lines.

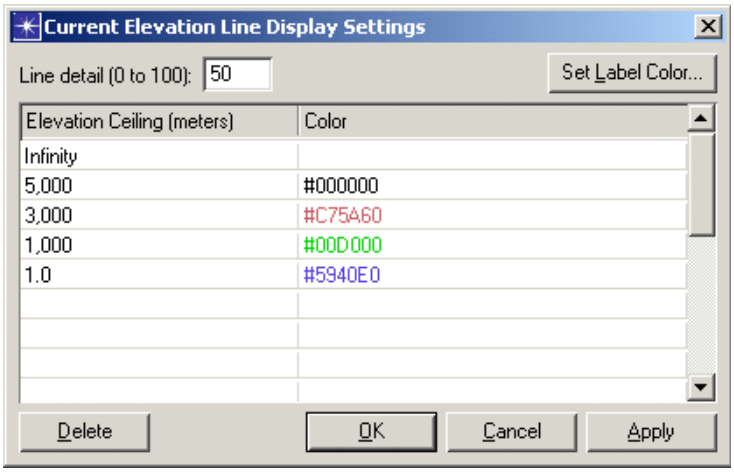


Figure 3-147 Current Elevation Line Display Settings dialog box

Define Trajectory

Topology > Define Trajectory: Create a trajectory defined in segments.

1) Define a trajectory type, how long the unit waits at the starting location before it starts moving, and the altitude at which the unit is when it starts its trajectory.

   a) **Trajectory name** – Sets the name of the trajectory.

   b) **Trajectory type** – The types are Variable interval and Fixed interval.

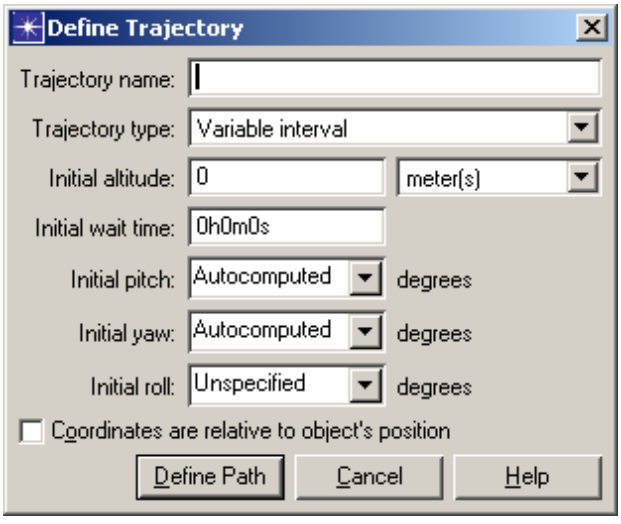


Figure 3-148 Define Trajectory dialog box

c) **Initial altitude** – This field sets the initial altitude of the trajectory. Choose the units of measurement for the altitude with the pull-down menu.

d) **Initial wait time** – In a variable-interval trajectory, the initial wait time pauses a mobile object for a specified time before it covers the next segment.

e) **Initial pitch** – Sets the initial pitch of the trajectory. Possible assignments are: **Autocomputed** to assign a value automatically according to the trajectory; **Unspecified** to not assign any value; or **0.0,** which you can edit to assign an explicit value.

f) **Initial yaw** – Sets the initial yaw of the trajectory. Possible assignments are: **Autocomputed** to assign a value automatically according to the trajectory; **Unspecified** to not assign any value; or **0.0,** which you can edit to assign an explicit value.

g) **Initial roll** – Sets the initial roll of the trajectory. Possible assignments are: **Autocomputed** to assign a value automatically according to the trajectory; **Unspecified** to not assign any value; or **0.0,** which you can edit to assign an explicit value.

h) **Coordinates are relative to object's position** – You can specify a trajectory that is either relative to an object's initial position or has its own absolute initial position. If you define a relative trajectory, all objects that are assigned this trajectory will trace out the same path starting at their initial positions. If you define an absolute trajectory, all the objects that are assigned this trajectory jump to the trajectory's initial position before traversing it. Also indicate whether the trajectory's path points are absolute or relative to the object's current location.

2) Click **Define Path** to enter trajectory definition mode, display the Trajectory Status dialog box, and change the cursor into a line tool in the Project Editor**.** This is used to draw a node's mobile trajectory; the path displays in red.

3) When you define a variable interval trajectory, the Trajectory Segment Information dialog box displays after you create a path. This dialog box is used to specify trajectory information for each segment.

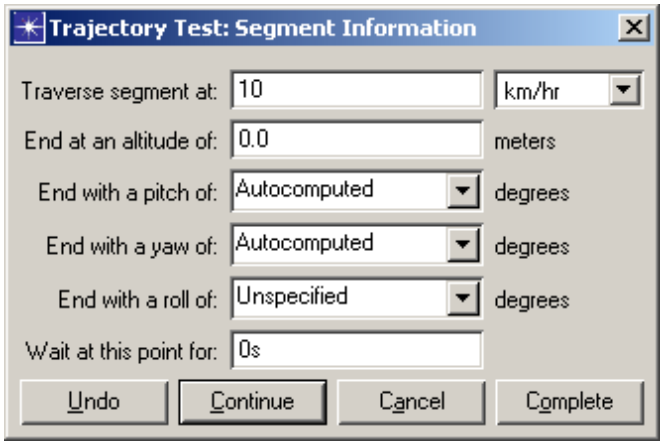


Figure 3-149 Trajectory Segment Information dialog box

4) Click **Continue** to add a segment, and/or click **Complete** when you are finished.

5) From the Scenario Builder, right-click on the desired unit to open the shortcut menu.

6) Select **Edit NETWARS Attributes**. The OPFAC/Organization Attributes dialog box displays.

7) Select the name of the trajectory from the **Trajectory** drop-down menu.

8) Click **OK**.

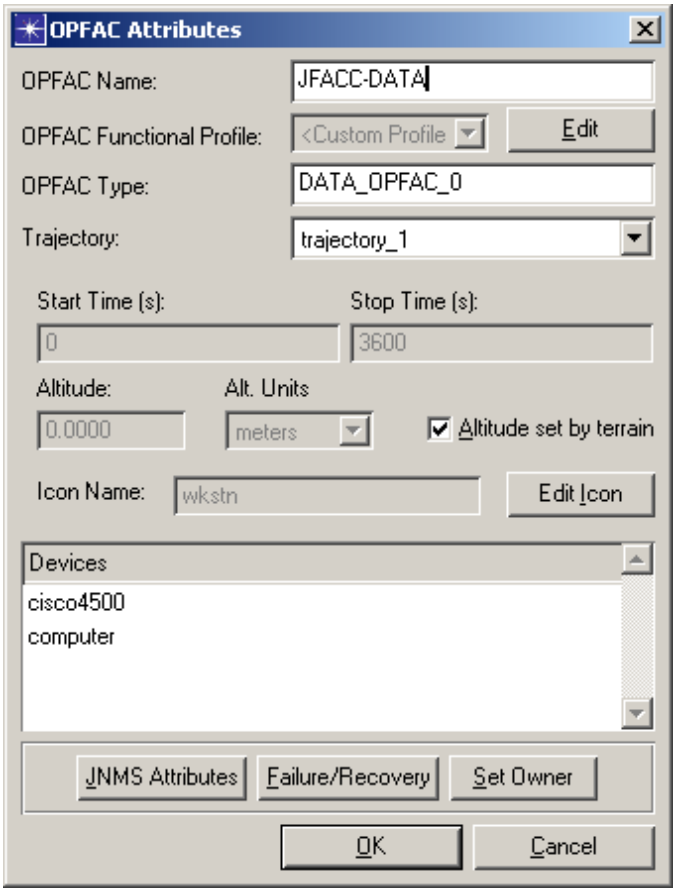


Figure 3-150 Assigning Trajectory

The assigned trajectory displays as a light-green arrow.

Figure 3-151 An Assigned Trajectory

Traffic Menu

Traffic Wizard

Traffic > Traffic Wizard: Launch the Traffic Wizard which provides users with a quick and simple way to deploy traffic within JCSS. Select two OPFACs, and then the Traffic Wizard walks you through selecting the end devices (and the profile to deploy between the client and server devices), and deploying a link between them.

Note—The Traffic Wizard will launch only if you have selected two OPFACs between which the traffic is to be deployed.

---

1) Select **Traffic > Traffic Wizard** (or press **<Ctrl>** + **W**) to launch the Traffic Wizard.

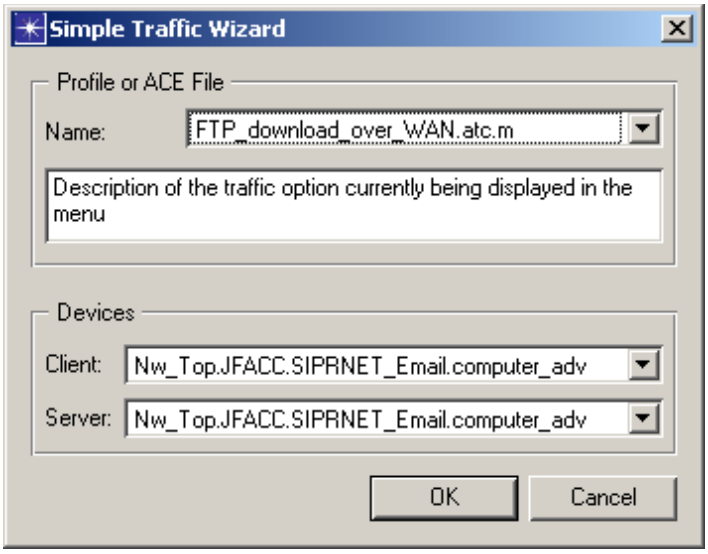


Figure 3-152 Traffic Wizard

2) Select a profile to deploy, specify client and server devices, and then click **OK**.

IERs > Set Import Options

Traffic > IERs > Set Import Options: Specify the various sources from which traffic is imported into the scenario.

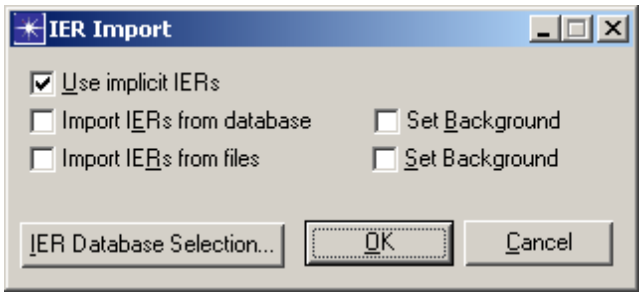


Figure 3-153 IER Import dialog box

- **Use implicit IERs**: If this option is checked, the software creates implicit relationships based on the user-created relationships and creates IERs for these newly created relationships.
- **Import IERs from database**: JCSS can query databases containing IERs in the specified format. This is based on the producer and consumer OPFACs' functional names and the URC between them. If you check this option, the software imports IERs from the JCSS database.
- **Import IERs from files**: In addition to the database, JCSS can also import IERs from text files in the specified format. This is based on the producer and consumer OPFACs' functional names and the URC between them. If you check this option, the software imports IERs from such files.
- Additionally, you can specify all traffic coming from a source as **background**. This causes the simulation to treat all IERs coming from that source as background traffic, resulting in improved simulation efficiency.
- **IER Database Selection**: Designate a source database for a scenario by clicking this button. Since the database may exist anywhere on the network, you must pick both a server and a database.

  a) The first step in selecting a database is to login to the server. You only need to login to a database server once, but since each server may have a different set of passwords, changing servers may require a new server login.

  Note: During installation, JCSS creates a default database user account with username **scenario_builder** and password **netwars**.

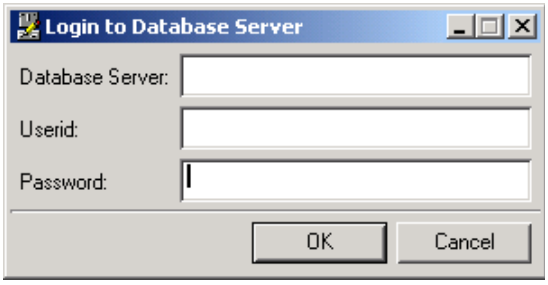


Figure 3-154 Database Server Login

b) Once logged into the server, the second dialog box displays to allow you to change the server or the database. The selection lists will contain all available servers, and the databases for the currently selected server. Of course, if you change the server, you may need to log in to that server before seeing the list of available databases.

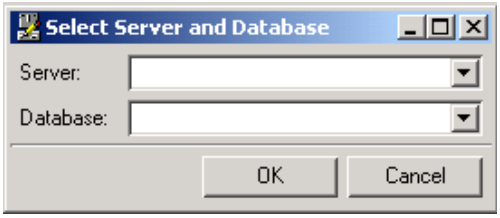


Figure 3-155 Select Server and Database

c) Click **OK** to update the scenario to use the designated server and database combination for database IER queries.

IERs > Specify IERs

Traffic > IERs > Specify IERs or click the **Specify IERs** toolbar button: Create IERs, and view existing IERs in the scenario.

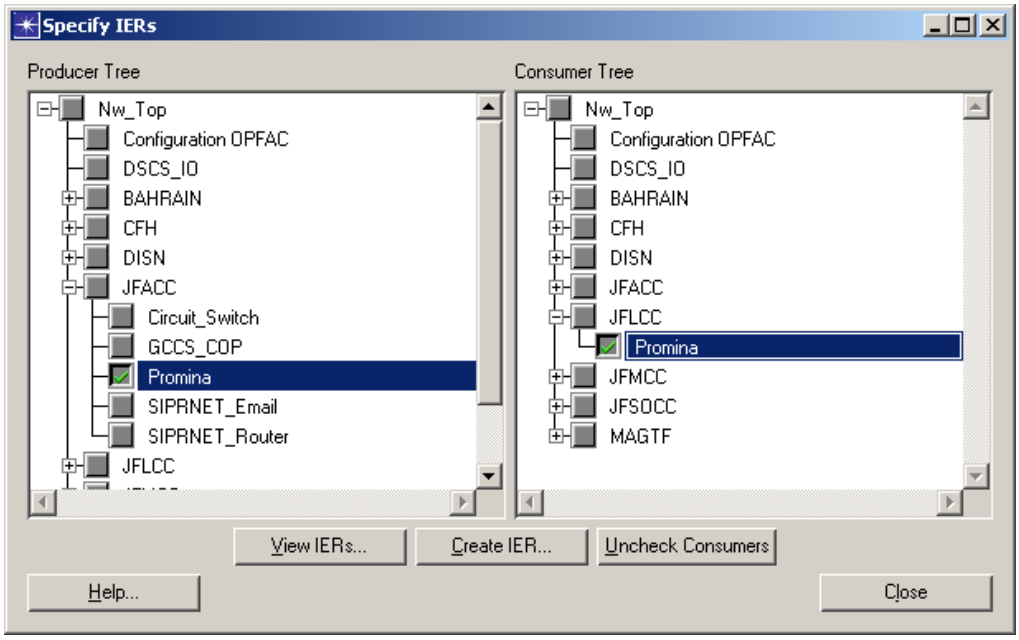


Figure 3-156 Specify IERs dialog box

- Producer Tree: This tree provides a hierarchical view of all the OPFACs and organizations in the current scenario. When creating new IERs, the producing OPFAC is selected in this tree. Only one OPFAC may be selected at a time.
- Consumer Tree: This tree provides the same hierarchical view of all the OPFACs and organizations that the Producer Tree does. When creating IERs, the consuming OPFAC(s) are selected in this tree. Multiple OPFACs may be selected at once.

- **View IERs**: If a producer is selected in the producer tree, clicking this button displays the Edit IER Parameters dialog box which shows all of the IERs produced by the selected OPFAC.

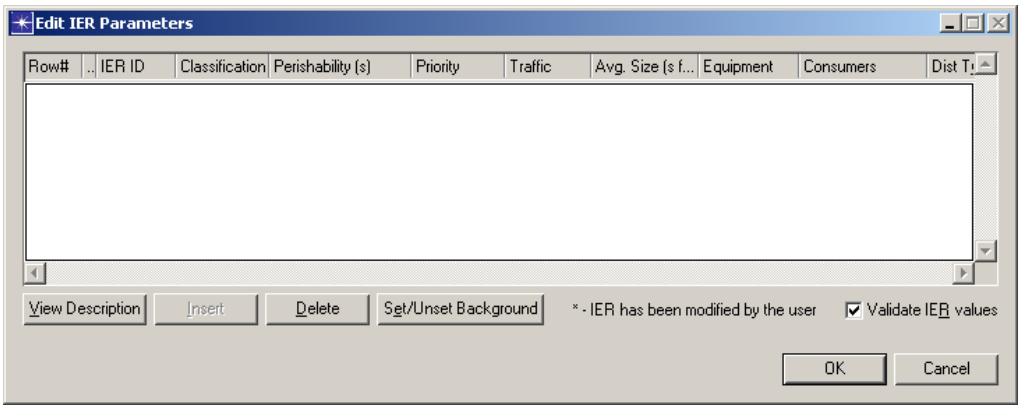


Figure 3-157 Edit IER Parameters dialog box

The following list provides definitions of IER attributes:

— **IER ID**: Identifies an IER in the database. IDs for IERs that are created by a user start with the prefix USER, as not to conflict with IER IDs in the database. Background IERs start with the prefix BKGD.

— **Classification**: Specifies the security classification of an IER. The security classification of an IER is one criterion that determines the system element through which the IER is transmitted.

— **Perishability**: Specifies the time in seconds for which the IER is alive.

— **Priority**: Determines the number of transmission retries and the wait time between successive retries.

— **Traffic**: Specifies the type of IER traffic.

— **Average Size**: Indicates the average size of the IER in bytes.

— **Equipment**: Specifies the system element on which the IER can be transmitted.

— **Consumer**: Identifies OPFAC consumer of the traffic.

— **Distribution Type**: Indicates the inter-arrival distribution for the IERs.

— **Interarrival**: Represents the time, in seconds, between IER firings.

— **Start Time**: Identifies the time, in seconds, in which the IER will begin firing after a simulation begins.

— **Stop Time**: Identifies the time, in seconds, in which the IER will stop firing after the simulation begins.

— **Producer Device** and **Consumer Device**: Indicate the devices transmitting and receiving the IER.

— **Transport Protocol**: Defines the underlying Transport Layer protocol that traffic (IERs) will use for transmission. Typical transport layer protocols include TCP and UDP.

- **Create IER**: If a producer and at least one consumer are selected, clicking this button displays the IER Specification dialog box where new IER(s) can be created. If only one consumer is selected, the single IER creation dialog box (shown below) displays. If multiple consumers are selected, a similar dialog box for multiple IERs displays instead.

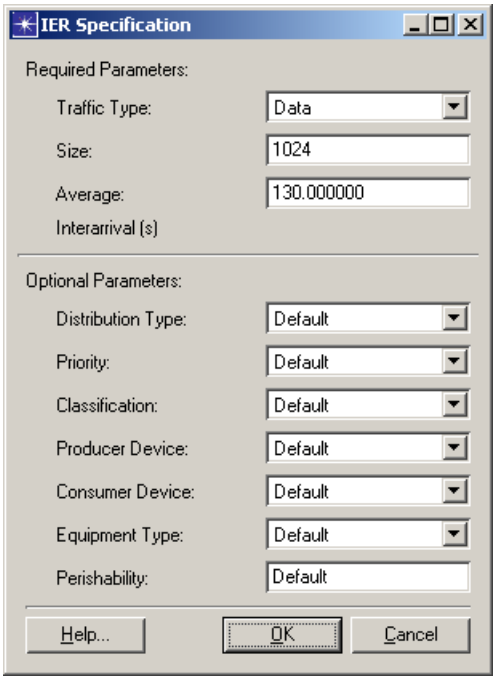


Figure 3-158 IER Specification dialog box for a single IER

The Required Parameters section has a set of parameters that have to be specified for the IER. The Optional Parameters section has some additional parameters. If no values are specified for the Optional Parameters, default values will be used.

— **Traffic Type**: This is the type of IER - voice, VTC or data.

— **Size**: This is the size of the IER. For data IERs, the size is specified in bytes. For voice or VTC IERs, the size is specified in number of seconds.

— **Average**: This is the time (specified in seconds) between the start of two consecutive IERs.

— **Distribution Type**: The distribution type combined with the Average value is used to determine the load caused by this IER on the links and broadcast networks. The default distribution type is "Exponential."

— **Priority**: This is the IER's precedence. The default precedence is "Routine."

— **Classification**: This is the IER's security classification. The default classification is "Unclassified."

— **Producer Device**: This is the device in the producer OPFAC which will be used to transmit this IER. If the producer device is not specified, the Capacity Planner will choose a device in the producer OPFAC to transmit this IER.

— **Consumer Device**: This is the device in the consumer OPFAC which will receive this IER. If the consumer device is not specified, the Capacity Planner will choose a device in the consumer OPFAC to receive this IER.

— **Equipment Type**: This is the type of equipment that will be used to transmit this IER.

— **Perishability**: If the time taken by the IER to reach the destination from the source is greater than the perishability, the IER is recorded as "failed."

IERs > Import from IER Report

Traffic > IERs > Import from IER Report: Load a previously created IER Report.

IERs > Export IER Report

Traffic > IERs > Export IER Report: Export all traffic in the scenario (excluding threads) to a text file.

1) The Create IER Report dialog box displays.

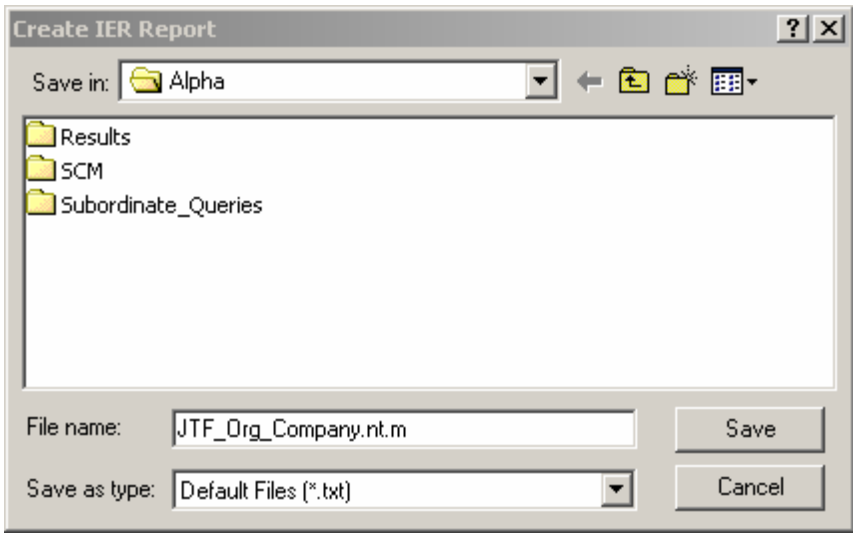


Figure 3-159 Create IER Report dialog box

2) Select a location and enter a name for the file, and then click **Save**.

IERs > Export IERs to XML

Traffic > IERs > Export IERs to XML: Automatically sends IERs to an XML file.

Flows > Open Flows Browser

Traffic > Flows > Open Flows Browser: Opens the Traffic Center, which allows you to view and edit traffic flows in your network.

Threads are groups of IERs that are fired in segments. Threads are a conditional sequence of IERs that represent complex real world information exchanges.

1) Right-click on an OPFAC with an established relationship to be the starting point of the IER thread.

2) Select **View Threads** from the right-click menu to open the View Threads dialog box.

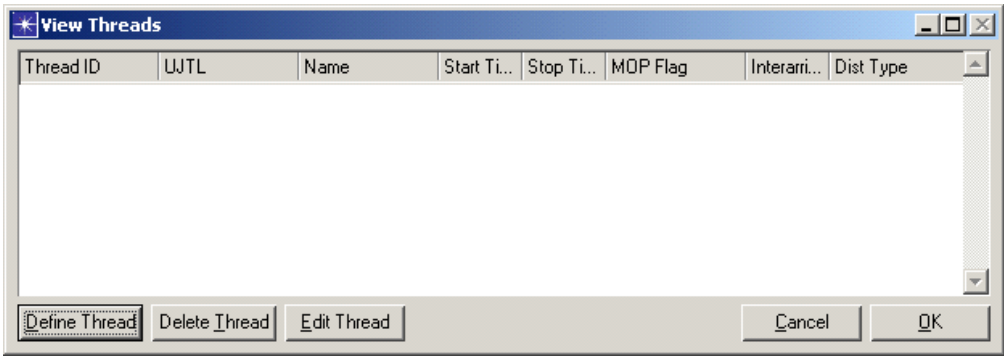


Figure 3-160 View Threads dialog box

3) Click **Define Thread** to open the Define Thread dialog box.

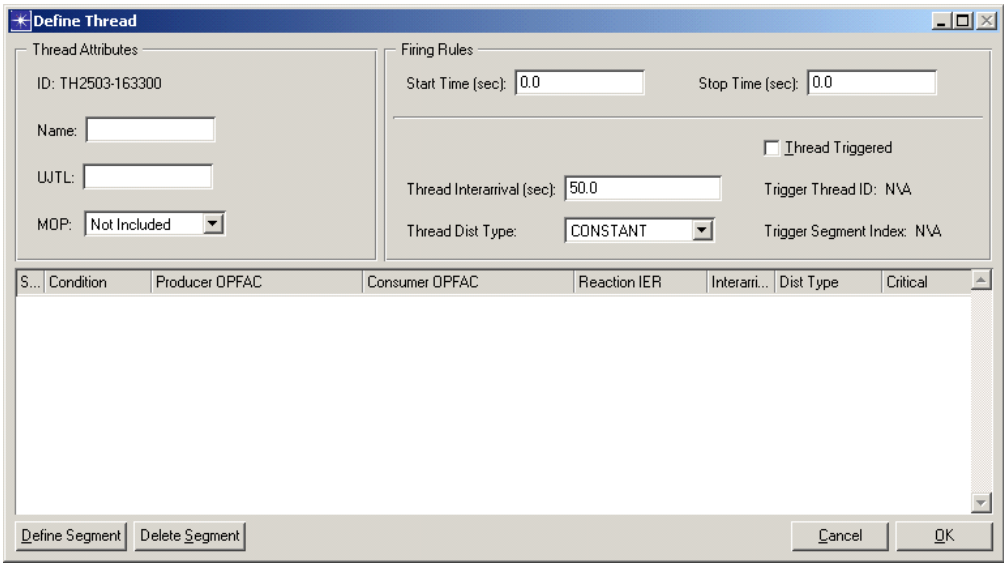


Figure 3-161 Define Thread dialog box

4) In the **Name** field, enter a name for the thread.

5) If you want to gather statistics on this thread after running a simulation, select **Included** in the **MOP** field.

6) On the Firing Rules panel, set the **Start Time** and the **Stop Time**, which control when the thread will start firing and stop firing after the simulation begins.

7) Set the **Thread Interarrival**, which determines the firing interval in seconds (e.g., 50 means the thread will fire every 50 seconds).

8) Select an option from the **Distribution Type** field, which indicates the inter-arrival distribution for the IERs.

9) Set the desired **Thread Trigger** option. The thread trigger is used to fire off a thread once a condition is met. For example, when a status report thread reaches the commanding OPFAC, that condition could kick off another thread being sent back down the chain of command representing marching orders.

10) Click **Define Segment**. A default IER segment inserts into the table.

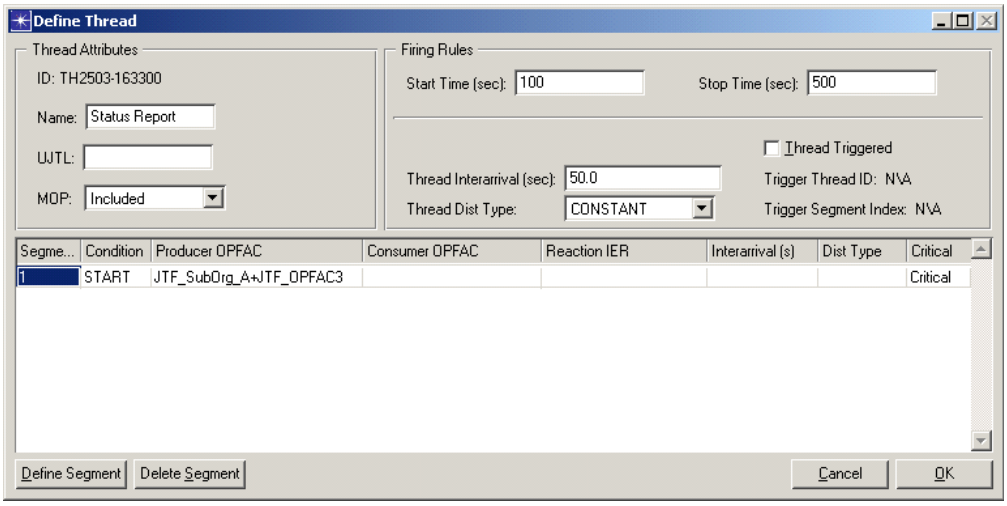


Figure 3-162 Adding a thread segment

Notice the default values for the first three attributes: **Segment Index**, **Condition**, and **Producer OPFAC**.

This IER is the first segment of the thread, so it is assigned **Segment Index 1**. This sequential number identifies unique segments within the thread.

The **Condition** attribute defines the condition leading to the execution of the respective segment. This field can be either **Start**, to signify the start time event of the thread, or an integer, to refer to a segment index within the thread that causes this segment to fire.

The OPFAC originally selected in step 2 in the Define Infrastructure dialog box is identified as the **Producer OPFAC**.

11) Click in the **Consumer OPFAC** cell to specify an OPFAC consumer of the thread.

12) Next, click in the **Reaction IER** cell. This is where you actually define the IER segment. The View Thread IERs dialog box displays. This dialog box is similar to the View Traffic dialog box.

13) Click the **Insert** button. A new explicit IER containing default settings is created in row 1 of the table.

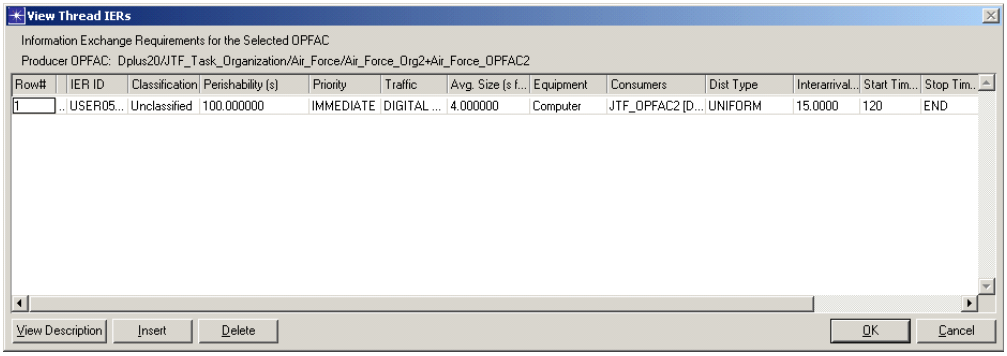


Figure 3-163 View Thread IERs dialog box

14) Click on the **Traffic** cell and use the drop-down menu to select a value that specifies the type of IER traffic.

15) Click in the **Average Size** cell and change the bytes to indicate the average size of the IER in bytes.

16) In the **Equipment** cell, specify the system element on which the IER can be transmitted.

17) Click in the **Start Time** cell and select **Thread**. This means the IER will only fire as part of the thread. If the **Begin** value is selected, the IER will fire as part of thread and as an individual IER.

18) Specify the time between IER firings in the **Interarrival** cell.

19) Use the **Distribution Type** cell to indicate the inter-arrival distribution for the IERs.

20) The **Critical** cell is used for reporting/results analyses purposes. A thread is considered a success only if all critical IER segments are received. A thread is still considered a success if non-critical IER segments fail.

21) Create the next segment of the thread. Click **Define Segment**. Another default IER segment inserts into the table.

22) In the **Condition** cell of segment 2, enter **1** to define the condition leading to the execution of this segment. This means segment 1 within the thread causes segment 2 to fire.

23) Click in the **Consumer OPFAC** cell of Segment 2. From the drop-down menu, choose **Select Other OPFAC**. The Select Other OPFAC dialog box displays. Expand the scenario tree, and then select the desired OPFAC. Click **OK**.

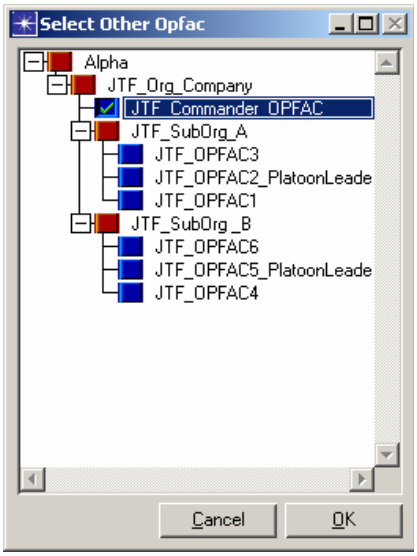


Figure 3-164 Select Other OPFAC dialog box

24) Next, click in the **Reaction IER** cell to define the IER segment. The View Thread IERs dialog box displays. When defining a segment for an OPFAC, all previously created traffic originating for that OPFAC also becomes available.

25) Create a new IER or use one that was previously created. Select the desired IER by clicking in the **Row#** cell, and then clicking **OK**.

26) From the Define Threads dialog box, either add another thread segment using the steps outlined previously, or click **OK**. When you click **OK**, a summary of the newly created thread displays in the View Threads dialog box.

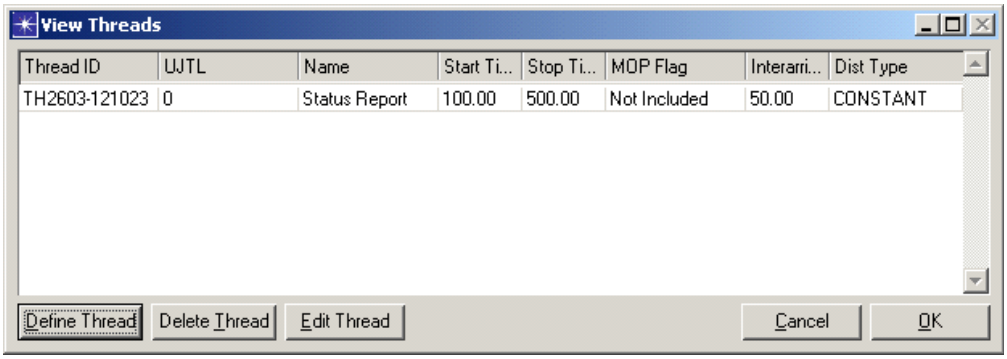


Figure 3-165 Newly Created Thread

27) Click **OK** to save the thread.

Flows > Import

Traffic > Flows > Import > (*option*): The Import sub-menu contains operations related to importing traffic; consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**, for further details. What follows below is a sample workflow for importing Cisco Netflow traffic files; workflows are similar for the other data sources listed above; however, there may be subtle differences.

Netflow Import Process

The traffic import for Netflow traffic files involves a two-step process. The first step involves the conversion of the traffic information included in the Netflow traffic file into traffic profiles for the appropriate nodes in the network while the second step involves converting the imported flow profiles into JCSS-specific traffic.

During the first step, each line in the Netflow traffic files is integrated into a traffic profile based on parameters such as the source and destination IP addresses and the aggregation scheme. Thus, if there are multiple lines in the file that have the same values for source, destination and aggregation scheme, then these lines will be integrated into a single traffic profile by the software. In the second step, each flow profile is converted into a single IER using the information that you provided in the Traffic Import Defaults dialog box.

The average size for the IER is calculated based on the total number of packets and total number of bits of the traffic profile while the inter-arrival time is calculated based on the difference between start and stop times and the total number of packets.

The minimum inter-arrival time, displayed in the Traffic Import dialog box, is used for those IERs where the calculated inter-arrival time is less than the user-specified minimum. In this case, the average size of the IER is increased proportionately. For example, if you specified inter-arrival time is 1 sec and the calculated inter-arrival time is 0.0002 secs, then the calculated average size is multiplied by the ratio of the user-specified minimum and the calculated inter-arrival time (1/0.0002) to get the new average size (new avg size = avg size x 1/0.0002).

If the calculated inter-arrival time is greater than the user-specified inter-arrival time, the calculated average size is not modified. For the case where the user-specified value for the minimum inter-arrival time is greater than the difference between the start and stop times, the size is computed based on the total number of bits and this IER is sent only once during the simulation.

1) Collecting Netflow Traffic Data

   The first step in importing traffic data is capturing or creating the Netflow traffic data file(s) to be imported. When capturing the Netflow traffic data, please ensure that the Netflow data file meets the following requirements:

   a) The data file must use one of the following aggregation schemes:

– RouterAS or RouterPrefix

– CallRecord or DetailCallRecord

– AsHostMatrix or DetailHostMatrix

b) The file may use either Format A or Format 2.

c) The file must be an uncompressed ASCII file. You may need to manually uncompress a file and/or convert it from binary to ASCII before you can import it.

d) The file must have a four-digit extension where the extension follows the "MMSS" (mins-secs) format.

e) The header of the file must include the correct format type, i.e. if the file is of Format A then the format field in the header must say Format A and not Format 2.

f) The number of records in the file must match the number of records specified in the header of the file.

If the header of the file is incorrect, the Start and End times in the Import Traffic Flows dialog box will be set to Jan 1, 1970.

For additional details, please refer to the *Traffic Import* section in the *MVI User's Guide* of your Modeler documentation.

2) Getting Started

To import traffic from Cisco Netflow traffic files, a project must currently be open in the Scenario Builder. Once a project has been loaded, start the import process by choosing the **Traffic > Flows > Import > From Cisco Netflow** menu item. Note that this menu item will be disabled (grayed out) if no project is open.

3) Deleting Previously Imported Netflow IERs

If you already performed a Cisco Netflow traffic import and chose to save the IERs created from that import, the software launches the following dialog box. This dialog box provides you with the option to delete all those IERs that were created as a result of previous Cisco Netflow traffic data imports.

4) Specifying the Traffic Import Defaults

The Traffic Import dialog box displays. Specify values for the parameters required in converting the imported flow profiles into IERs.

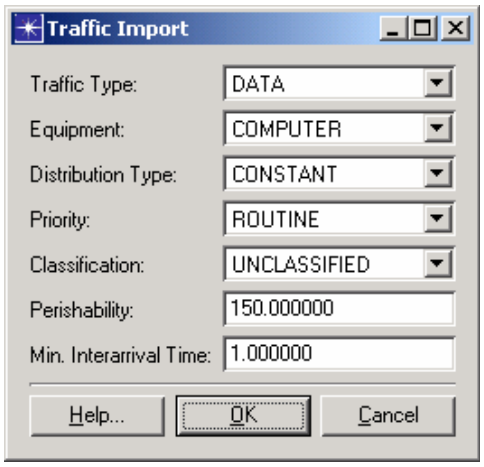


Figure 3-166 Traffic Import dialog box

The minimum inter-arrival time is used for those IERs where the calculated inter-arrival time is less than the user-specified inter-arrival time. When this occurs, the average size of this IER is increased proportionately. If the calculated inter-arrival time is greater than the user-specified inter-arrival time, the average size computed from the flow profile is not modified. For the case where the user-specified value for minimum inter-arrival time is greater than the difference between the start and stop times, the size is computed based on the total number of bits and this IER is sent only once during simulation.

5) Choosing the Netflow File

After selecting the **OK** button in the Traffic Import dialog box, the Traffic Flows Import dialog box will be launched. Click the **+** signs for all the subdirectories that appear under the Netflow directory. Click the **10.0.0.1.0000** traffic file that is included in the ASHostMatrix directory.

The directory that appears in the Traffic Flows Import dialog box is determined by the traffic_archive_dir preference that may be set via the Preferences dialog box. By default, this preference is set to the …\User_Data\Traffic_Data directory on your machine.

6) Add and/or remove directories and/or servers from the Traffic archive directories tree by clicking the **Add Directory**, **Add Server**, or **Remove** buttons.

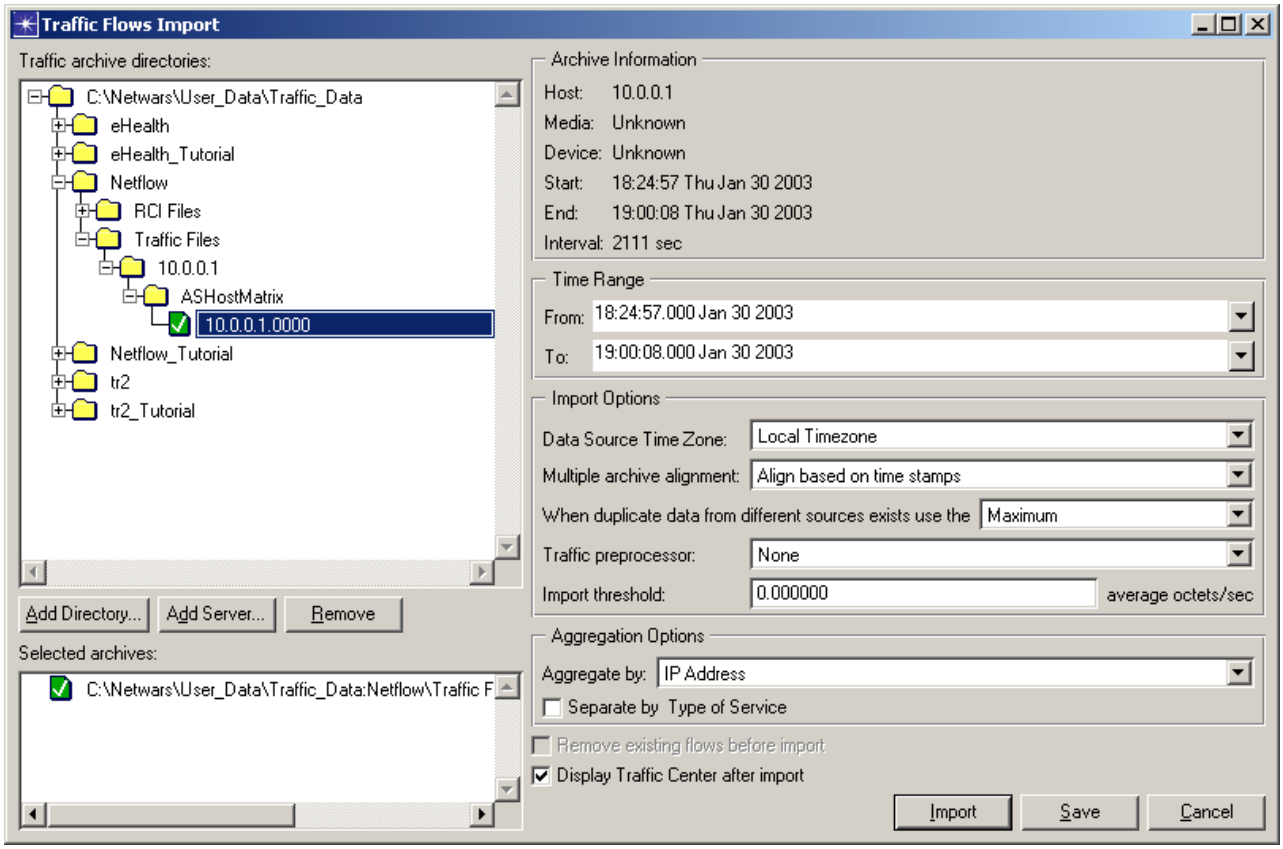


Figure 3-167 Traffic Flows Import dialog box

7) Specifying the Netflow Traffic Import Parameters

The Archive information corresponds to each traffic archive file and provides identifying information about the file:

a) **Host –** Specifies the IP address of the probe.

b) **Media –** The type of NetMetrix probe (Ethernet, FDDI, etc.) used to collect network traffic.

c) **Device –** The name of the NetMetrix probe.

d) **Start –** The start time of the traffic capture period.

e) **End –** The end time of the traffic capture period.

f) **Interval –** The amount of time between traffic collections.

The remaining sections include the following options that control the traffic import:

a) **From –** Indicates the start time of the period of traffic to import.

b) **To –** Indicates the end time of the period of traffic to import.

c) **Data Source Time Zone –** Indicates the time zone.

d) **Multiple archive alignment –** Indicates whether JCSS aligns or concatenates multiple archives from a single traffic server. This menu has three options:

Align Start of Each Archive – Each archive is aligned so that all traffic starts simultaneously (at simulation time = 0) and runs concurrently. This option is useful for archives from separate network probes that started simultaneously.

Align Archives Based on Time Stamps – The traffic is aligned based on the time stamps in each archive to replicate its real-world behavior. This option is useful for archives (from separate probes) that have overlapping time frames *but did not* start simultaneously.

Concatenate Archives – The archives are aligned by start time so that traffic runs consecutively: the traffic for each subsequent archive follows the traffic in the preceding archive. This option is most useful if you made successive runs from a single network probe.

e) **When duplicate data from different sources exists use the –** Indicates what merging mode to use when (1) you are importing from multiple traffic files, or, (2) you choose to retain (not overwrite) the current traffic data in your scenario. In both these cases the merge mode resolves any conflicting data in multiple traffic sources. Merging is useful when you have nodes on different segments, and probes on each segment as well. When nodes send traffic to one another, the probe on each segment will record the traffic, so that multiple values for the same traffic will be stored in the archive file. During import, you can set the Archive Merging Mode option either to Maximum, Addition, or Minimum. Setting the merge mode to Maximum selects the maximum of the two traffic values during import (this setting is most commonly used), while setting Minimum selects the smaller of the two traffic values. Choosing Addition adds the two traffic values.

f) **Traffic Preprocessor –** This menu lets you import traffic archives that use ASCII formats unsupported by JCSS. You can create preprocessor scripts (either shell scripts or executable files) that convert a traffic file to a format that JCSS can import. By selecting an archive and then choosing a preprocessing script in this menu, you can automatically convert the archive to Flexible CSV (.tr2) format prior to the import. Note that you cannot assign a script to multiple archives—you must assign a script to each archive individually. The initial setting for all archives is None. This tells JCSS to import the files “as is,” and is the correct setting for CSV, NetMetrix and Sniffer files.

g) **Import threshold –** Specifies the import threshold in average octets per second.

h) Aggregation options **–** Allows you to specify the method by which to aggregate traffic.

Select the **Import** button to proceed with the traffic import process.

8) Unrecognized Traffic

If OPNET could not map all the imported traffic to specific nodes in the network/scenario, the Unrecognized Traffic Assistant dialog box displays. You may then manually specify sources and/or destinations for any unrecognized traffic.

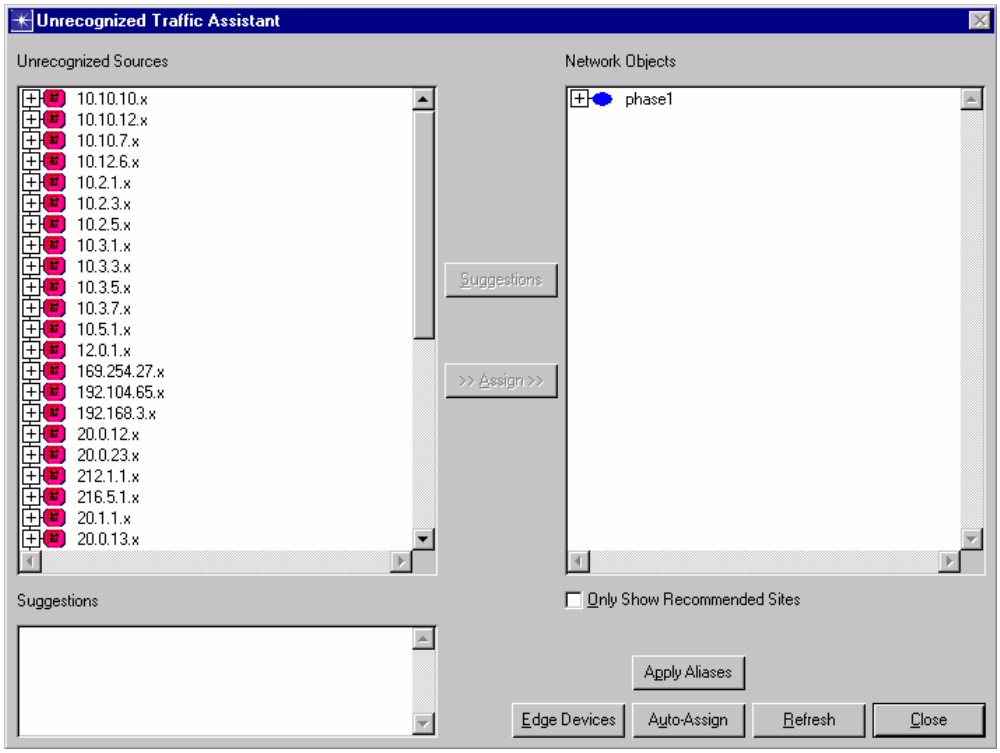


Figure 3-168 Unrecognized Traffic Assistant dialog box

For the cases where the exact source and destination nodes for the unrecognized traffic are unknown, you may assign certain nodes in your network to be edge devices. Specifying edge devices in your network will route this unrecognized traffic randomly between the edge devices in your network.

After you finish mapping some/all of your unrecognized traffic, select the **Close** button in the Unrecognized Traffic Assistant dialog box. The Remaining Unrecognized Sources dialog box displays, prompting you for confirmation to proceed with the traffic import.

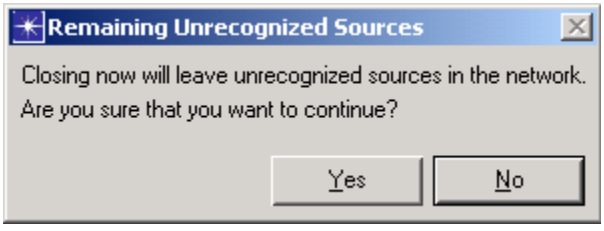


Figure 3-169 Remaining Unrecognized Sources dialog box

Selecting **Yes** in this dialog box launches the Reimport Traffic dialog box, giving you the option of re-importing all the traffic in this network.

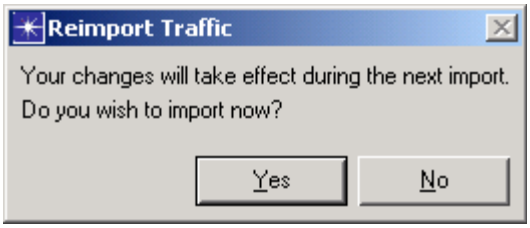


Figure 3-170 Reimport Traffic dialog box

If you click the **Yes** button, the Import Flows dialog box will be displayed again and this time choosing the Import button will include all of your recognized traffic as well as any unrecognized traffic that was mapped using the Unrecognized Traffic Assistant. Clicking the **No** button will not include any unrecognized traffic that was mapped during the import.

9) Examining the IERs

After the traffic import is complete, a message indicating the number of IERs created from this traffic import displays in the message buffer at the bottom right corner of the Scenario Builder.

The IERs created from this import are populated as background IERs on the source node and therefore may be viewed using the View Traffic dialog box. The View Traffic dialog box is accessible both from the Define Infrastructures treeview as well as from the right-click menu option of an OPFAC.

IERs created from a Cisco Netflow traffic import are tagged with DMD_CNF in their IER ID.

10) Visualizing the IERs

To view these IERs in your workspace, zoom in so that your units are visible and select the **Traffic > Show Aggregate Traffic** option.

Flows > Import > Open Import Log

Traffic > Flows > Import > Open Import Log: Opens the traffic flow import log, which contains error and diagnostic information generated during previous imports.

Flows > Import > Clear Import Log

Traffic > Flows > Import > Clear Import Log: Clears the link load import log.

Flows > Export > To Spreadsheet

Traffic > Flows > Export > To Spreadsheet: Exports traffic flows to an ASCII file that can be viewed as a spreadsheet in Microsoft Excel.

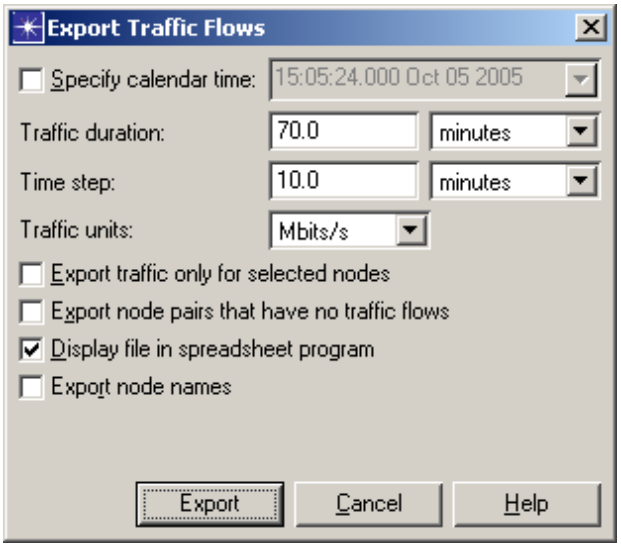


Figure 3-171 Export Traffic Flows dialog box

These options allow you to set parameters for the exported traffic:

- **Specify calendar time** checkbox and drop-down menu: Sets the start time of the traffic that the file spans. The time steps reference this date and time in the file. The default is determined by the earliest time set in all existing traffic pairs, network base time, or current time. To specify a date and time other than the default, select **Choose date/time** from the **Specify calendar time** pull-down menu. The Set Date/Time... dialog box appears. Select a date and time and click **OK**.
- **Traffic Duration** field: Sets the total time the traffic file spans. It is determined by the length of the conversation pair traffic in the scenario.
- **Time Step** field: Sets the bucket size of the exported traffic.
- **Traffic Units** drop-down menu: Lists the units of the exported traffic. This is specified in bits/sec, Kbits/sec, Mbits/sec, or Gbits/sec.
- **Export traffic only for selected nodes** check box: When selected, only traffic for selected nodes in the scenario is considered for export. When unselected traffic for all nodes is considered for export. This option is useful when you want to view and edit traffic for a few objects.
- **Export node pairs that have no traffic flows** check box: When selected, the entire matrix of node pairs is exported, including node pairs that have no traffic flows. When unselected, only node pairs that have traffic flows between them are exported.
- **Display file in spreadsheet program** check box: When checked, the exported traffic file displays automatically in the program specified by the `spreadsheet_prog` preference (environment attribute).

- **Export node names** check box: Exports full hierarchical node names with the traffic data. If unchecked, node names are only exported if IP address information is missing. Including the node names with the export data affects the import of the data. The import engine attempts to match the traffic flow to its endpoints based on node name first, if available. If that lookup fails, then the endpoint IP addresses are used for matching.
- **Export** button: Accepts the current export settings, exports to a spreadsheet program, and closes the dialog box.

Flows > Create Flows

Traffic > Flows > Create Flows: Create IP traffic flows, which represent background traffic, between all or selected nodes in the network. The traffic flows can be a full mesh between all or selected nodes, or can be from one node to the remaining selected nodes (or all nodes in the case of no selection). In the latter case, you can specify the direction of the traffic.

This feature filters out all IP nodes in the network and provides a list of eligible nodes for which you can configure the flows. The traffic can also be specified in bits/packets.

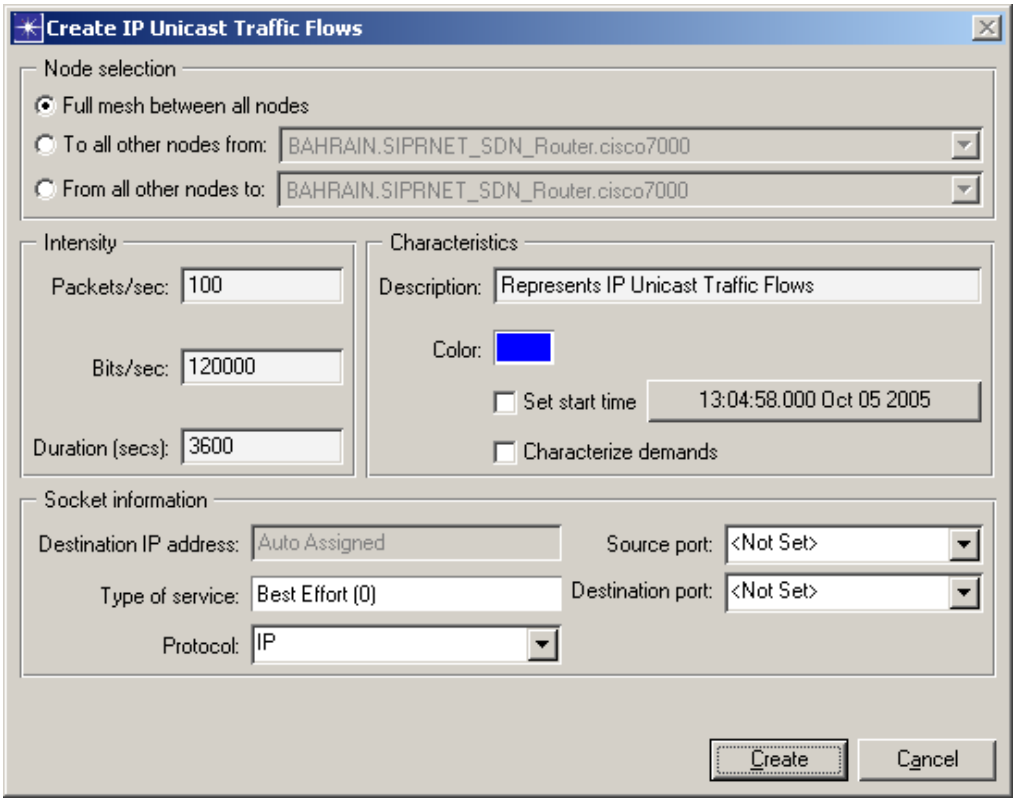


Figure 3-172 Create IP Unicast Traffic Flows dialog box

Device /Link Loads > Import

Traffic > Device/Link Loads > Import > *(option)*: The Import sub-menu contains operations related to importing background traffic loads on devices and links; consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**, for further details. The following options are available from the sub-menu:

Table 3-3 Device/Link Loads > Import Sub-menu Items

Menu Item	Description	Reference
From CA eHealth…	Imports the background traffic load on a device or link from CA eHealth. (MVI module only)	*MVI User's Guide*: *Importing Device/Link Loads*
From MRTG	Imports the background traffic load on a device or link from MRTG traffic file. (MVI module only)	*MVI User's Guide*: *Creating MRTG Link-Load Data*
From InfoVista	Imports the background traffic load on a link from InfoVista reports. (MVI module only)	*MVI User's Guide*: *Creating InfoVista Link-Load Data*
From HP Openview Performance Insight	Imports the background traffic load on a link from HP Openview Performance Insight reports. (MVI module only)	*MVI User's Guide*: *Creating HPOpenview Link-Load Data*
From Text Files	Imports the background traffic load on a device or link from ASCII (.gbu) traffic files.	*MVI User's Guide*: *Importing Device/Link Loads*

Device /Link Loads > Open Import Log

Traffic > Device/Link Loads > Open Import Log: Opens the traffic flow import log, which contains error and diagnostic information generated during previous imports.

Device /Link Loads > Clear Import Log

Traffic > Device/Link Loads > Clear Import Log: Clears the link load import log.

Convert Traffic > Applications to Flows

Traffic > Convert Traffic > Applications to Flows: Automatically convert standard OPNET application and profile configuration parameters into IP flow objects. This process turns the discreet application traffic into background flows for use in simulation or traffic visualization. After creating the flows, the application and profile configuration objects are cleared of all traffic information

in order to prevent duplication of traffic during simulation. It is recommended that a duplicate scenario be created prior to using this feature if you want to keep the application and profile configurations for later use. The following dialog box displays to input the required flow generation parameters.

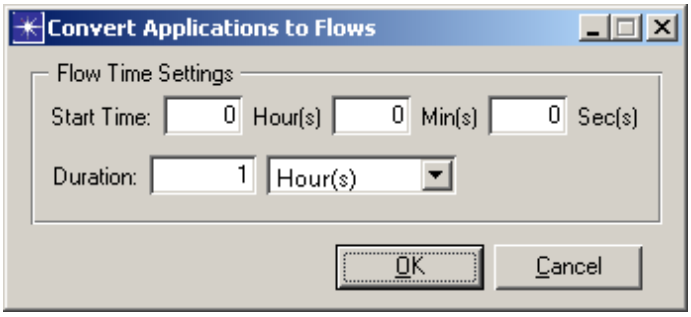


Figure 3-173 Convert Applications to Flows dialog box

As shown above, the required parameters include:

- **Start-Time (Hours, Minutes, Seconds)**: The start-time of the flow objects. This will effectively act to truncate all flow traffic recorded prior to the value provided. The flow's bps and pps values during this time will be set to 0.0. The input values must be integers.
- **Duration (Length, Units)**: The duration of the simulated calculation window. This method corresponds to the Capacity Planning Optimization feature. The input value must be an integer.

Flows will be aggregated for any source/destination pair. Thus, any number of applications that contribute to traffic between a given client and server will be aggregated into two flows with one for each direction of travel.

Generate Traffic Web Report

Traffic > Generate Traffic Web Report: Launch the most recent Traffic web report which details all of the different types of traffic contained in the scenario.

Set Aggregate Traffic Preferences

Traffic > Set Aggregate Traffic Preferences: Specify the type of traffic to visualize—voice, data, and VTC. The default preference is to display all traffic. When you change the traffic flow preference, any rendered flows are updated.

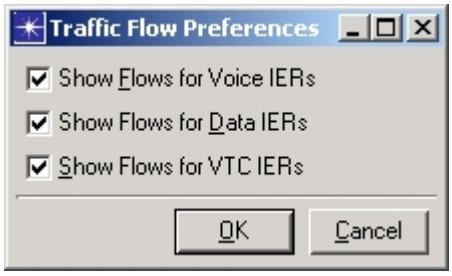


Figure 3-174 Traffic Flow Preferences dialog box

Traffic flows are automatically updated in the following cases:

- Closing the View IERs table, the flows on the producer unit are updated.
- Closing the Specify IERs interface, all displayed flows in the scenario are updated.
- Choosing **File > Refresh IER Text Files**, all displayed flows in the scenario are updated.
- Changing the thresholds or the traffic flow preference, all flows are updated.

Show Aggregate Traffic

Traffic > Show Aggregate Traffic: You can turn on (or off) the visualization for all the units in the scenario, or for individual units. To turn the visualization on or off for individual units, options are provided on their right-click menus.

When you activate the traffic flow visualization, colored path objects are drawn between units to represent the flow of traffic between them. The color of the path objects represents the total load caused by the IERs between the units.

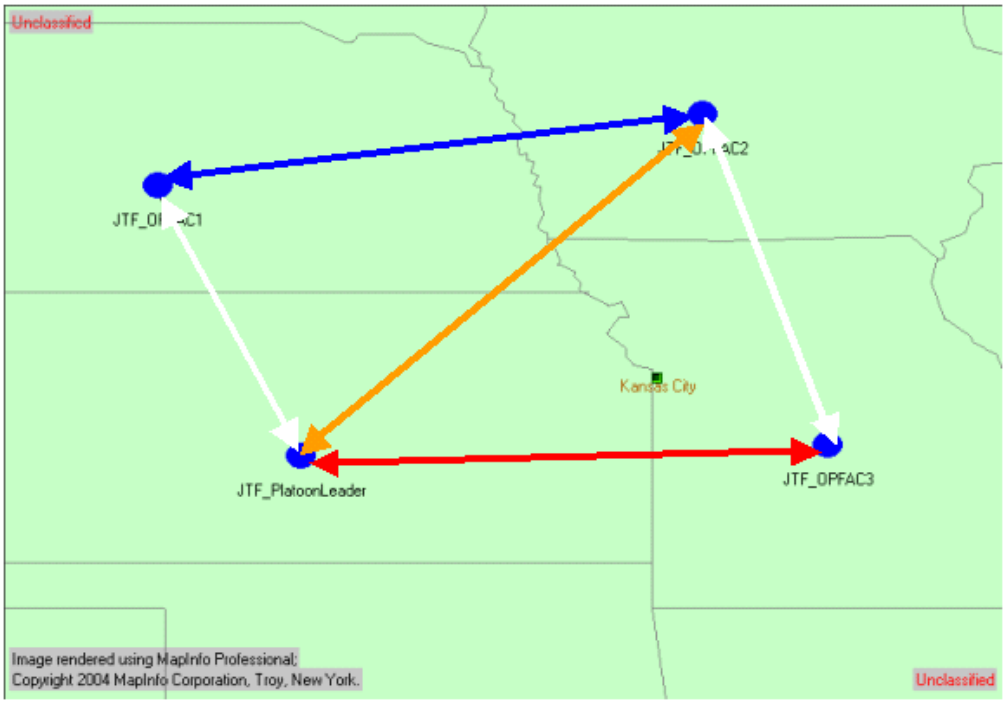
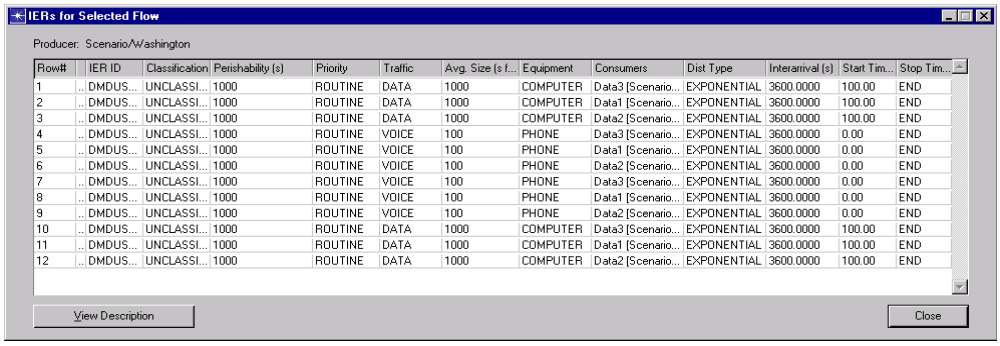


Figure 3-175 Viewing aggregate traffic flows

If the visualization for an OPFAC is turned on, the traffic flow includes IERs produced by that OPFAC. If the visualization for an Organization is turned on, the traffic flow includes IERs produced by all of the OPFACs in that Organization and its sub-Organizations. Turning on the visualization for an Organization and then expanding that Organization or drilling into it will cause the traffic flows to be drawn at the OPFAC level.

An option is provided on the right-click of a traffic flow to view the IERs that constitute that flow. The individual IERs are displayed in a table. They cannot be edited in this table.



Row#	IER ID	Classification	Perishability (s)	Priority	Traffic	Avg. Size (s f...	Equipment	Consumers	Dist Type	Interarrival (s)	Start Tim...	Stop Tim...
1	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data3 [Scenario...	EXPONENTIAL	3600.0000	100.00	END
2	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data1 [Scenario...	EXPONENTIAL	3600.0000	100.00	END
3	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data2 [Scenario...	EXPONENTIAL	3600.0000	100.00	END
4	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data3 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
5	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data1 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
6	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data2 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
7	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data3 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
8	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data1 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
9	DMDUS...	UNCLASSI...	1000	ROUTINE	VOICE	100	PHONE	Data2 [Scenario...	EXPONENTIAL	3600.0000	0.00	END
10	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data3 [Scenario...	EXPONENTIAL	3600.0000	100.00	END
11	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data1 [Scenario...	EXPONENTIAL	3600.0000	100.00	END
12	DMDUS...	UNCLASSI...	1000	ROUTINE	DATA	1000	COMPUTER	Data2 [Scenario...	EXPONENTIAL	3600.0000	100.00	END

Figure 3-176 Traffic that constitutes a flow

The tooltip on a flow will provide high-level details about the traffic flow, such as the number and average size of the voice and data IERs and their loads.

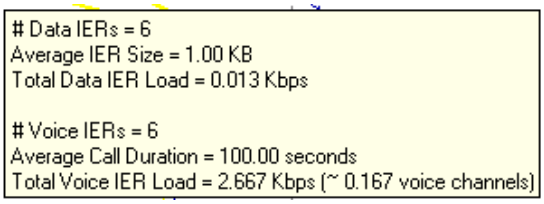


Figure 3-177 Tooltip for a flow

Flows are colored based on their total loads. For data IERs, the load is computed as follows: Traffic load (kbps) = (average IER size (bytes) * 8) / (interarrival time (sec) * 1000)

For voice IERs, the load is computed as follows: Traffic load (kbps) = (average call duration (sec) * channel size (kbps)) / interarrival time (sec)

A default channel size of 16Kbps is assumed when computing the traffic load for voice traffic.

Hide Aggregate Traffic

Traffic > Hide Aggregate Traffic: Hide the traffic flows on all units in the scenario. Options are provided on the right-click menus to hide flows for individual OPFACs or organizations.

Import Node Aliases > From Text File

Traffic > Import Node Aliases > From Text File...: Select a file from which to import node aliases into a scenario.

Export Node Aliases > To Text File

Traffic > Export Node Aliases > To Text File...: Export node aliases from a scenario to a text file. Edit the aliases using a spreadsheet program like Excel, and then import the aliases back into a scenario.

Protocols Menu

The Protocols menu contains operations related to the various protocols supported in the standard and specialized model libraries. The protocols are not documented in this manual; please refer to the OPNET documentation listed for each menu option below (consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**):

Table 3-4 Protocols Menu Items

Menu Item	Description	Reference
Applications	Adds ACE application models to the existing network and configures application traffic.	*Model User's Guide*
Servers	Manages server models.	*Model User's Guide*
Mainframes	Manages mainframe models.	*Model User's Guide*
TCP	Configures TCP.	*Model User's Guide*
IP	Configures IP addresses, dynamic routing protocols, type of service characteristics for conversation pair traffic, autonomous system numbers on routers, and ping traffic.	*Model User's Guide*
Link16	Configures/assigns Time Slot Blocks (TSBs) on terminals in the network.	*Model User's Guide*
IPv6	Configures IPv6 addresses, routing protocols, interface status, and runs readiness assessment and migration planner.	*Model User's Guide*
BGP	Configures BGP start time, autonomous system numbers, and route re-distribution from other protocols into BGP.	*Model User's Guide*
EIGRP	Configures EIGRP start time and route re-distribution from other protocols into EIGRP.	*Model User's Guide*
IGRP	Configures IGRP start time and route re-distribution from other protocols into IGRP.	*Model User's Guide*
IS-IS	Configures interface metrics, interface circuit types, system types, metric styles, and SPF calculation parameters.	*Model User's Guide*

Table 3-4 Protocols Menu Items

Menu Item	Description	Reference
OSPF	Configures interface cost, OSPF start time, areas, and route re-distribution.	*Model User's Guide*
RIP	Configures RIP start time and route re-distribution from other protocols into RIP.	*Model User's Guide*
MPLS	Configures MPLS.	*Model User's Guide*
LDP	Enables/disables LDP protocol status on all/selected routers.	
RSVP	Enables/disables RSVP protocol status across connected interfaces on all/selected links.	*Model User's Guide*
ATM	Configures over-subscription on a per-service class basis for all or selected nodes in the network.	*Model User's Guide*
Frame Relay	Configures PVCs between nodes in the network.	*Model User's Guide*
Ethernet	Configures Ethernet.	*Model User's Guide*
FDDI	Configures FDDI.	*Model User's Guide*
VLAN	Configures virtual LANs (VLANs) for selected nodes and links.	*Model User's Guide*
STP	Applies STP visualization to the current scenario.	
MANET	Configures MANET.	*DSR Model User's Guide*
UMTS	Configures UMTS.	*Model User's Guide*
Wireless LAN	Configures Wireless LAN.	*Model User's Guide*

Capacity Planning Menu

Evaluate **Capacity Planning > Evaluate** or click the **Evaluate Current Network** toolbar button: Evaluate the scenario in its current state during the specified time interval or intervals. This is different from running an optimization in that it evaluates the scenario with the current capacities for links and networks. It does not mutate the capacities of the links and networks. It routes the traffic through the links and broadcast networks, and provides a summary of the scenario's current condition.

1) After choosing the **Evaluate** menu item or clicking the **Evaluate Current Network** toolbar button, the Evaluation Settings dialog box displays.

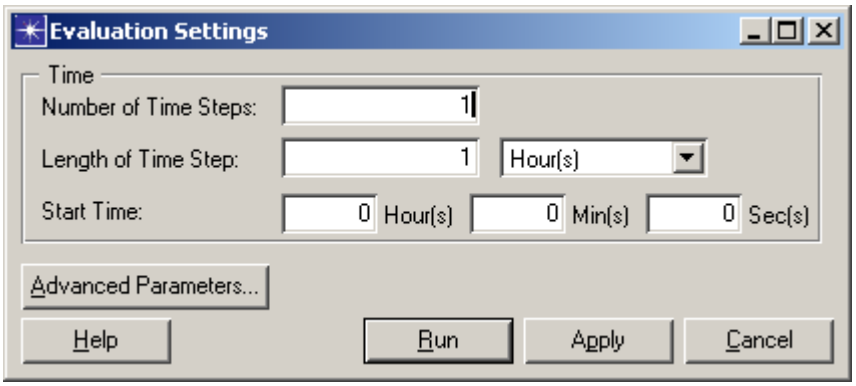


Figure 3-178 Evaluation Settings dialog box

The Evaluation Settings dialog box allows you to set the number of time steps, length of each time step, and the start time. For example, if you know that your peak usage occurs from 9 A.M. to noon, you could evaluate that time period by setting number of time steps to 3, length of time step to 1 hour, and start time to 9 hours 0 minutes 0 seconds.

You can set the following parameters for the evaluation:

a) **Number of Time Steps**: A Capacity Planning Evaluation can be done for multiple time steps by setting this value to an integer greater than one. Reports will be created for the evaluation at each time step as well as reports reflecting results over the entire time of evaluation. Must be an integer greater than zero.

b) **Length and Units of Time Steps**: The length and units of the time steps used in the evaluation. The value selected must be an integer greater than zero. Units can be seconds, minutes, hours, days or weeks.

c) **Start Time**: The start time used for the evaluation in hours, minutes and seconds. All values must be integers greater than or equal to 0.

d) **Advanced Parameters**: Opens a dialog box for more parameters for the evaluation and reporting of the evaluation.

e) Click the **Apply** button to save any changes you made to the settings.

2) If you click the **Advanced Parameters** button to access advanced evaluation settings, the Evaluation Settings Advanced dialog box displays.

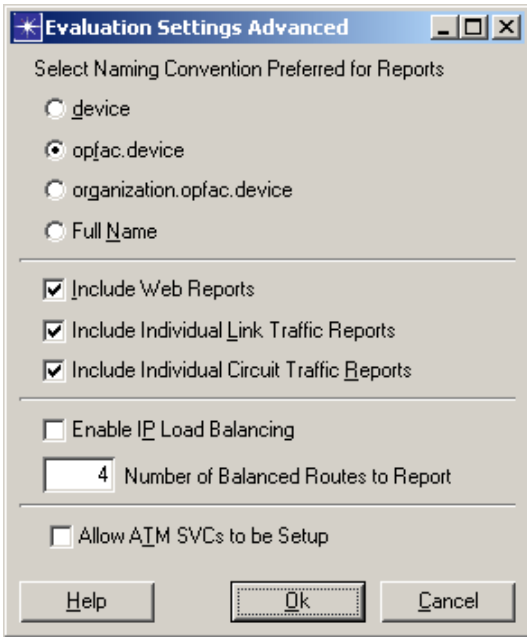


Figure 3-179 Evaluation Settings Advanced dialog box

You can set the following advanced parameters for the evaluation:

a) **Naming Convention**: The names of all the devices and links that are in the report will use the naming convention selected here.

b) **Include Web Reports**: If this option is selected, the link and circuit traffic reports are also selected. Unchecking this will disable the options for the individual reports.

c) **Include Individual Link Traffic Reports**: If this option is selected, individual reports will be included in the web report for each link in the scenario. The reports will include all the traffic going directly over the link as well as any circuits that are going over the link. The reports will take longer to generate if this option is selected.

d) **Include Individual Circuit Traffic Reports**: If this option is selected, individual reports will be included in the web report for each routable circuit. The reports will include all the traffic using the circuit. The reports may take longer to generate if this option is selected.

e) **Enable IP Load Balancing**: If this checkbox is selected, IP load balancing will be done during the evaluation so that traffic will use multiple equal hop length paths.

f) **Number of Balanced Routes to Report**: If IP load balancing is selected, then the number of balanced routes to report can be selected. The entry must be an integer greater than 0. This number of route reports will be reported in the web reports

g) **Allow ATM SVCs to be Setup**: If this checkbox is selected, ATM SVCs will be setup during the evaluation when applicable.

h) Click the **OK** button to save any changes you made to the settings.

3) Once everything is set, click the **Run** button in the Evaluation Settings dialog box to start the evaluation.

The evaluation engine evaluates the specified number of time steps using only the traffic, circuits, devices, and links active at each time step. For traffic that is active during only a portion of the time step, the load will be prorated according to the fraction of the time step the traffic is active. Circuits, devices, and links will be considered active only if they are active at the end of the time period.

4) Once the evaluation is complete, the evaluation web reports launch automatically.

The web reports include the following reports for the overall evaluation: Executive Summary, Overall Peak Results, and Overall Average Results. Additionally there are a group of reports which are generated for each time step: Traffic Report, Link Utilization Report, Residual Bandwidth Report, Circuit Switch Link Report, Transmission Link Report, Data Link Report, ATM & CellXpress TVC Report, Frame Relay Report, TSSP Circuit Report, and Multiplexer Circuit Report. There are also Route Reports for each routed circuit and traffic IER, and Unroutable Reports for each unroutable circuit and traffic IER. Each Individual Link also has a group of tables that show the circuit and traffic IERs which flow over it specifically.

If a load object (IER, IP Flow, Promina circuit, etc.) is unroutable, the Unroutable Report will list reasons why the load could not be routed. The following table lists all possible failure messages and their associated suggestions.:

Table 3-5 Route Failure Messages and Suggestions

Message	Suggestion
There was no path at the Promina layer.	Make sure all of the necessary links between Promina devices have been added to the scenario. Also, check for lower-level CellXpress circuit failures that could prevent Promina devices from communicating.
There was no path at the ATM layer.	Make sure all of the necessary links between ATM devices have been added to the scenario.
There was no path at the Frame Relay layer.	Make sure all of the necessary links between Frame Relay devices have been added to the scenario.
There was no path at the Voice layer.	Make sure all of the necessary links between Voice devices have been added to the scenario. Also, check for lower-level Promina circuit failures that could prevent Voice devices from communicating.

Table 3-5 Route Failure Messages and Suggestions

Message	Suggestion
There was no path at the IP layer.	Make sure all of the necessary links between routers have been added to the scenario. Also, check for lower-level circuit failures that could prevent IP routers from communicating.
An end device that can support this IER could not be located in the producer or consumer OPFAC.	Make sure the source and destination OPFACs both contain a non-failed end device that supports the IERs traffic type.
There was a mismatch in the number of layer 3 encryptors and decryptors in this route.	Make sure each layer 3 encryptor (NES, IP_ATM_TACLANE, etc.) has a corresponding decryptor.
Selected path for Promina circuit is invalid.	Make sure all necessary links between Prominas in the Selected Path have been added to the scenario. Also, check for lower-level CellXpress circuit failures that could prevent Promina devices from communicating.
Selected path for Promina circuit does not have sufficient bandwidth.	Increase the capacity of the links between Prominas in the selected path, or add additional links between the Prominas.
<xxx> circuit does not have enough bandwidth.	Increase the reserved bandwidth of the circuit.
<link_name> does not have enough bandwidth.	Increase the capacity of the link or add an additional link between the endpoints.

Capacity Optimization

Capacity Planning > Capacity Optimization > (*option*): Run an optimization for capacity. Given a network and the traffic, the optimization process provides suggestions for optimal link and network capacities. If a link is over-utilized, optimization will suggest an alternate capacity that will support the IERs and keep the cost of the link down to the required minimum. If a link is under-utilized, optimization will decrease the capacity for that link. Capacity optimization options are discussed in the sections below.

Application profiles are converted to IERs when a user evaluates or optimizes a scenario for the first time. Once the profiles have been converted to IERs, they are retained as IERs. The conversion process is not initiated again unless you add more application profiles.

Capacity Optimization > Start Optimization

Capacity Planning > Capacity Optimization > Start Optimization: Specify parameters that control how optimal solutions for the scenario are computed.

1) Select **Start Optimization** or click the **Start Capacity Optimization** toolbar button: The Capacity Optimization Settings dialog box displays.

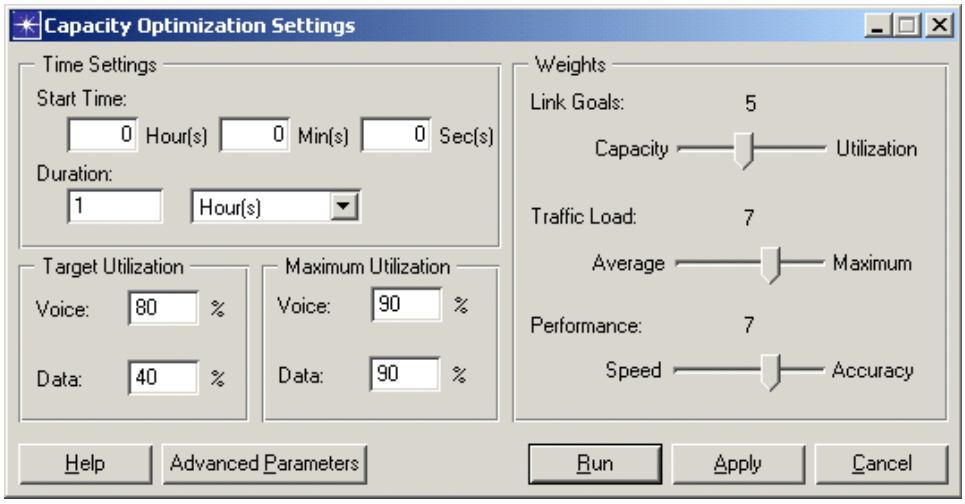


Figure 3-180 Capacity Optimization Settings dialog box

These options let you set optimization parameters:

a) **Time Settings**: Allows you to specify the start time and the duration for the optimization process. Only traffic that is active during the specified time frame is considered while optimizing.

b) **Voice** and **Data Target Utilization**: Specifies the average percentage utilization we are trying to achieve on the voice and data portions of the trunks.

c) **Maximum Utilization**: Specifies the maximum percentage utilization we are trying to achieve on the voice and data portions of the trunks.

d) Link Goals, Traffic Load, and Performance Weights:

Link Goals: This value allows you to optimize for different criteria. If you want to minimize the capacity, move the slider to the left (towards Capacity). This will produce a final solution in which the capacity is reduced. If the target utilization is more important to you, move the slider to the right (towards Utilization). This will produce a final solution in which the link utilization values are closer to the target.

Traffic Load: This value allows you to control the load caused by the traffic on the network. If you set this value to 1, the engine uses the mean values from the distribution. As you increase this value, the engine uses values from the tail of the distribution. If the distribution type is "Constant," this parameter does not make a difference.

Performance – This value allows you to specify how fast or how accurate you want the optimization solution to be. If you set this slider at 1, the engine converges on a solution quickly at the cost of accuracy. If you set this slider at 10, the engine tries more configurations and the solution may be better, but it will take longer.

2) Click the **Advanced Parameters** button on the Capacity Optimization Settings dialog box. The Advanced Parameters dialog box displays.

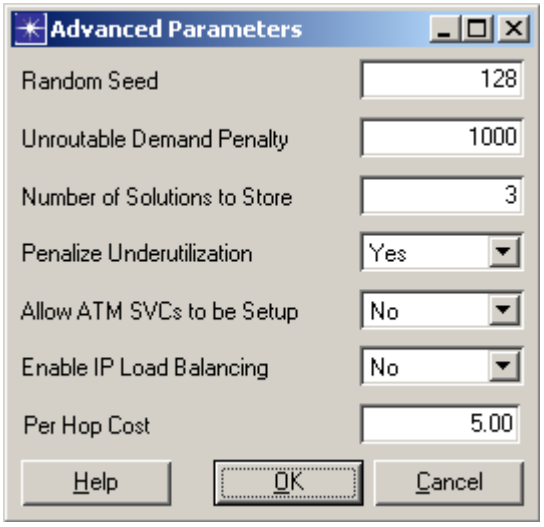


Figure 3-181 Advanced Parameters dialog box

Configure advanced parameters, if needed, or use the default values:

a) **Random Seed**: This is the seed value used by the random number generators in the optimization engine.

b) **Unroutable Demand Penalty**: This is the penalty that will be added to the optimization cost for each IER that is unroutable in this configuration.

c) **Number of Solutions to Store**: The optimization engine stores the configurations corresponding to the 'n' best solutions and you can switch back and forth between these solutions. This value specifies the number of such solutions to store.

d) **Penalize Underutilization**: The utilization cost depends on the difference between actual link utilization and target utilization, regardless of whether link utilization is over or under target utilization.

e) **Allow ATM SVCs to be Setup**: By not allowing ATM SVCs to be setup, you are forcing the Scenario Builder to always look for PVCs between gateway ATM devices. In this case, the route taken by the IER in the Scenario Builder will be the same as that in the simulation.

f) **Enable IP Load Balancing**: Allows you to disable or enable IP load balancing to be done during the evaluation so that traffic will use multiple equal hop length paths.

g) **Per Hop Cost**: This is a fixed cost that is computed for each load based on the number of device-to-device hops in the route. If there are multiple solutions that have the same objective cost, adding this cost will ensure that the solution in which the loads have the shortest path is chosen as the best solution.

3) Click **Run** to begin the optimization.

JCSS computes the routes for the traffic and then starts the optimization. After the optimization completes, a dialog box displays a summary of the optimization process:

a) **Original Average Values** - Displays the values before the optimization process was started.

b) **Final Average Values** - Displays the values at the end of the optimization.

c) **Summary Table** - Displays a list of links and broadcast networks for which the optimization engine suggested changes. Click on a row in this table to display the Attributes dialog box for that link or network. This Attribute dialog box shows the original values and the current values suggested by the engine. You can accept the changes suggested by the engine or selectively reject the changes for some links while retaining the changes for the others by setting the Status field to “Reject.”

Capacity Optimization > Save Optimization As

Capacity Planning > Capacity Optimization > Save Optimization As: Save the results of an optimization as part of the scenario, so that it is available for future comparisons.

During an optimization, JCSS might suggest changes to the link and network bandwidths or to the inter-arrival times of the IERs. If you save the project after an optimization, these values become the actual values and you will not be able the previous values.

To prevent losing the original values accidentally, you will be prompted by a dialog to save the scenario under a different name. If you choose this option, a copy of the scenario is made under the current project and the values in the original scenario are left unchanged.

1) Enter a name for the optimization and click **OK**. The name is used to identify the optimization in the Manage Results interface.

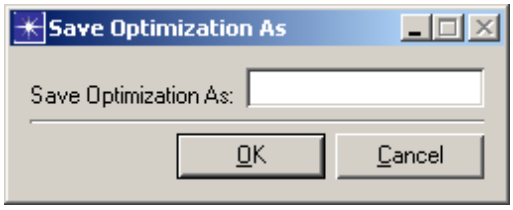


Figure 3-182 Save Optimization As dialog box

Capacity Optimization > Manage Optimization Results

Capacity Planning > Capacity Optimization > Manage Optimization Results: Access the Manage Capacity Optimization Results to delete optimization results from the scenario's state, compare optimizations, and make one of the optimization solutions the current solution.

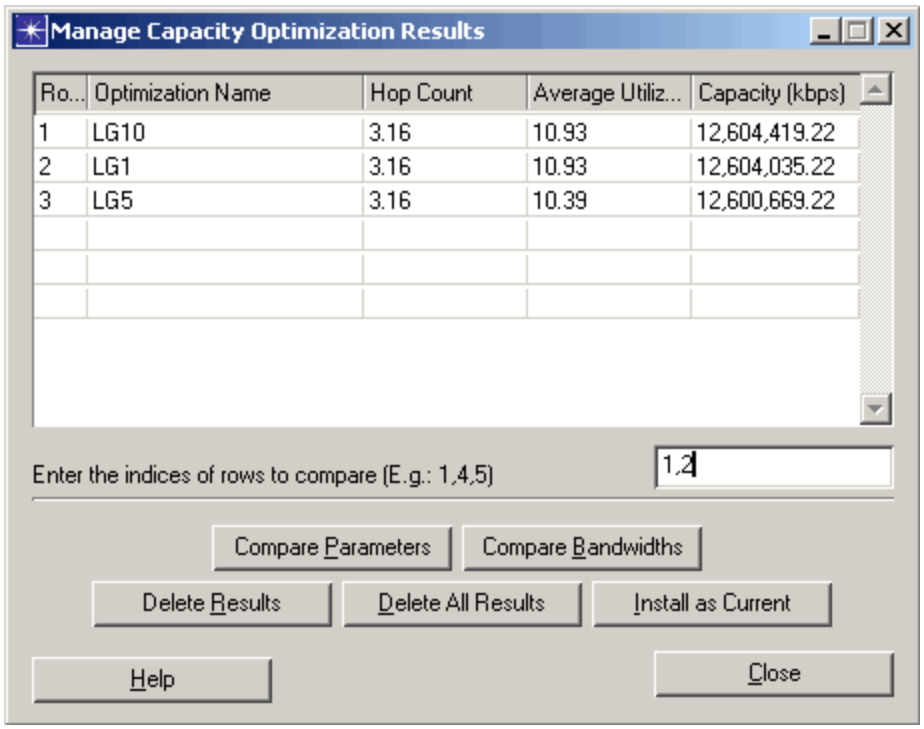


Figure 3-183 Manage Capacity Optimization Results dialog box

Select the row number(s) for the optimizations to compare, delete, or install as current solution, and then click the corresponding button.

Capacity Optimization > Examine Top Solutions

Capacity Planning > Capacity Optimization > Examine Top Solutions: Examine top solutions returned by the optimization. At the end of optimization, the engine returns the best solution that it found so far. Sometimes the best solution might have suggestions that you do not agree with. In such cases, review a few of the top solutions and use the one that you think is the best. You can specify the number of top solutions that the engine must store. At the end of optimization, you can review the top solutions and pick the one that suits your needs.

You can specify the number of solutions to store using the Advanced Optimization Parameters.

1) The top most solution is highlighted in yellow. If you prefer a different solution, you can click on the drop-down menu at the top of the dialog box and change the currently visible solution.

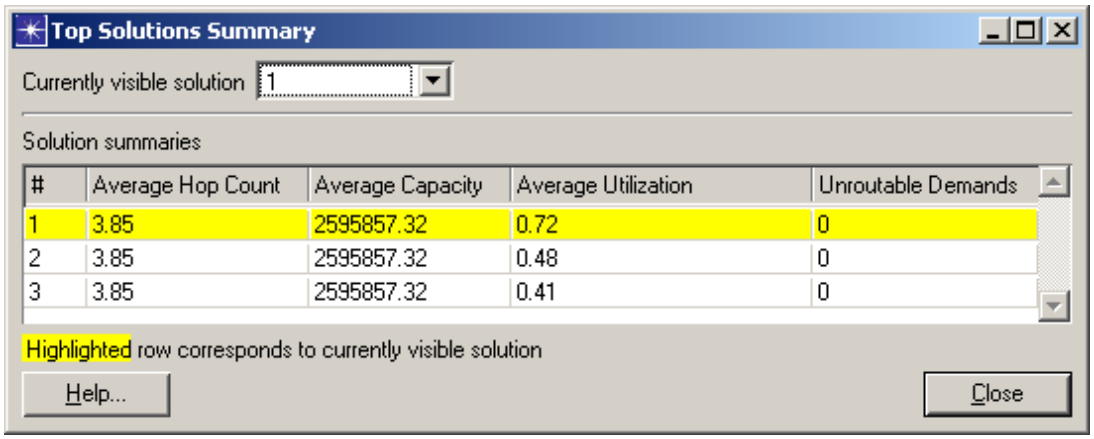


Figure 3-184 Examining the Top Solutions

Capacity Optimization > Restore Original Capacities

Capacity Planning > Capacity Optimization > Restore Original Capacities: Restore the scenario's original capacities after an optimization is run.

Reports

Capacity Planning > Reports > (*option*): Open the listed reports.

Reports > View Web Report

Capacity Planning > Reports > View Web Report: Launch the most recent Capacity Planner web report for this scenario.

Reports > Settings

Capacity Planning > Reports > Settings: Web report settings are used in the web reports produced after a Capacity Planning Evaluation or Optimization.

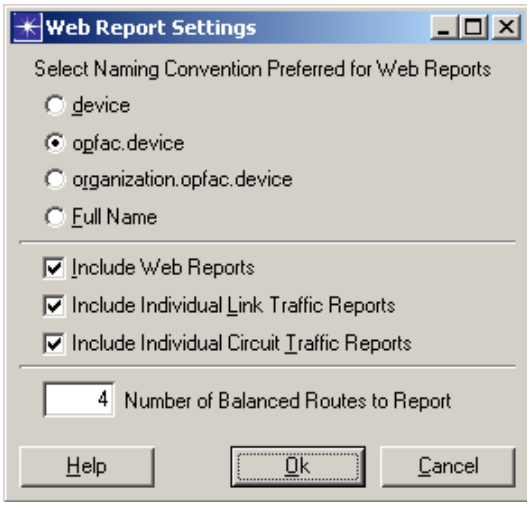


Figure 3-185 Web Report Settings dialog box

These options let you set report parameters:

- **Naming Convention**: The names of all the devices and links that are in the report will use the naming convention selected here.
- **Include Web Reports**: If this option is selected, the link and circuit traffic reports are also selected. Unchecking this will disable the options for the individual reports.
- **Include Individual Link Traffic Reports**: If this option is selected, individual reports will be included in the web report for each link in the scenario. The reports will include all the traffic going directly over the link as well as any circuits that are going over the link. The reports will take longer to generate if this option is selected.
- **Include Individual Circuit Traffic Reports**: If this option is selected, individual reports will be included in the web report for each routable circuit. The reports will include all the traffic using the circuit. The reports may take longer to generate if this option is selected.
- **Number of Balanced Routes to Report**: If IP load balancing is selected, then the number of balanced routes to report can be selected. The entry must be an integer greater than 0. This number of route reports will be reported in the web reports.

NetDoctor

NetDoctor is a powerful rules-based engine that proactively identifies incorrect device configurations, including policy violations and inefficiencies. NetDoctor exposes hidden problems that can be difficult to discover due to the sheer volume of configuration information distributed throughout the network.

NetDoctor rules are normally run against model attributes, which are set from device configurations that you can import from a production network. Also, some NetDoctor rules use simulation data to identify inconsistencies in the operation or performance of the network.

You can run all of the NetDoctor rules on a JCSS network, except for the ones that have JCSS devices in them, usually on the networks that are created from Device Configuration Import (DCI). For the networks with JCSS devices in them, you can run only rules that are devised for individual devices or a device group.

Note—A separate NetDoctor license is required in order to run NetDoctor in JCSS. A JCSS license does not include a NetDoctor license.

The table below outlines the main menu options under NetDoctor:

Table 3-6 NetDoctor Menu Items

Menu Item	Description
Configure/Run NetDoctor...	Configure, save, and run NetDoctor report templates.
Run NetDoctor...	Run NetDoctor using an existing report template.
Auto-Generate Report Template...	Create a NetDoctor Report Template.
View Recent NetDoctor Reports...	Choose a recently run NetDoctor report and open it in the appropriate viewer.
Configure/Run Trending	Configure and generate NetDoctor Trending reports. Note: NetDoctor trending is project-specific. You cannot trend NetDoctor data using scenarios from more than one project.
Open NetDoctor Log	Display the NetDoctor Log file contents.
Suppress Messages	Choose not to display NetDoctor messages.
Rule Development...	Create new or edit existing NetDoctor rules.
Options	Configure NetDoctor operation and report generation settings.

Please refer to the *NetDoctor User's Guide* (Chapter 3, Using NetDoctor) under the ITGuru product documentation for details on using NetDoctor.

DES (Discrete Event Simulation) Menu

Configuration OPFAC

Every time a new scenario is created (or an older scenario is converted), a Configuration OPFAC will automatically be placed in the top-level subnet. Simulations cannot be run if this object is not present in the scenario, therefore, the Configuration OPFAC cannot be deleted or duplicated. The Configuration OPFAC is an actual OPFAC, and can be accessed via the treeview or the workspace (the default location for the OPFAC is 60, -150). The Configuration OPFAC contains (at least) these four utility nodes: Failure/Recovery, Wireless Failure/Recovery, Standard NETWARS Node, and QoS.

Failure/Recovery Node

JCSS failure/recovery functionality allows you to specify a time interval or period of time for which devices of an OPFAC are inoperable. You can also specifying failure and recovery times for organizations, links, individual devices, and OPFACs. This feature simulates equipment failures or loss of forces. Failure/recovery times are specified on the individual objects listed above, however, once specified, the failure/recovery times for all objects in the scenario reside on the Failure/Recovery node. This node can be deleted.

1) From the Scenario Builder, right-click on an organization, link, individual device, or OPFAC to open the shortcut menu.

2) On the shortcut menu, select **Edit NETWARS Attributes**. The Attributes dialog box displays.

3) Click the **Failure/Recovery** button at the bottom of the dialog box. The Failure/Recovery Times dialog box displays.

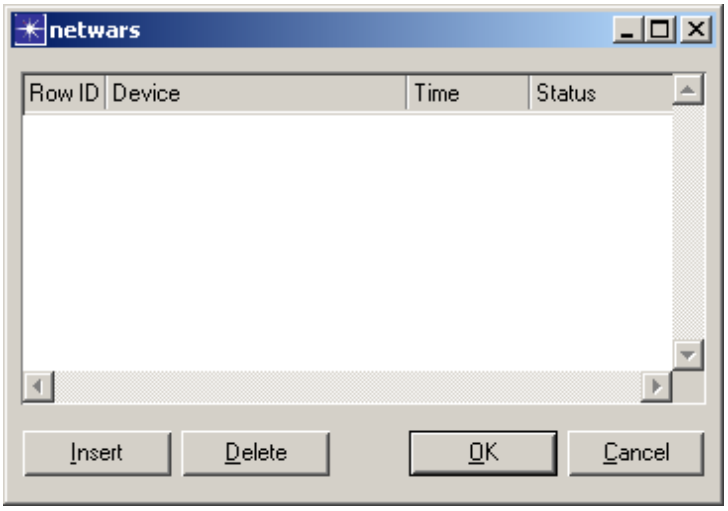


Figure 3-186 Failure/Recovery Times dialog box

4) Click **Insert**. A failure/recovery time with default values displays in row 1. Edit the **Device**, **Time**, and **Status** fields as needed. Repeat as needed to create all failure/recovery times for the selected object.

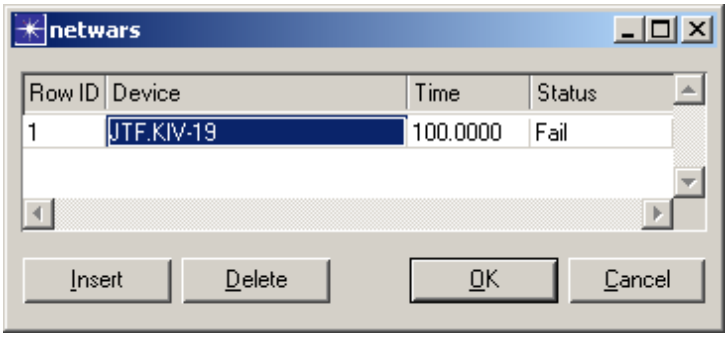


Figure 3-187 Inserting a Fail Time

5) Click **OK** to close the Failure/Recovery Times dialog box, and then click **OK** again to close the Attributes dialog box. Repeat steps 1–5 to create failure/recovery times for other scenario objects.

6) To view all failure/recovery times for the scenario, right-click on the Failure/Recovery node in the Configuration OPFAC and select **Edit Attributes**.

7) Next, click in the **Value** cell of the **Link Failure/Recovery Specification** attribute and select **(…)** to view link failure/recovery times, or click the **Value** cell of the **Node Failure/Recovery Specification** attribute and select **(…)** to view node failure/recovery times.

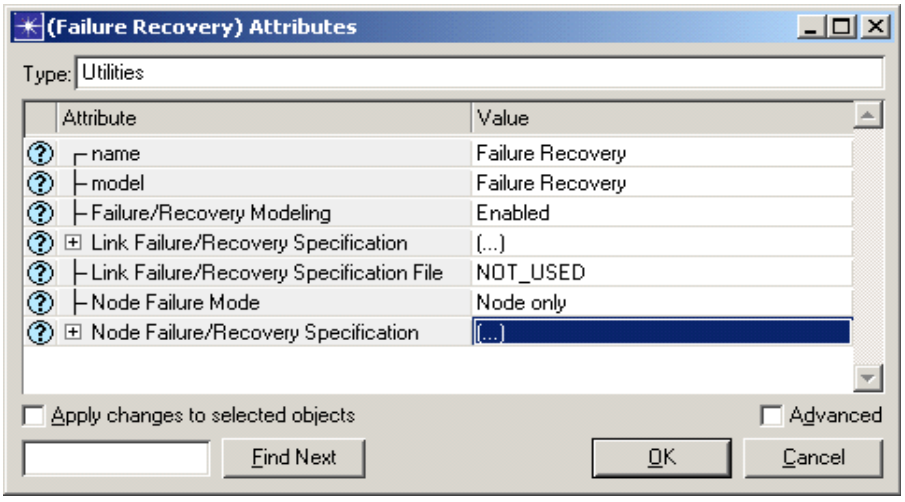


Figure 3-188 Failure/Recovery node attributes

Wireless Failure/Recovery Node

The Wireless Failure/Recovery node works in the same manner as the Failure/Recovery node explained above, except it is only used for wireless networks (i.e., broadcast networks). This node can be deleted.

Standard NETWARS Node

The Standard NETWARS node must exist to run a simulation; therefore, it cannot be deleted. Before a simulation is executed, certain parameters that influence the behavior of the simulation can be configured from this node. These parameters, which specify the behavior of traffic during simulation, are configured through the Priority Table, Decision Table, and System Element Table. These tables are accessible via the Standard NETWARS node shortcut menu by selecting **Edit Attributes**.

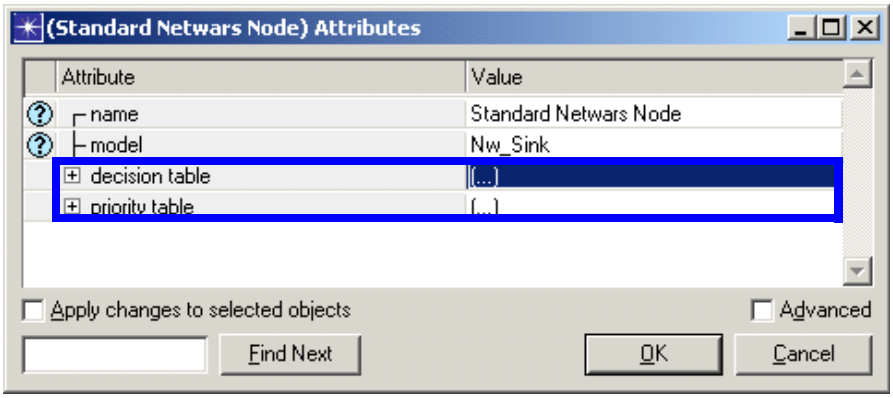


Figure 3-189 Accessing the Decision Table and Priority Table

Priority Table This table can be used to specify how the simulation deals with failed IER transmissions due to busy sending and receiving devices. For each precedence type, you can specify the number of retries and the wait time between retries.

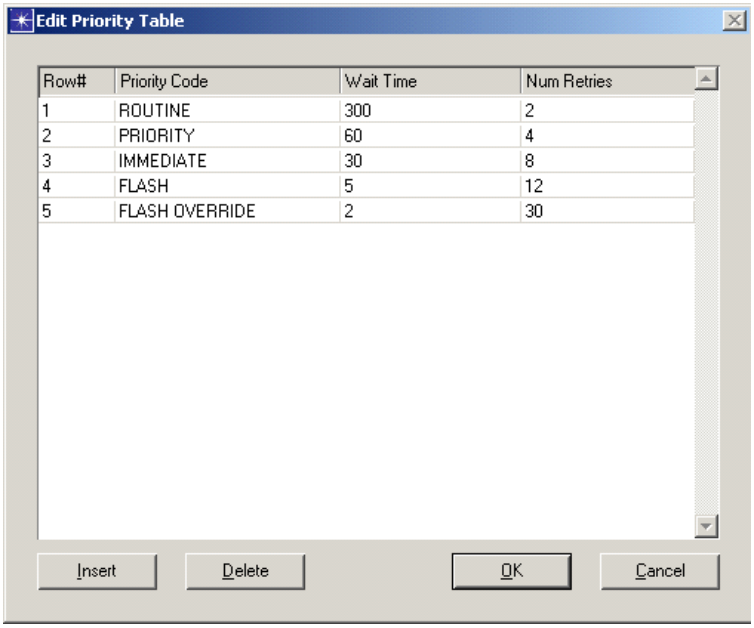


Figure 3-190 Priority Table

From this table we can see that if a Flash IER cannot be fired at the specified time, the simulation will wait for 5 seconds and try to send it again. For Flash, the simulation will try a maximum of 12 times before failing the IERs.

Decision Table and System Element Table When creating IERs, you specify the producing and consuming OPFACs. You do not have to specify the device within the OPFAC that produces or consumes the traffic, but the simulation needs to know the device within the OPFAC that handles the traffic.

This device selection process can be left to the simulation with the help of the Decision Table and the System Element (SE) Table. These tables have a series of entries tying the type of traffic, the classification, and the equipment type to be used for that traffic to the actual device within an OPFAC.

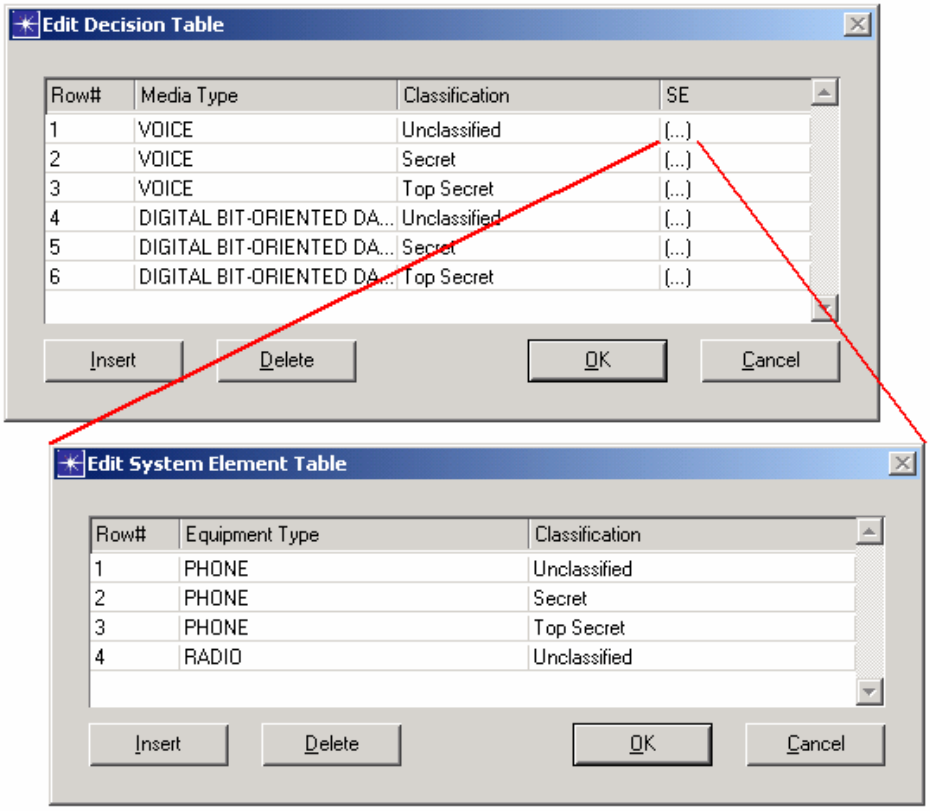


Figure 3-191 Decision Table and System Element Table

When it is time for the simulation to send traffic, it looks for a matching entry in the Decision Table. If an entry is found, it looks in the SE table corresponding to that entry for a matching equipment type. Using the equipment type and classification obtained from these two tables, the simulation looks for a device within the producer and consumer OPFACs that can produce and consume the traffic.

The Priority Table and Decision Table can also be configured in the Configuration OPFAC utility.

Quality of Service (QoS) Node

The QoS node is used to define custom QoS schemes and apply quality-of-service constraints to your network. This node can be deleted.

Before a simulation is executed, certain parameters that influence the behavior of the simulation can be configured.

Choose Individual Statistics

DES > Choose Individual Statistics...: Choose statistics on devices, inter-OPFAC, and intra-OPFAC links to be collected during simulation.

1) Select the object in the workspace for which the statistics will be collected (includes all devices, the OE, inter-OPFAC, and intra-OPFAC links).

2) Select **DES > Choose Individual Statistics** or select **Choose Individual Statistics** from the shortcut menu. The Choose Results dialog box displays.

3) Select statistics to be collected during a simulation by clicking the checkboxes next to the desired statistics.

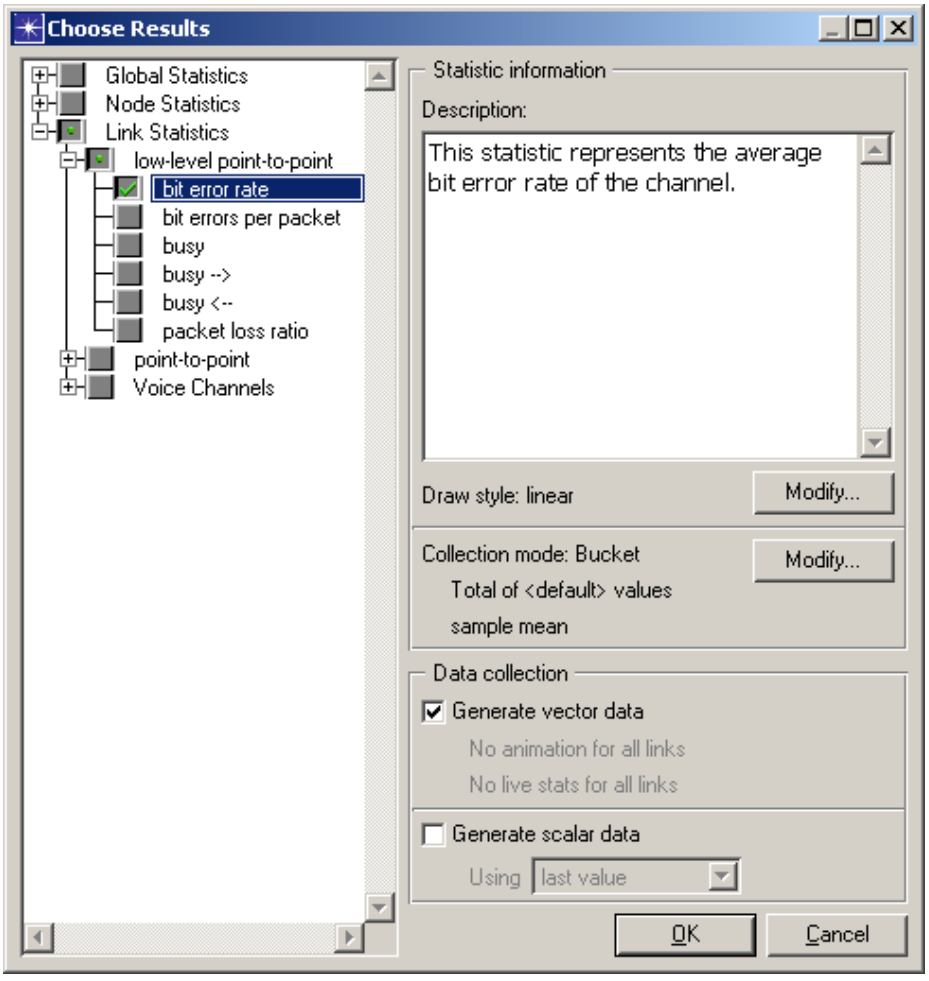


Figure 3-192 Choose Results dialog box

4) Right-click on the individual statistic to make changes to the collection mode or draw style.

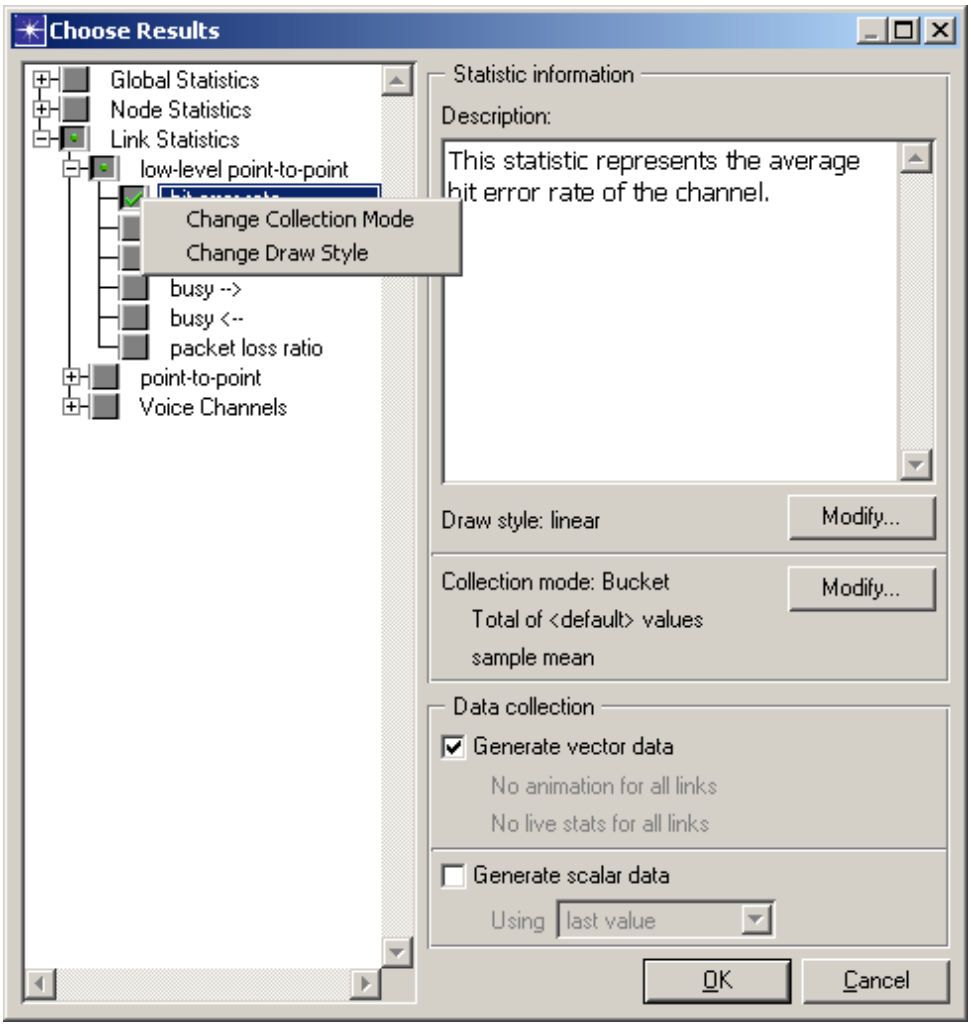


Figure 3-193 Change Collection Mode or Draw Style

Choose Statistics (Advanced)

DES > Choose Statistics (Advanced): In the Probe Model window, choose advanced statistics on devices, inter-OPFAC, and intra-OPFAC links to be collected during simulation.

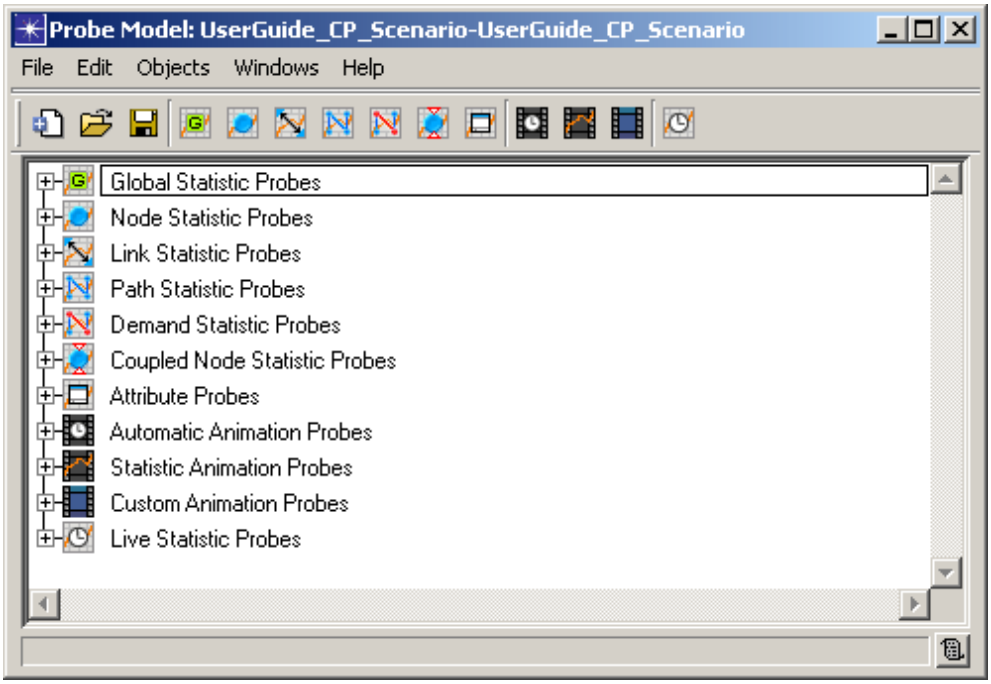


Figure 3-194 Probe Model Window

Record Packet Flow 2D Animation for Subnet and Record Node Movement 2D Animation for Subnet

DES > Record Packet Flow 2D Animation for Subnet and **DES > Record Node Movement 2D Animation for Subnet**: Turn on or off the collection of animation data from the subnetwork. This data allows you to view the packet flow and (with the Wireless Module) node movement that occurs during a simulation.

Expert Service Prediction > Define Service Level Agreement

DES > Expert Service Prediction > Define Service Level Agreement: Defines the thresholds for a service level agreement (SLA.)

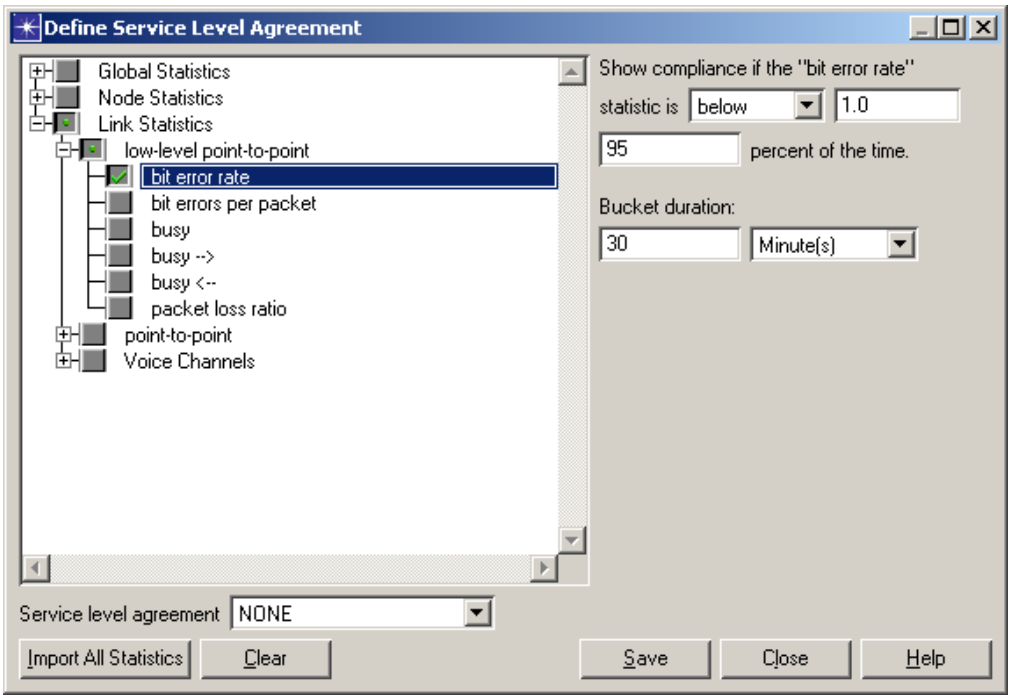


Figure 3-195 Define Service Level Agreement dialog box

- **Show Compliance** pull-down menu/fields: Displays a statistic's compliance with an SLA. An SLA can include thresholds for one or more statistics. To view a statistic's compliance, select it in the Statistic Browser. You must set each statistic's threshold individually; if you select multiple statistics and set a threshold, the threshold applies only to the last statistic you selected.
- **Bucket Duration** field/pull-down menu: Sets the duration for buckets for the simulation run. If a statistic has a bucket-capture mode set to time average, sum/time, or count, the SLA measures compliance for the entire simulation run. For all other statistics, it divides the total simulation run into buckets (set by the Bucket Duration field) and measures compliance for each bucket.
- **Service Level Agreement** pull-down menu: Allows edits to existing service level agreements.
- **Import All Statistics** button: Lists statistics for all models listed in the mod_dirs directory.
- **Clear** button: Deselects all statistics, and resets the statistics tree.
- **Save** button: Opens the Save As dialog box, which allows you to save the thresholds defined for the scenario.
- **Close** button: Opens the Confirm dialog box, which prompts you to save your changes before closing.

Configure/Run Discrete Event Simulation

DES > Configure/Run Discrete Event Simulation or click the **Configure/Run Discrete Simulation** button: Set the values of simulation attributes and control how a discrete event simulation (DES) runs in the current (or specified) scenario. The Configure/Run DES dialog box displays in simple mode. This dialog box has a set of commonly used dialog box controls, and two tabbed windows to configure DES inputs (Global attributes) and outputs (Reports.)

1) Set the duration of the simulation in the **Duration** field. The units are specified by the pull-down menu next to this field. This sets the duration simulation preference.

2) Set the number of values collected for each statistic in the **Values per statistic** field. This sets the num_collect_values simulation preference.

3) Set the simulation attributes in the Global attributes window to the desired values. The Attribute table displays all simulation attributes declared by models involved in the scenario to be simulated and the values assigned to each attribute. Click on a question mark icon to display information about the corresponding attribute. Click on an Attribute Value to change it.

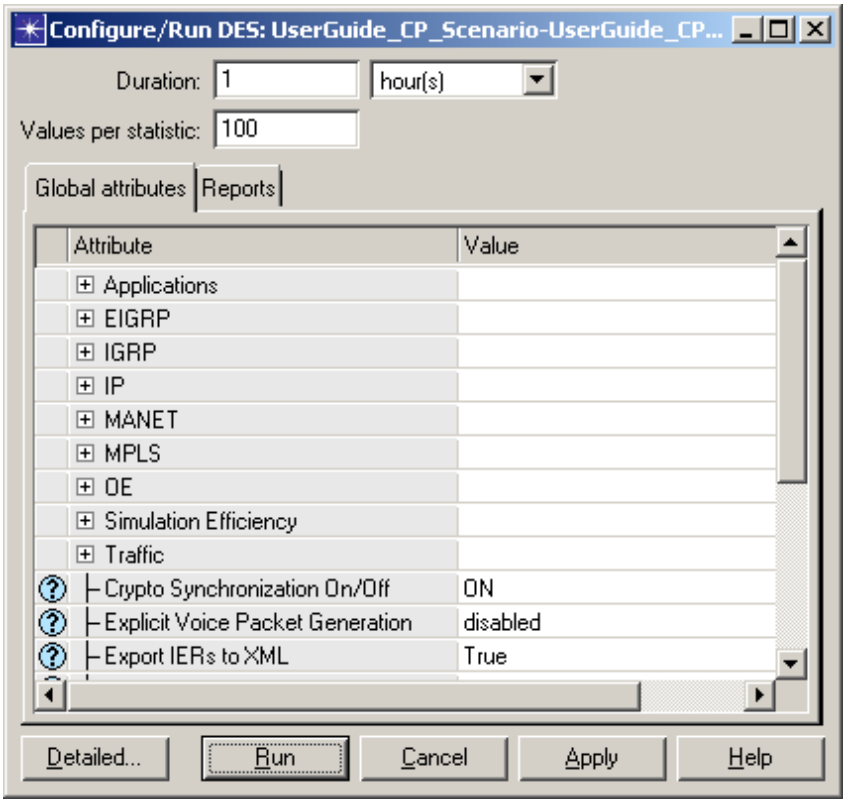


Figure 3-196 Configure/Run DES dialog box

For details on the attributes that can be configured on the Global attributes page, refer to the *OPNET Editors Reference Manual* (consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**).

4) Click the Reports tab to select Statistics and Service Level Agreement (SLA) reports for the simulation to collect. Reports are predefined sets of statistic probes.

   - **Record the following date and time in results** label—Displays the date and time associated with the network model, if any.
   - **Generate web report for simulation results** checkbox—When selected, simulation results are included in a report viewed from a web browser.
   - **Minimum number of decimal places for statistics in output reports** counter—Sets the number of decimal places to use in number formatting. If you expect values to differ in minute amounts, use a large number of decimal places.
   - **Number of entries displayed in 'Top N' reports** counter—Sets the value of 'N'.
   - **Statistics reports** pane—Specifies the statistics reports to generate at the end of the simulation. Each selected value is added to the reports simulation preference.
   - **Define Statistics Report** button—Opens a dialog box used to define new statistics reports.
   - **SLAs** pane—Specifies the SLA reports to generate at the end of the simulation. Each selected value is added to the sla_reports simulation preference.
   - **Define SLA Report** button—Opens a dialog box used to define new SLA reports.

5) Click the **Detailed** button if you want to open the Configure/Run DES dialog box in “detailed” mode. Detailed mode lets you configure all of the parameters of DES, some of which are not available in the simple mode. This dialog box contains a treeview of information categories for configuring different aspects of the simulation.

6) Click **Cancel** to close the dialog box without saving any changed settings, or click **Apply** to save the current settings and keep the dialog box open, or click **Run** to save the current settings, close the dialog box, and run the simulation immediately. The simulation runs for set duration, displaying frequent progress updates in the viewing window until completion.

Configure/Run Discrete Event Simulation (Advanced)

DES > Configure/Run Discrete Event Simulation (Advanced): Set the values of advanced simulation attributes and control how a simulation runs.

Run Discrete Event Simulation

DES > Run Discrete Event Simulation: Run a simulation of the current scenario. The dialog box that appears displays the status of the simulation as it runs and allows you to pause or stop the simulation, if desired.

1) Choose **DES > Configure/Run Discrete Event Simulation** to open the Simulation Set dialog box.

2) If desired, set the simulation attributes.

3) Click the **Run** toolbar button. The Simulation Execution dialog box opens.

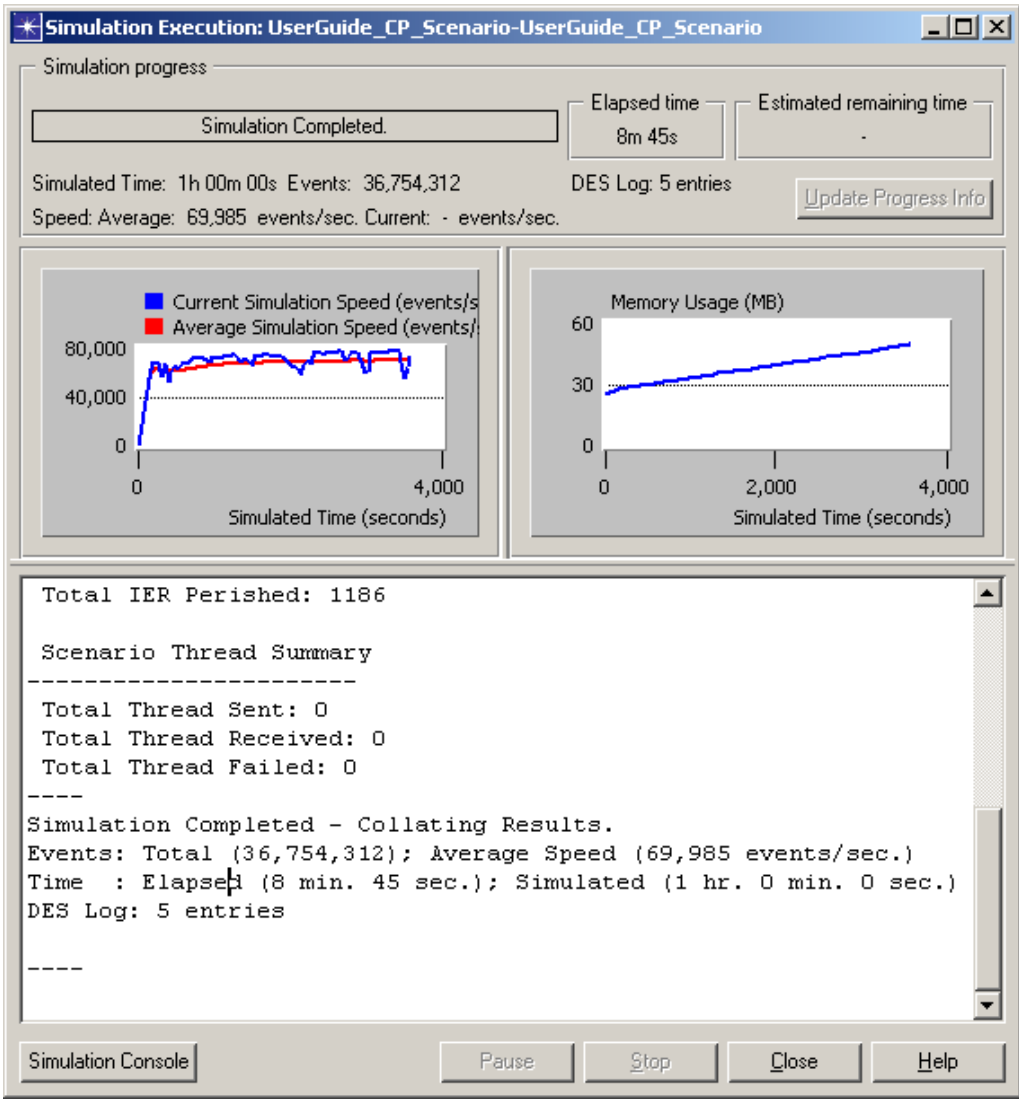


Figure 3-197 Simulation Execution dialog box

JCSS displays a series of messages in this dialog box. These messages show the progress of the simulation, beginning with the initialization stages and continuing with run-time statistics. The following statistics are displayed at each update:

- **Time**: Current simulation time.
- **Events**: Total number of events processed. An event usually signifies the arrival of a packet or an interrupt at a module, triggering some kind of response within the module.
- **Speed: Average –** Simulation Sequence graph displays the average number of simulation events per second of real time (measured since the beginning of the simulation).
- **Speed: Current –** Simulation Sequence graph displays the instantaneous number of simulation events per second (measured using real time over the last update interval).
- **Time: Elapsed –** Real running time of the simulation.

The following buttons stop, pause, and resume the simulation:

- **Pause** / **Resume** button—Pauses / Resumes the current simulation. You can save the output generated by the simulation up to this point when pausing the simulation.
- **Stop** button—Stops the current simulation. You can save the output generated by the simulation up to this point. (This acts as if op_sim_end() was invoked by model code.)

The following panels display:

- Simulation Speed panel—Displays a graph of the current and average event speed for the simulation.
- Memory Usage panel—Displays a graph of the OPNET-managed memory usage of the simulation.

4) When the simulation is finished, click **Close**.

If fatal errors are present in the scenario when a simulation is run, the simulation process is aborted. A pop-up dialog box instructs you to access the Scenario Conversion Log file, which details errors that correspond to incorrect scenario elements requiring adjustment *prior* to a successful simulation. The log file, which resides on the on the same level as the nt.m file, can be opened using any text editor. When the simulation is complete, a file named <project name>-<phase name>.ov is saved in the same location as the nt.m file. This file will be used for results analysis.

Restart Discrete Event Simulation

DES > Restart Discrete Event Simulation: Restart a simulation of the current scenario.

Open DES Log

DES > Open DES Log: Display the log of errors and significant simulation events maintained by JCSS during discrete event simulation. The DES log is written to a tab-delimited ASCII file. Because there may be many events in a DES log, a log browser is provided so that you can view only those events of interest to you.

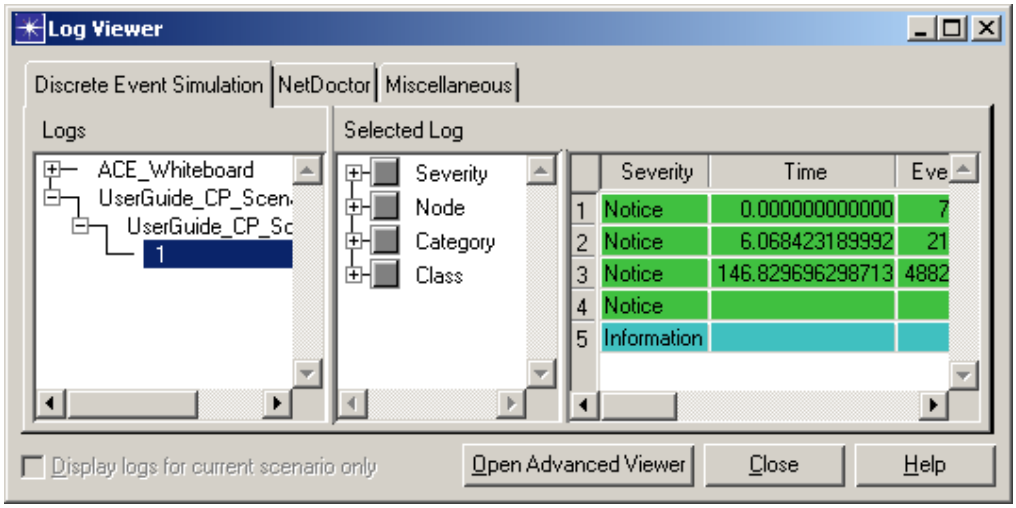


Figure 3-198 Log Viewer

1) Click the module tabs to move between modules for which logs are available. The Miscellaneous tab gives you access to session and error logs.

2) Use the treeview provided in the Logs pane to select from the available logs associated with the selected module.

3) For text-based logs, the Selected Log pane displays a single pane with simple text descriptions of events. For event-based logs, it displays two panes. The first pane displays sections of the log. Select the section(s) that you want displayed. The second pane displays the log data, one event per row.

   - Click and drag the vertical lines in the header row to resize columns.
   - Click on an entry in the Message column to open an edit pad containing the full text of the message.

4) Check (or uncheck) the **Display logs for current scenario only** checkbox to specify whether the treeview (in the Logs pane) should list all logs or only those for the current scenario.

5) Click the **Open Advanced Viewer** button to open an advanced log viewer that provides additional capabilities using a menu bar. The additional capabilities include: changing which column is used to color-code a log entry, changing which columns are visible or filterable, saving the current viewer settings, and exporting the log information to a file.

6) Click the **Close** button to close the Log Viewer dialog box.

IER Results — **DES > IER Results > (*option*)**: Select one of the listed options to view the following simulation results: Sent IERs, Received IERs, Failed IERs, Blocked IERs, Sent Threaded IERS, Received Threaded IERs, and Failed Threaded IERs.

Results — **DES > Results > (*option*)**: Select one of the listed options to view simulation results in various formats.

Results > View Results — **DES > Results > View Results**: Launch a graphical interface, the Results Browser, that allows the planner to browse a list of MOPs loaded from an output vector file and create graph panels to display the MOPs.

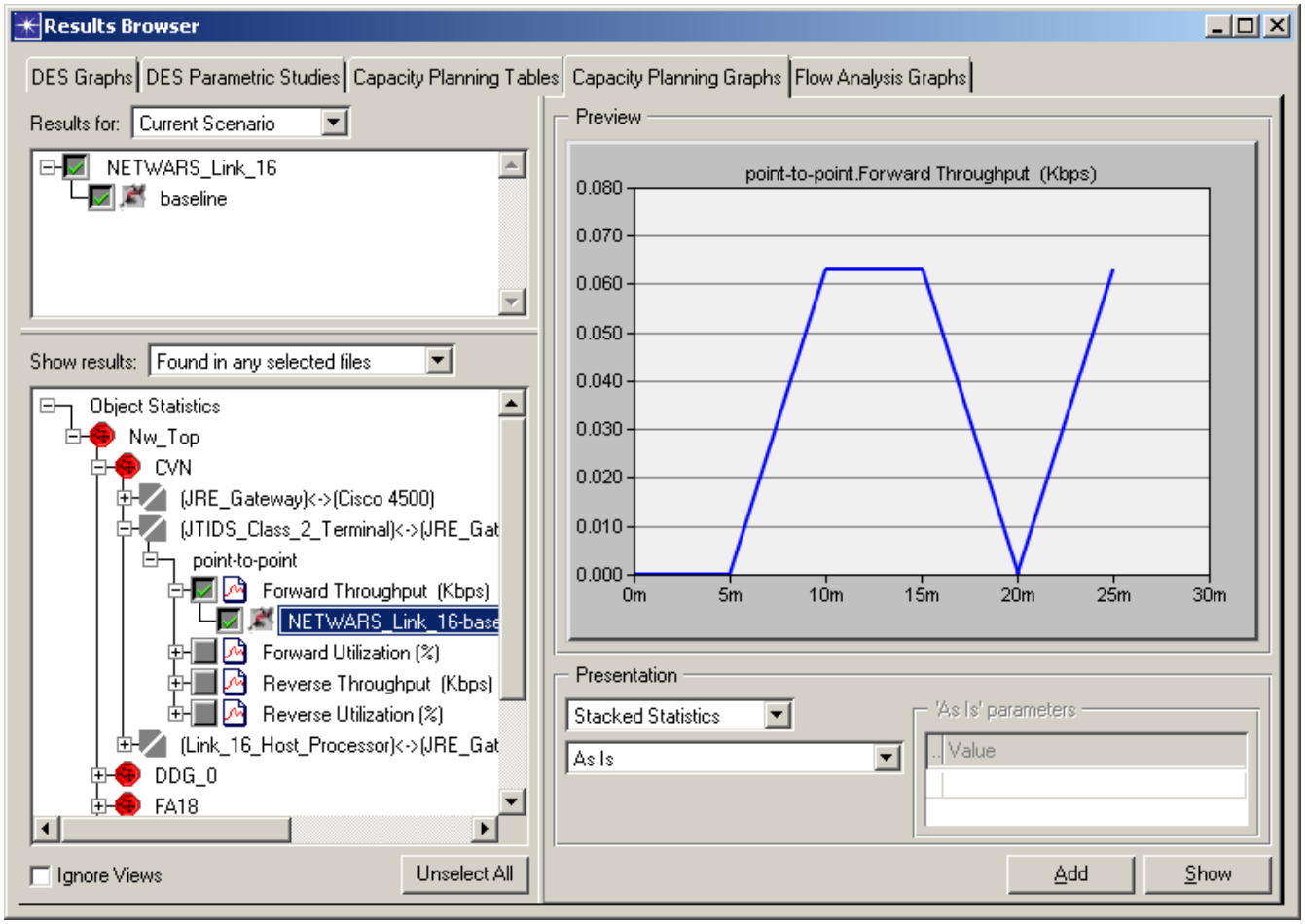


Figure 3-199 View Results

The list of MOPs is categorized and displayed in a tree on the left of the interface. Global statistics are calculated from all objects in the simulation. Object statistics are calculated from individual objects, which are primarily links and OPFACs. One or more statistics may be selected from multiple categories at any time.

Configuration options may be set in the drop-down lists immediately above the lists and below the graph preview. These options control how multiple MOPs or a single MOP from multiple scenarios are combined in a graph panel, and allow the planner to apply filters to the raw MOP data.

If more than one output vector file has been loaded, the upper-left list allows the planner to configure the graph panel to contain graphs of the selected MOP from the most recently loaded file (**Current Scenario**), from all files (**All Scenarios**), or from individually selected files (**Select Scenarios**). If All Scenarios or Selected Scenarios is chosen, only one MOP may be selected.

If multiple MOPs are selected, or a single MOP is being graphed from multiple files, the lower-right lists specify how the graphs will be combined on the graph panel. The **Stacked Statistics** option causes the panel to contain a separate graph for each MOP or file. The **Overlaid Statistics** option causes the panel to contain one graph on which all MOPs are plotted, using colors to distinguish the plots. The **Individual Statistic** option ensures that only one MOP can be selected at any time.

Once the graph has been configured, the planner may create an independent graph panel containing the MOP(s) by clicking on the **Show** button. The graph panel will have the same appearance as the preview, but will exist as an independent dialog and provide editing features that can further configure the graph. Any number of graph panels may be created by repeatedly selecting statistics, setting options, and clicking on **Show**.

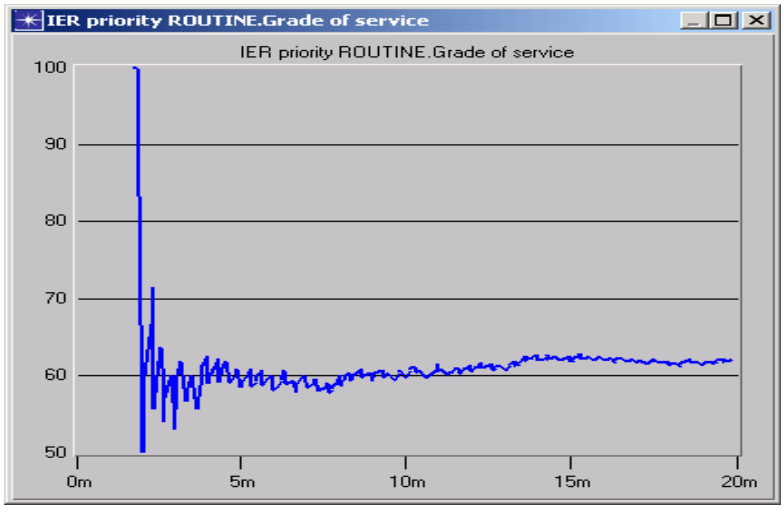


Figure 3-200 MOP graph panel

MOP graphs should be interpreted in the same manner as a standard line graph. The graph heading displays the name of the MOP; the X axis always represents simulation time; and the Y axis represents various units of measurement, depending on the MOP selected.

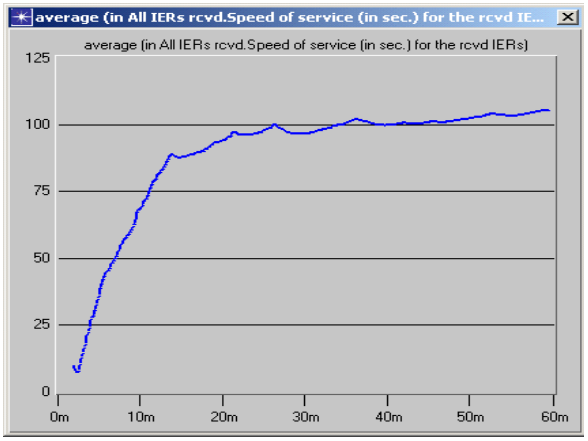


Figure 3-201 Speed of Service MOP

The table below lists MOPs and their descriptions:

Table 3-7 MOP Descriptions

MOP	Description
IERs sent count	A cumulative count of IERs sent.
Number of blocks for each IER sent	The number of times each IER was blocked.
Perishability for the rcvd IERs	A cumulative count of IERs received where the delay (IER_Rcvd IER_Start) is greater than the IER_Perish time.
Speed of service (in sec.) for the rcvd IERs	The delay (IER_Rcvd IER_Start) computed for each received IER.
Forward Data Throughput (in bits/sec.)	The throughput of the point-to-point link in the forward direction.
Reverse Data Throughput (in bits/sec.)	The throughput of the point-to-point link in the reverse direction.
Forward Link Utilization (percent)	The utilization of the point-to-point link in the forward direction.
Reverse Link Utilization (percent)	The utilization of the point-to-point link in the reverse direction.
Broadcast Network Utilization	The broadcast network utilization during the simulation. A value of ‘1’ indicates that the network was utilized, and ‘0’ indicates that it was not utilized.

Table 3-7 MOP Descriptions

MOP	Description
Connection latency (in sec.)	The latency (IER_Sent IER_Start) in establishing a connection.
Grade of service	The percentage of IERs received within the IER_Perish time.
Message completion rate	The ratio of the number of IERs of type ‘data’ received to the number of those sent.
Message error rate	The ratio of the number of IERs of type ‘data’ that failed to the number which were sent.
Call completion rate	The ratio of the number of IERs of type ‘voice’ received to the number of those sent.
Blocking probability	The ratio of the number of IERs that were blocked at least once to the number which were sent.
End-to-end delay (in sec.)	The delay (IER_Rcvd IER_Sent) computed for each received IER.
Channel Utilization (percent)	The percentage of channels on this link that are used.
Voice Throughput (in bits/sec.)	The number of voice calls that go through this link.

MOP graphs allow user interaction. For example, examine statistics by placing the cursor over a segment of the trace to display a tool tip regarding graph coordinates, or zoom in on any segment of the trace by drawing a small box around the segment with the cursor. Right-clicking on an MOP graph opens a shortcut menu that allows manipulation of graph properties and other capabilities such as modifying the look and style of the graph, exporting the graph data to a spreadsheet, and creating a graph template.

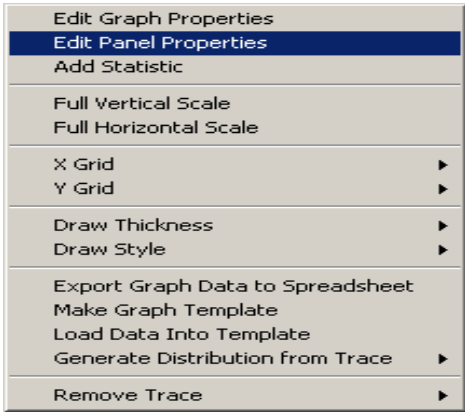


Figure 3-202 MOP shortcut menu

Results > Find Top Statistics

DES > Results > Find Top Statistics: Launch a graphical interface that allows the planner to browse a list of MOPs loaded from an output vector file and create reports listing the objects that have the highest or lowest values for that MOP. Average, maximum, minimum, and standard deviation values may be evaluated for each MOP.

The primary interface is a list of available MOPs that can be evaluated.

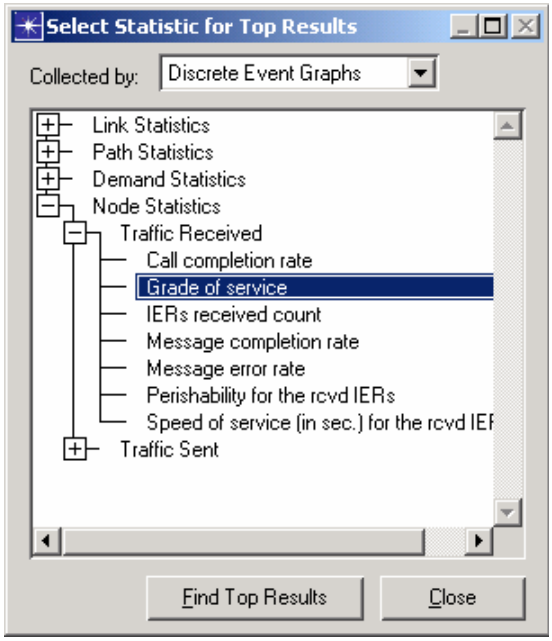


Figure 3-203 Select Statistic for Top Results

1) Select a statistic and click the **Find Top Results** button to launch a report interface. Several report interfaces may be opened concurrently by selecting different MOPs and launching the interface for each in turn.

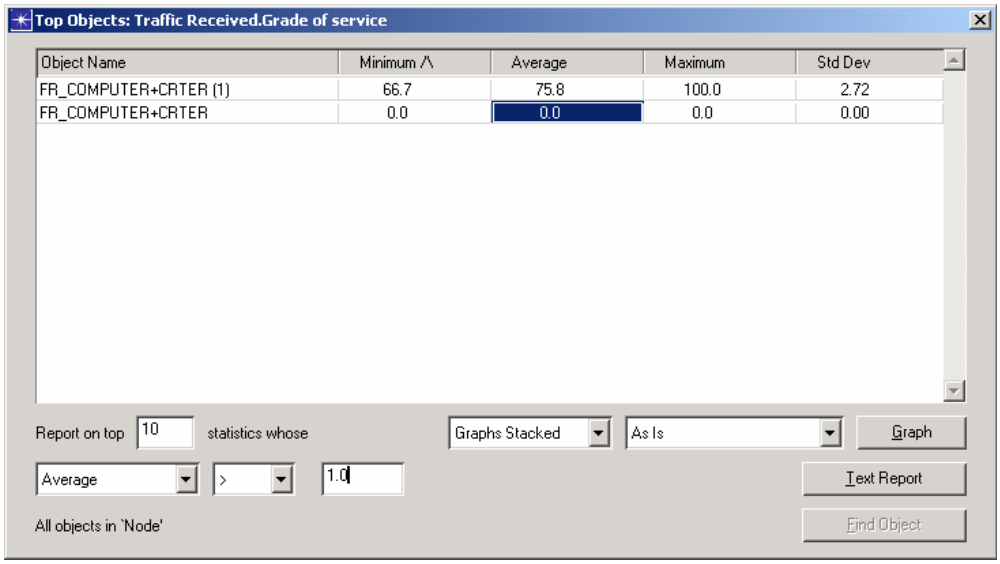


Figure 3-204 Top Results report interface

The report interface lists all objects that have recorded the selected statistic and displays the minimum, average, maximum, and standard deviation values calculated from each object's MOP data. Initially, the list is sorted in decreasing average values, but the planner may dynamically resort the list in increasing or decreasing order for each column by clicking on that column's header.

Both graph and text reports may be created from the report interface. The content of each report is controlled using the fields on the left of the report interface. The maximum number of objects included in the report is set in the **Report on top N statistics** field. Further constraints may be placed on the report by setting a boundary on allowed values for one of the four calculated columns. For example, the report can be constrained to include only objects for which the Average value is greater than (>) a specific value (1.0).

1) Create a graph panel containing the full MOP graph for all objects matching the constraints by clicking on the **Graph** button. The graphs for each object may be stacked or overlaid on the panel, and filters may be applied to the graphs, as specified in the two drop-down lists beside the **Graph** button.

2) Create a text report based on the constraints by clicking on the **Text Report** button. The text report will be displayed in a text-edit interface and may be saved from that interface.

The **Find Object** button is not operational, since the Scenario Builder does not contain a scenario model in which to find the objects named in the list.

Results > View Application Delay Tracking

DES > Results > View Application Delay Tracking: Select an application delay tracking file and use the Application Segment Tracking Viewer to display its data.

Results > View DES Reports

DES > Results > View DES Reports: Displays output tables generated during a discrete event simulation.

Results > Generate Web Report

DES > Results > Generate Web Report...: Generates a Web report that contains one or more of the reports available in the Results Viewer.

1) From the Select Tables for Web Report dialog box, choose the results you want to include in the Web report.

2) Select the Include Network Diagram checkbox if you want the web report to include a graphic of the network topology.

3) Click **Generate**.

4) Select a directory for the HTML files and click **OK**.

5) The web report is created and opens in the default Web browser.

Results > Launch Last Web Report

DES > Results > Launch Last Web Report: Opens most recently generated Web report in the HTML browser.

Results > Import

DES > Results > Import...: Load a previously created .ov file. Select the file to load and then click **OK**; the data is loaded into the current project and is available for analysis. When multiple files are loaded, each file will be a distinct scenario. The name of the scenario will match the name of the output vector file. Multiple scenarios may be active in the Scenario Builder concurrently, so that their MOPs may be graphed and compared.

Play 2D Animation

DES > Play 2D Animation: View collected animations. Toolbar buttons display to control operations for playing animation sequences, pausing animation, and speeding animation up or slowing it down. You can also load animation history files, view “real-time” animation (simultaneously with an executing simulation), and use a variety of operations to control how animation data displays. The largest area of the display—the animation viewing area—is where you can load and view animations. Animation displays in windows called viewers. A viewer is a rectangular region with an internal coordinate system. Viewer windows can be manipulated using the standard GUI operations for moving, resizing, scrolling, and circulating editor windows.

The following toolbar buttons control animation operations:

- **Play**: Activates any animation flow that has been established.
- **Pause**: Pauses an animation indefinitely, until another operation reactivates it.

- **Restart**: Restarts a paused or stopped animation flow from its beginning.
- **Terminate**: Stops an animation flow. After it is stopped, an animation can be restarted only using the Restart Animation flow operation.
- **Slow Down Animation**: Slows down the animation sequence in increments.
- **Speed Up Animation**: Speeds up the animation sequence in increments.

Panels

DES > Panels: Display a list of all open panels and select one to make active.

Panel Operations

DES > Panel Operations > *(option)*: Select one of the operations listed below to control panel display.

- **Arrange Panels**:
  - — **Show All**: This operation draws any analysis panels previously hidden.
  - — **Hide All**: This operation hides all analysis panels that have been drawn.
  - — **Distribute**: This operation arranges open panels along a grid but (unlike tile) does not redraw them to a uniform size.
  - — **Cascade**: This operation arranges open panels in an overlapping pattern that ensures the title bar of each panel is visible. Panels are not redrawn to a uniform size, unlike the tile operation.
  - — **Tile**: This operation redraws open panels to a uniform size and arranges them in a grid.
- **Panel Annotations**:
  - — **Make All Panels Into Annotations**: Creates annotation objects from all open analysis panels.
  - — **Delete All Panel Annotations**: Deletes all analysis panel annotations in the network.
- **Panel Templates**:
  - — **Create From All Panels**: Makes all analysis panels in the current scenario into templates.
  - — **Load With Latest Results**: Applies current statistics to all templates in the scenario.

- **Export Panels**:
  - **To BMP**: Creates BMP images of all graphs currently available in the Scenario Builder. Select a directory in which to place the new image files. One image file is then created for each graph.
  - **To TIFF**: Creates TIFF images of all graphs currently available in the Scenario Builder. Select a directory in which to place the new image files. One image file is then created for each graph.
  - **To PowerPoint**: Creates a PowerPoint document containing images of all graphs currently available in the Scenario Builder. Select a directory and provide a name for the new PowerPoint file.

- **Reload Data Into All Panels**: Reloads data into all panels.
- **Delete All Panels**: Deletes all analysis panels in the current scenario.

Windows Menu

Previous Editor

Windows > Previous Editor: Make the previously displayed editor the current editor.

Circulate Editors

Windows > Circulate Editors: One at a time, make each open editor the current editor.

Hide This Editor

Windows > Hide This Editor: Hide the current editor. If another editor is open, that editor is made the current editor.

Hide Other Editors

Windows > Hide Other Editors: Hide all open editor windows except the current one.

Show All Editors

Windows > Show All Editors: Re-display any hidden editors.

Configure Toolbar

Windows > Configure Toolbar: Customize the main toolbar by adding and deleting desired buttons, selecting button icons and names, and moving button order.

To add a toolbar button, select an available operation from the left pane and then click the **Add>>** button. Use the **Move Up** or **Move Down** buttons to position the button in the desired toolbar location. In the **Title** field, enter a button title. Next, click the **Choose** button and select the desired icon from the palette that displays. Click **OK**.

To remove a toolbar button, select a button in the right pane and then click **<<Remove**.

Use the **Move Up** or **Move Down** buttons to reposition a button in the toolbar.

Use the **Add Separator>>** button to add a divider between button icons.

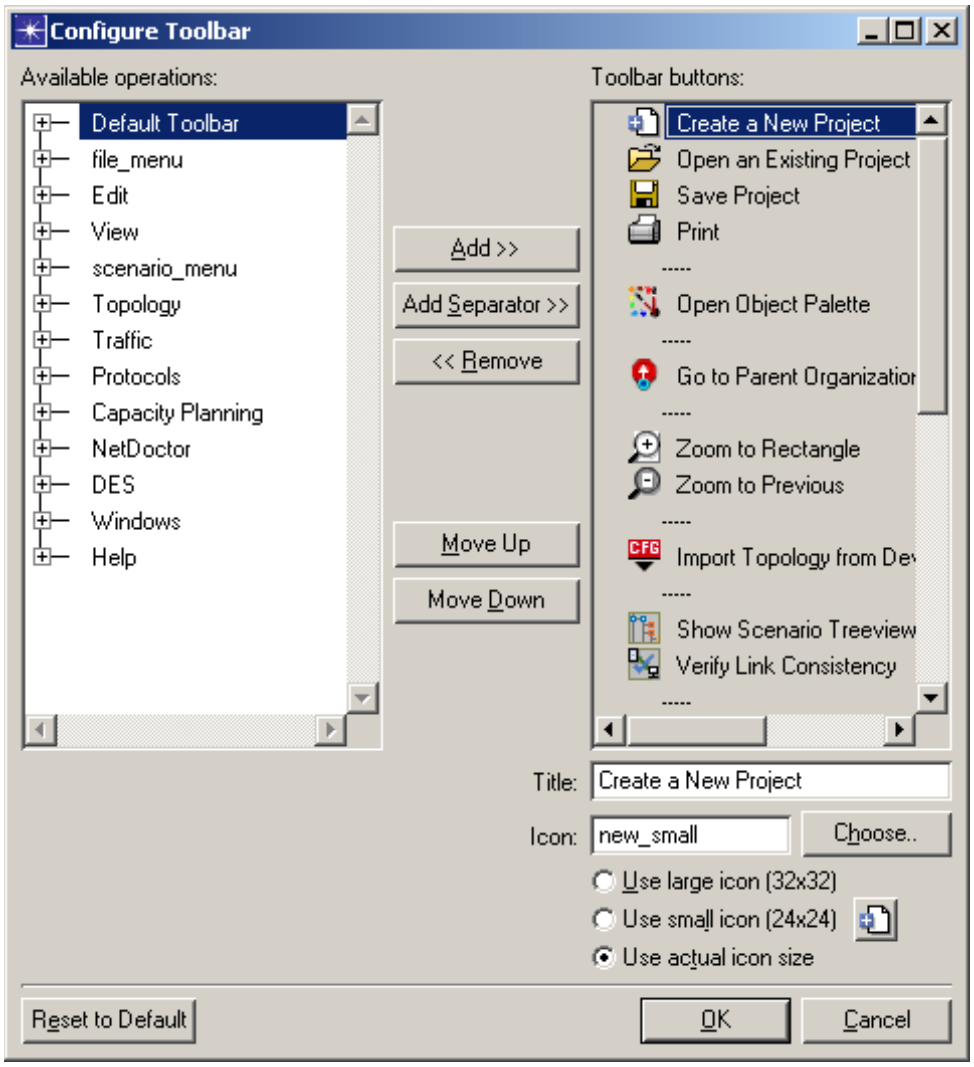


Figure 3-205 Configuring the main toolbar

Show Toolbar **Windows > Show Toolbar**: Show or hide the main toolbar.

Float Toolbar **Windows > Float Toolbar**: Allow the main toolbar to be dragged and dropped anywhere in the viewing area.

Figure 3-206 Floating Toolbar

Note—The **Configure Toolbar**, **Show Toolbar**, and **Float Toolbar** options are also accessible on a shortcut menu, which is available by right-clicking on the toolbar.

Help Menu

This menu is identical to the System Editor Help menu. See the *Help Menu* section in chapter 2 of this manual.

4 Icon Database

The Icon Database allows you to create new icons for OPFACs and organizations displayed within the Scenario Builder.

Accessing the Icon Database

Access the Icon Database using the **File > New Other** or **File > Open Other** commands in the System Editor. The Icon Database is used to create new icon palettes (databases) that can be a collection of new customized icons and icons that are already present. The Image Editor that is a part of the Icon Database is used to create / modify the icons. The individual icons are organized into icon database files, with the suffix .icons. Many icon databases are included as part of JCSS; you can also create your own icon databases.

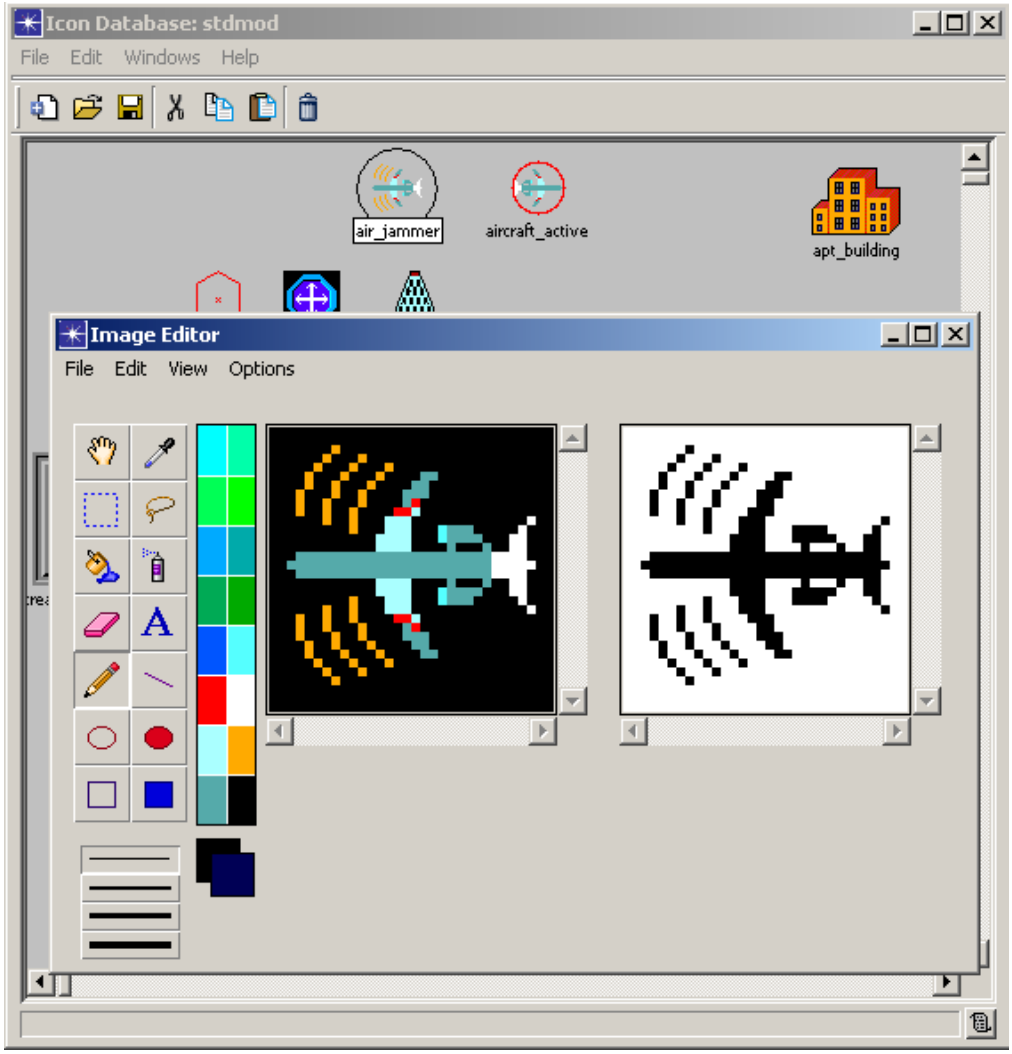


Figure 4-1 Icon Database and Image Editor

You may wish to modify icon databases; you may also wish to modify individual icons. Use the Icon Database window to modify the icon database. Use the Image Editor window to modify individual icons (double-click an icon in the Icon Database to launch the Image Editor.) You cannot create a single icon file, but you can create an icon database file that contains a single icon.

You cannot save an individual icon while in the Image Editor. To write the data related to an individual icon to disk, close the Image Editor and choose the **Save** operation in the Icon Database to save the entire database.

The icon databases included with JCSS are stored in various locations. Icons specific to JCSS are stored in:
`Netwars/Scenario_Builder/12.0.A/netwars/icons`.
COTS icons are stored in:
`Netwars/Scenario_Builder/12.0.A/models/vendor_models`.

Other icon databases are in the appropriate model directory or in the sys directory. When you create an icon database, it is stored in your default directory (the first directory listed in your `mod_dirs` preference, typically `op_models`).

File Menu

New — **File > New**: Create a new icon database. This will open a blank database to which new icons can be added and the database can be customized.

Open — **File > Open**: Open an existing icon database. Once the database is opened, new icons can be added, existing icons can be deleted or modified using the image editor.

Close — **File > Close**: Close the currently open icon database.

Save — **File > Save**: Retain changes to an icon database.

Save As — **File > Save As**: Save the current database under a different name. Specify a name and click **OK**; the icon database is stored in `<User_Data>\SB_Data`.

Manage Model Files — **File > Manage Model Files > (option)**: This submenu lists options which allow you to delete model files, add a model directory, or refresh model directories.

Exit — **File > Exit**: Closes all windows and exits JCSS.

Edit Menu

Cut Icon — **Edit > Cut Icon**: Cut the selected icon to the clipboard.

Copy Icon — **Edit > Copy Icon**: Copy the selected icon to the clipboard.

Paste Icon — **Edit > Paste Icon**: Paste the selected icon from the clipboard.

New Icon — **Edit > New Icon**: Create and add a new icon to an existing or new database. Click in the workspace to create a new icon with **missing name** as the name, and a default image for the icon.

Delete Icon — **Edit > Delete Icon**: Delete an icon from the icon database.

Preferences — **Edit > Preferences**: Display and edit JCSS preference settings.

Windows Menu

This menu is identical to the Scenario Builder Windows menu. See the Windows Menu section in chapter 3 of this manual.

Help Menu

This menu is similar to the System Editor Help menu. See the Help Menu section in chapter 2 of this manual.

App A Time Varying Infrastructure

You can define start and stop times for the infrastructure during a simulation run, which includes relationships, Promina circuits and satellite links. IERs will only be allowed to pass if they fall within the active time window of the infrastructure.

Relationships

The Relationship Attribute dialog box includes a button to specify start and stop times.

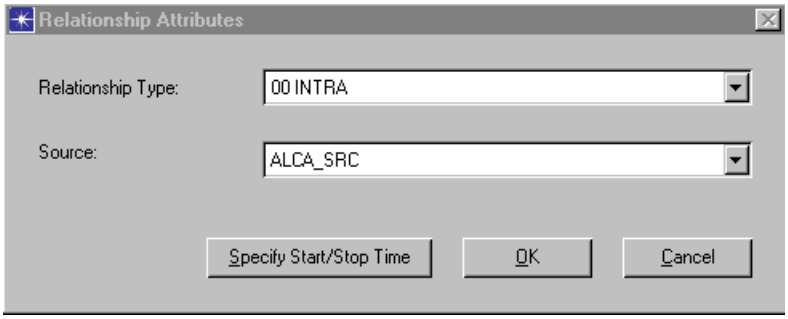


Figure A-1 Relationship Attribute dialog box

Clicking on the **Specify Start/Stop Time** button allows you to specify different start and stop times for a relationship. Start and stop times must be entered consecutively START/STOP, START/STOP, etc., as shown below.

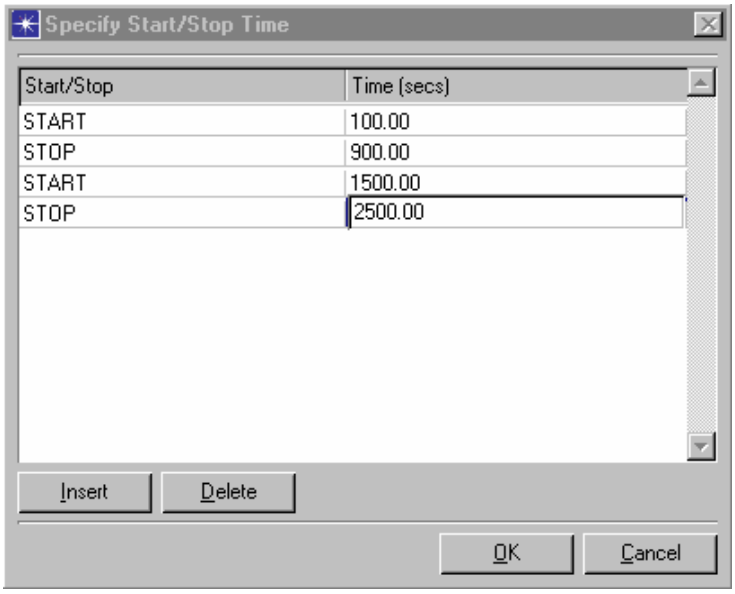


Figure A-2 Specify Start/Stop Time dialog box

IERs imported from a text file or the database that are tied to an explicit relationship will have the start and stop times assigned to the relationship applied to them.

User-created IERs can also have a relationship associated with them. You can set this relationship from a list of all relationship types attached to the producer OPFAC. Start and stop times assigned to the relationship will be applied to the IER.

Imported IERs with implicit relationships are not tied to a relationship, so you have to manually specify the start and stop times for these IERs when viewing/editing traffic.

Promina Circuits (smart multiplexers)

You can specify the start and stop time for a Promina circuit in the Portmap Configuration Table, accessed by opening the shortcut menu for a Promina utility, selecting **Edit Attributes**, and then selecting the **Portmap Configuration** option. In addition, you can also specify these times using the Promina Configuration Wizard, as shown below.

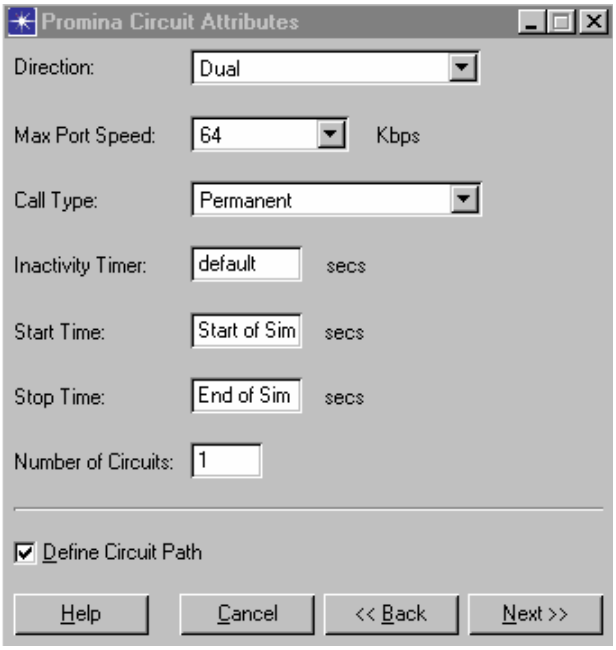


Figure A-3 Promina Configuration Wizard

Satellite Links

The Satellite Link Attributes dialog box allows you to specify start and stop times.

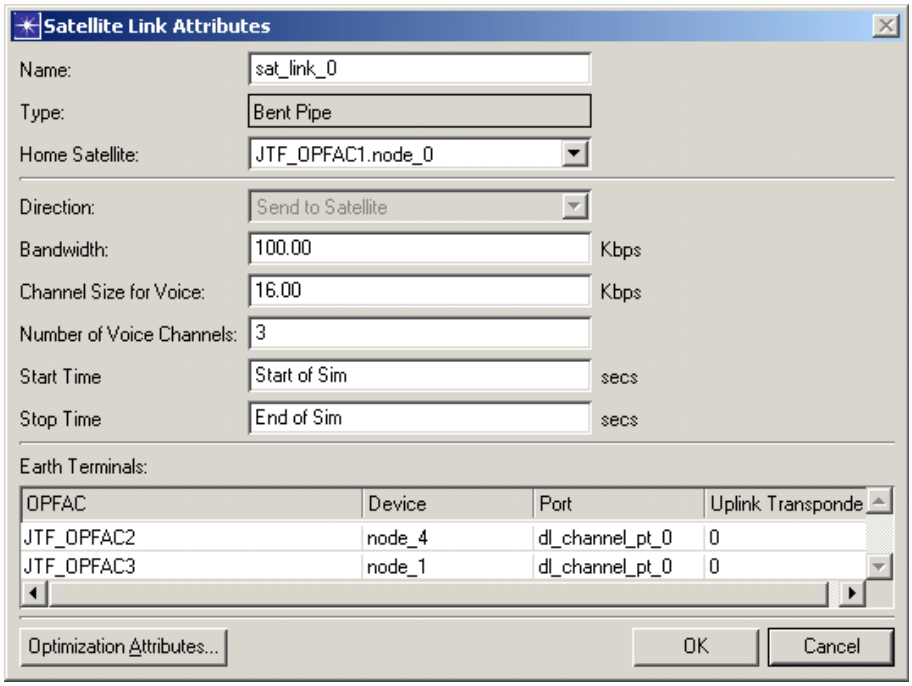


Figure A-4 Satellite Link Attributes

App B System Administration Utilities

Types of Administrators

System Administrators

- *System Administrators* are determined and managed by local policies. They alone have the access to install and remove software from computers. They are members of the **Administrators** group, based on the user ID and password submitted to the operating system authentication management.

Database Administrators

- *Database Administrators* are database accounts that have the authority to create new databases and manage database accounts. These accounts authenticate to the Microsoft SQL Server 2000 Desktop Engine (MSDE) helper application installed as part of JCSS.
- **sa** is a built-in Database Administrator account. JCSS requires a new password for **sa** during installation, because leaving the password unchanged represents a security vulnerability whereby an unauthorized individual could view and make changes to data and login accounts in the MSDE service. The default password is **netwars**.
- JCSS prompts for a Database Administrator user ID and password each time before performing any database operation requiring Database Administrator authority. These operations are determined by MSDE.

Local Administrator User Profile

- Every installation of JCSS has a *Local Administrator* user profile. You can login using this profile to make changes to the pre-configured section of the object library.
- JCSS stores the login information for the five (5) most recently used accounts. The Local Administrator profile is always available and is in addition to the five (5) most recent users.
- This special account does not have any security implications. Just as with any JCSS user profile, the Local Administrator account is not protected by a password.
- In addition to selecting a user profile when starting JCSS, the current user profile can be changed from the **Edit > Change User** menu item of the main JCSS window.

Database Administration

Database User Account Creation

During installation, JCSS creates a database user account with username **scenario_builder** and password **netwars**. Only Database Administrators registered with MSDE may create new Database User accounts.

Procedure B-1 Create a New User Account

1 In the System Editor, select **IER Database > Manage Accounts > Administrator**.

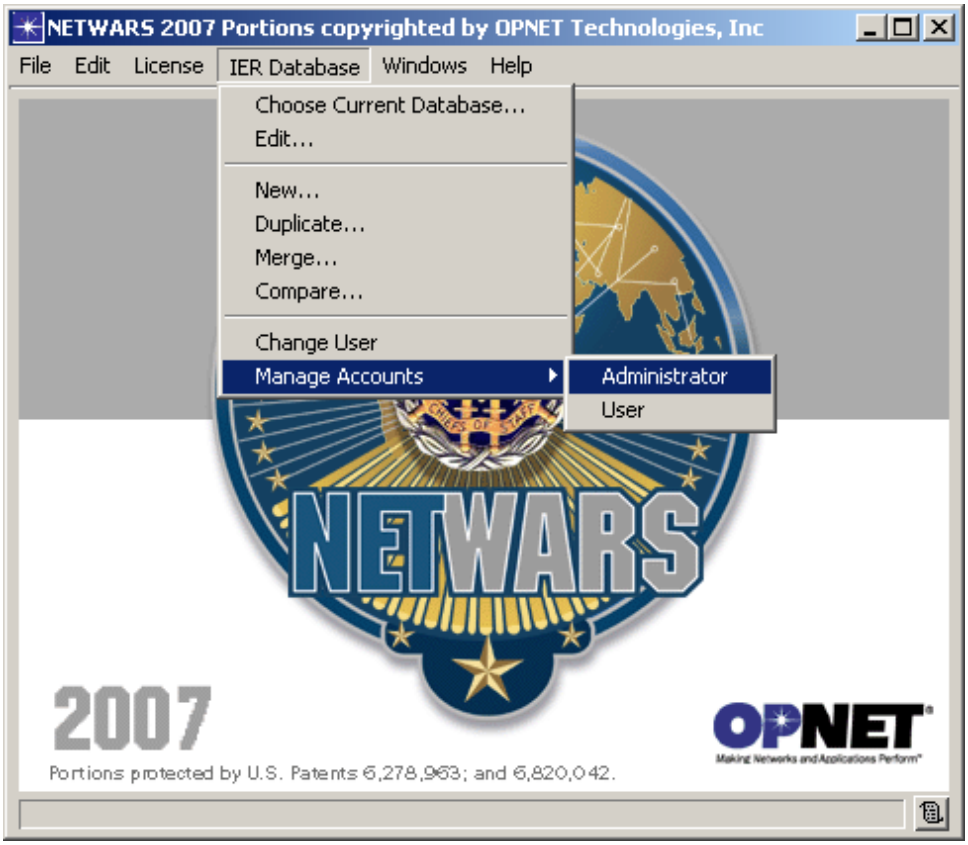


Figure B-1 IER Database Menu's Manage Accounts Submenu

2 Type your Database Administrator ID and password as prompted.

3 If you authenticate properly, a dialog box displays. Type a new user ID and password. The password will not be visible after it is entered, so you must enter it twice to confirm it.

End of Procedure B-1

Database Admin Account Creation

Only Database Administrators registered with MSDE may create new Database Administrator accounts.

Procedure B-2 Create a New Admin Account

1. In the System Editor, select **IER Database > Manage Accounts > User**.
2. Type your Database Administrator ID and password as prompted.
3. If you authenticate properly, a dialog box displays. Type a new Administrator ID and password. The password will not be visible after it is entered, so you must enter it twice to confirm it.

End of Procedure B-2

Database Creation

Only Database Administrators registered with MSDE may create new databases.

Procedure B-3 Create a New Database

1 In the System Editor, select **IER Database > New**.

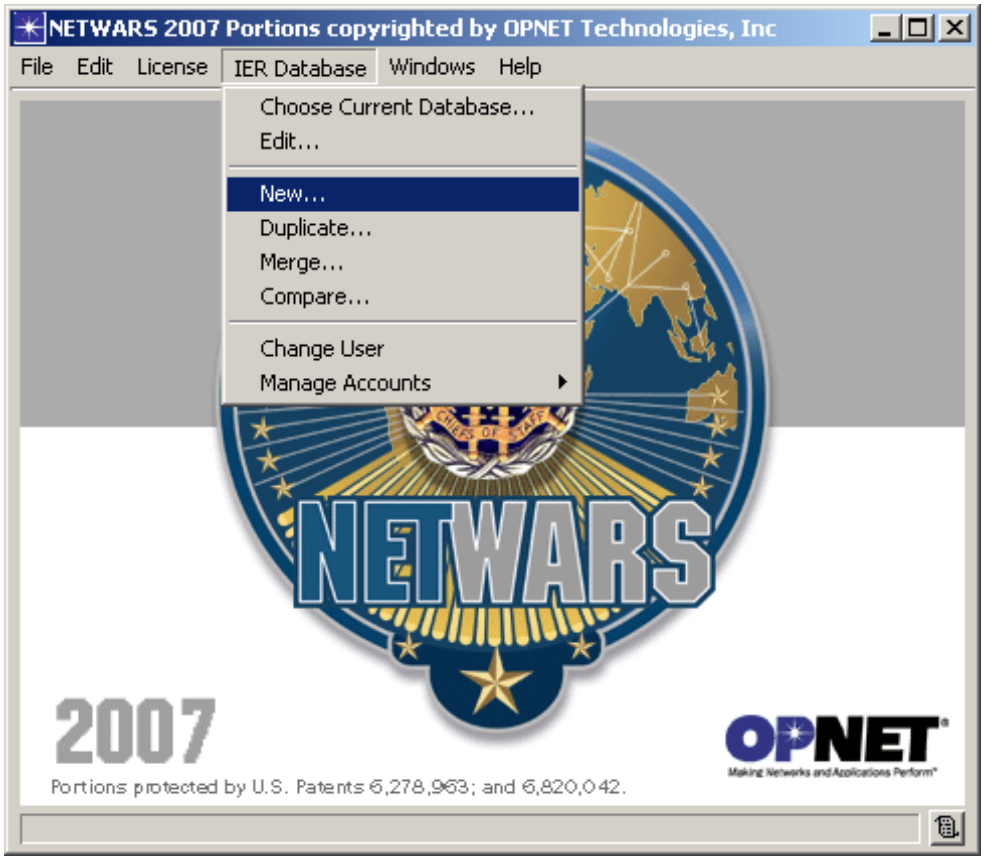


Figure B-2 IER Database Menu of the System Editor

2 Type your Database Administrator ID and password as prompted.

3 If you authenticate properly, a dialog box displays. Type a name for the new database. A new database will be created in that name, if valid, and filed with the default database information.

End of Procedure B-3

Database Copying

Only Database Administrators registered with MSDE may copy an existing database to a new database.

Procedure B-4 Copy a Database

1. In the System Editor, select **IER Database > Duplicate**.
2. Type your Database Administrator ID and password as prompted.
3. If you authenticate properly, a dialog box displays. Type a name for the new database. A new database will be created in that name, if valid, and filed with the default database information.

End of Procedure B-4

App C Operation/Maintenance Procedures

Object Library

JCSS does not enforce access restrictions on the Organization and OPFAC Library. Users should follow the policies and procedures of their organization when creating or modifying library contents.

OPFAC and Organization Library

The OPFAC and organization library are divided into two parts:

- Custom Library
- Pre-configured Library

Custom Library

The custom library contains OPFACs and organizations that were created or modified by the user. All users have privileges to edit the contents of the custom library. You can make copies of the standard templates, store them in the custom library and then make changes to them. OPFACs and organizations in the custom library are called custom templates.

Pre-Configured Library

The pre-configured library contains doctrinal OPFACs and organizations. OPFACs and organizations in the pre-configured library are called pre-configured templates.

Only the local administrator has privileges to modify the contents of the pre-configured library, such as creating new or editing existing OPFACs and organizations, creating new folders, etc. All other users are prevented from making changes to the pre-configured library and their corresponding object palettes.

Even though editing privileges on the pre-configured library is limited to only the local administrator, any user can use its contents when building scenarios. A scenario can contain a combination of pre-configured and custom templates.

It is also possible to have sub-folders within the above set of folders. Use of sub-folders allows you to further categorize the organizations and OPFACs within a service folder.

The figure below shows an example of a library treeview with the pre-configured folders and custom folders created by the user.

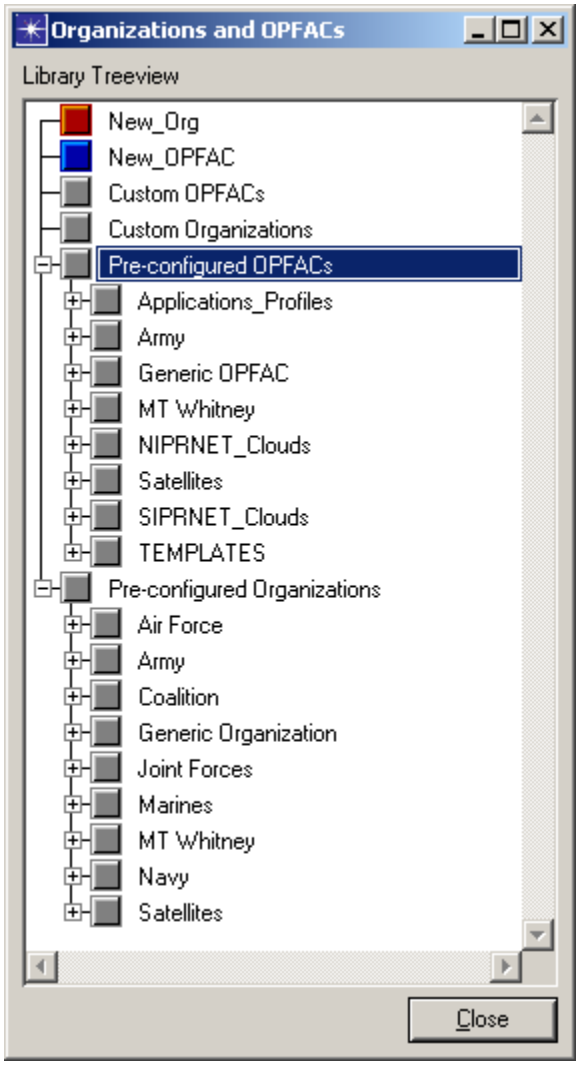


Figure C-1 Library Treeview with Custom and Pre-Configured Folders

OPFAC and Organization Palettes

In addition to the above treeview interface, users can also use objects from the OPFAC and organization palettes to build the scenario. To build a scenario, users can drag-and-drop OPFACs and organizations from the palettes on to the workspace.

There are four main folders to hold the custom and configured OPFACs and organization. They are:

- Custom OPFACs
- Custom Organizations
- Pre-configured OPFACs
- Pre-configured Organizations

Regardless of the sub-folder it is in, every OPFAC or organization will belong to one of the above four palettes.

Editing the Contents of the Library

You can edit the contents of the organization and OPFAC library. You can rename templates, save them under a different name, delete them from the library, and undo or redo template units. Right-click menu options are provided for performing these operations.

If you login using the local administrator profile, you can perform the above operations on the pre-configured and custom sections of the library. If you login as a regular user, you can change the OPFACs or organizations in the custom section, but not the pre-configured section.

Procedure C-1 Rename Template OPFAC/Organization

1. Right-click on a template OPFAC/organization in the library treeview and choose **Rename Template**.

2. Type a new name for the selected OPFAC/organization in the dialog box that displays, and change the folder to which the OPFAC/organization belongs. You can also create a new folder to store this OPFAC/organization.

   The selected OPFAC/organization will be renamed, and if you changed the folder, the files will be moved to the new location on disk. The treeview is also automatically updated.

End of Procedure C-1

Procedure C-2 Save Template OPFAC/Organization As

1. Right-click on a template OPFAC/organization in the library treeview and choose **Save As**.

2. Type a new name for the selected OPFAC/organization in the Save As dialog box, and change the folder to which the OPFAC/organization belongs. You can also create a new folder to store this OPFAC/organization.

   A copy of the selected OPFAC/organization will be made, and if you changed the folder, the files will be moved to the new location on disk. The treeview is also automatically updated.

End of Procedure C-2

Procedure C-3 Delete Template OPFAC/Organization

1. Right-click on a template OPFAC/organization in the library treeview and choose **Delete Template**.

2. Confirm the deletion as prompted to delete the selected OPFAC/organization from the library. The selected OPFAC/organization will be removed from its current location on disk. The treeview is also automatically updated.

End of Procedure C-3

Procedure C-4 Undo/Redo Template Units

1. Right-click on a template OPFAC/organization in the library treeview and choose **Undo** or **Redo**.

2. Confirm the operation as prompted. The treeview is also automatically updated.

End of Procedure C-4

OPNET License Server Management

Standalone Mode

The OPNET application licenses reside on the application computer. The licenses are not available to any other system.

Floating Mode (local)

The OPNET application licenses reside on the application computer, but the licenses are available to the local network as well as the host machine. The license server must run as a network service.

The OPNET License Server shuts down each time a user logs off or shuts down the machine. Users can also shut down the OPNET License Server through the License Manager, available from the JCSS main menu.

Floating Mode (external)

The OPNET application licenses reside on another computer, so this machine must compete for licenses with other computers on the local network.

License Server Files and Directories

C:\OPNET_license

This folder cannot be moved or renamed. It will contain license files for Standalone and Floating (local) licensing modes. The folder and all its files must be writable by Authenticated Users, or those users will be unable to acquire a lock on a license to use the application.

licensing.ef

The location of this file is:

- `NETWARS\Scenario_Builder\12.0.A\sys\configs\global_prefs\licensing.ef`

The format of the `licensing.ef` file is:

- `license_server_standalone : true | false`
- `license_server : /LICENSE_SERVER/`
- `license_port : /LICENSE_PORT/`

where:

- `/LICENSE_SERVER/` is the local hostname
- `/LICENSE_PORT/` is the desired port for the server
- `license_server_standalone` may be either true or false

env_db12.0

The `env_db12.0` file contains option settings for JCSS and OPNET commercial modules. Among these options are license server settings. The `env_db12.0` file can be edited manually or through the **Edit > Advanced** menu item. The `env_db12.0` file license settings should be consistent with the settings in the `licensing.ef` file. The absence of license server settings in the `env_db12.0` file means that the `licensing.ef` values take precedence.

The `env_db12.0` file is located in the following location: `NETWARS\Scenario_Builder\op_admin\env_db12.0`.

Manually Configuring License Settings

Normally, configuration of license settings should be performed through the JCSS user interface. Manually changing configuration files should be used only as a last resort.

`/hostname/` represents the hostname of the localhost system. To determine the hostname of a system, open a DOS console and run the hostname command.

`/port/` represents the desired port, one of port_a, port_b, or port_c.

Procedure C-5 Configure Localhost Floating Licensing

1. Make sure that all JCSS windows are closed. This includes any JCSS editors as well as the OPNET License Manager.
2. The file named `licensing.ef` is located in `NETWARS\Scenario_Builder\12.0.A\sys\configs\global_prefs` Modify it as follows:
   - **2.1** Set `license_server_standalone` to FALSE
   - **2.2** Set `license_server` to `/hostname/`
   - **2.3** Set `license_port` to `/port/`
3. Modify the `env_db12.0` file located in `NETWARS\Scenario_Builder\op_admin\` as follows:
   - **3.1** Set `license_server_standalone` to FALSE
   - **3.2** Set `license_server` to `/hostname/`
   - **3.3** Set `license_port` to `/port/`

End of Procedure C-5

Procedure C-6 Change from Localhost to External Floating Licensing

1. Using the License Manager, make sure that the localhost license server is shut down.
2. Close all JCSS applications.
3. The file named `licensing.ef` is located in `NETWARS\Scenario_Builder\12.0.A\sys\configs\global_prefs` Modify it as follows:
   - **3.1** Set `license_server_standalone` to FALSE
   - **3.2** Set `license_server` to `/hostname/`
   - **3.3** Set `license_port` to `/port/`

4 Modify the `env_db12.0` file located in `NETWARS\Scenario_Builder\op_admin\` as follows:

4.1 Set `license_server_standalone` to FALSE

4.2 Set `license_server` to `/hostname/`

4.3 Set `license_port` to `/port/`

End of Procedure C-6

Procedure C-7 Change from Localhost to Standalone Licensing

1 Using the License Manager, make sure that the localhost license server is shut down.

2 Close all JCSS applications.

3 The file named `licensing.ef` is located in `NETWARS\Scenario_Builder\12.0.A\sys\configs\global_prefs` Modify it as follows:

3.1 Set `license_server_standalone` to TRUE

3.2 Set `license_server` to `/hostname/`

3.3 Set `license_port` to `/port/`

4 Modify the `env_db12.0` file located in `NETWARS\Scenario_Builder\op_admin\` as follows:

4.1 Set `license_server_standalone` to TRUE

4.2 Set `license_server` to `/hostname/`

4.3 Set `license_port` to `/port/`

End of Procedure C-7

License Management and Product Modules

JCSS comes with a built-in license that allows you to run the program. A special "simulation runtime license" is required in order to run simulations. Additional licenses are required for ACE and NetDoctor modules.

If the module licenses are installed on a single license server rather than on individual machines, various machines can share licenses, which may reduce the total number of licenses required. Simulation runtime licenses, ACE licenses, and NetDoctor licenses must be purchased from OPNET (http://www.opnet.com/products).

To install module licenses, start by running JCSS (go to **Start > Programs > NETWARS > NETWARS v2007**). From the System Editor main menu, select **License > License Management**.

Current instructions for loading and configuring OPNET License permits are available from the OPNET Support web site (http://www.opnet.com/support).

Log Files

The various log files assist in troubleshooting or tracking user activity. Users should examine the relevant log files before consulting NETWARS Technical Support, and may be asked to include log files in submissions.

Note—All log files grow over time and can potentially become a constraint on system resources. Administrators should periodically check the sizes of the log files and prune the logs as needed.

nw_odbc_log

- Located in `NETWARS\User_Data`
- Runtime log that records unusual events associated with extracting information from the IER Database, service provided by MSDE (Microsoft SQL Server Desktop Engine).

err_log

- Located in `NETWARS\Scenario_Builder\op_admin`
- OPNET and NETWARS application warnings, recoverable errors, errors, and program abort messages, with stack traces for context, are stored in this file.

Archiving and Backup Procedures

Project Packaging

Each project has a single .prj file in the project directory under NETWARS\User_Data\Projects. The project folder and the .prj file have the same name as the project itself. A JCSS project may make use of other library files. Library files may be used by any number of projects. The only way to know which library files are included in a project is to examine the contents of the project.

Scenario Files

Each project consists of one or more scenarios. Each scenario is a subfolder under the main project folder. The scenario consists of a file with the extension '.nt.m' and possibly an XML file with the same base name.

Traffic Files

Demand Files

A demand file for each scenario, if one exists, would be located in the NETWARS\User_Data\Projects\<Project>\<Scenario> directory. Its name would be the Project Name followed by the Scenario Name connected with a plus ('+') and with an extension of '.dm'.

IER Text Files

All IER Text files are available for use by every scenario. IER Text files are located in the NETWARS\User_Data\IER_Text_Files directory and have an extension of '.txt'.

Organization Files

Organizations are represented by '.nt.m' files located in the NETWARS\User_Data\Organizations directory. The project may include any, all, or none of the template organizations.

OPFAC Files

OPFACs consist of '.nd.m' and '.nt.m' files. The names of the various files that relate to a specific OPFAC have similar base names, though not identical.

All .nd.m file names contain the prefix "opfac_".

OPFAC .nd.m and nt.m files

The OPFACs used by a project/scenario may reside in any folder under the NETWARS\User_Data\OPFACs folder. The 'nt.m' file includes all the OPFAC specific information while the '.nd.m' file is only necessary for an OPFAC to appear in a palette.

Trajectories

Trajectory files, ending in '`.trj`', may be applied to any organization or OPFAC in its JCSS attributes. Trajectory files may be stored in any directory included in the system's mod_dirs environment attribute. Many trajectory files are stored in the `NETWARS\User_Data\SB_Data` directory, although they may be in other directories also.

IER Database Archiving

Procedure C-8 Save to Text File

1. To store relevant database entries in to an IER Text File for transport or backup, load the entries into the IER Editor Table using the Query functionality.

   A query with no criteria specified will retrieve all records in a database.

2. Click the **Export** button and name a text file.

End of Procedure C-8

Procedure C-9 Restore from Text File

1. To import IERs from a text file to the database, open the IER Editor Table and select **Import**, and then select the file containing the IERs.

   The IERs in the text file will load into the editor table.

2. Click the **Save to Database** button and save the records into the database.

   These imported records will overwrite records with corresponding IER IDs in the database.

End of Procedure C-9

App D Error Recovery Guidelines

Automatic Backup of Project Files

Project files are backed up automatically at a predetermined interval (the default setting is 15 minutes.) Users may modify the backup interval by selecting **Edit > Preferences** and changing the “backup_interval” preference. The backed up files have ‘`-backup<index>`’ appended to the filename. You can load these backup files into the Scenario Builder.

Note—Setting the “backup_interval” preference to 0 turns the backup off.

Glossary

B

Broadcast Networks Network of radio system devices that are tuned to same frequency and belong to same frequency hop group. All the radio devices belonging to same broadcast network hear transmission by one radio device.

C

Connectivity Links Created based on the subscriber requirements. Connectivity links are different from planning links in that they are connected to actual devices instead of organizations.

Courses of Action Define specific conditions or situations, such as when an enemy uses chemical, biological, or nuclear weapons. It is important to note that time phases and courses of action perform exactly the same way in JCSS. You simply have the option of calling your scenario subdivisions time phases or courses of action.

I

IERs The acronym for **I**nformation **E**xchange **R**equirement. IERs are the primary traffic representation in JCSS. A relationship between OPFACs is needed to create an IER.

K

Kernel Procedure An OPNET provided function that supports the development of protocols and algorithms. All Kernel Procedures start with op_.

M

Mainframe Characterization Module MCE is an optional module that allows you to import measured performance data, and analyze it to create workloads suitable for the mainframe models.

Mod_Dirs An environment attribute that tells OPNET which folders to look in for locating files. The mod_dirs attribute is found under **Edit > Preferences**.

MOPs The acronym for **M**easures **o**f **P**erformance. Analyze the results of a simulation session to produce a set of MOPs. Manipulate the set of MOPs using data filters and produce graphs of the data. The MOPs explain various communication burdens, latency, and other critical performance measures associated with communication infrastructures.

O

Online Documentation An Adobe Acrobat manual that has information about the OPNET models, kernel procedures, modeling concepts etc. The manual can be launched from Modeler by choosing the Online Documentation option under the Help menu.

OPFACs The acronym for **Op**erational **Fac**ility. OPFACs are the fundamental building blocks of JCSS. They are collections of communications devices that are assigned to military units, which are used to construct Organizations.

Organizations Hierarchies of military units. An organization is a container that can hold OPFACs and other organizations (sub-organizations).

P

Planning Links Placeholders for the connectivity links that will be created based on the subscriber requirements. These links can have organizations or OPFACs as terminating points, and show the anticipated bandwidth. Planning links cannot be used for discrete even simulation and capacity planning analyses.

Process Registry It is a model-wide registry where any process mode can register itself and any process model can obtain information about other process models that are registered. For a list of kernel procedures available for using the process registry, refer to the OPNET Modeler online documentation, *General Model's Manual*, OPNET Model Support chapter, Process Registry section (consult the IT Guru documentation set, available via **Help > Documentation > IT Guru Documentation**).

Projects Projects are containers for scenarios, time phases or courses of action. Related scenarios can be grouped under one project. JCSS enables you to open and work with multiple projects simultaneously.

S

Scenarios Scenarios are instances of C4I communications architectures that consist of associated OPFACs and Organizations, Information Exchange Requirements (IERs) and links. Scenarios are stored as part of a project created within the Scenario Builder. Thus, projects act as containers for scenarios and a single project may include several scenarios, i.e., instances of C4I communications architecture.

Server Characterization Editor SCE is an optional module that allows you to import server performance information for analysis of the processes and applications, and characterization into workloads used as the basis for simulation modeling.

Simulation Engine The COTS OPNET Modeler tool. It processes the simulation events and provides the output to the Scenario Builder.

U

Unified Modeling Language UML is an industry standard set of graphical notations to describe a system from an object-oriented approach. Diagrams include a set of static notations (class diagrams, use case diagrams) and a set of dynamic notations (state diagrams and sequence diagrams). UML does not require a specific design process and does not require implementation with any specific object-oriented languages or tools. The state diagrams, for example, are consistent with OPNET Modeler's Process Model notation.

Index

G

H

I

J

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z