

RECLAMATION

Managing Water in the West

Technical Report: SRH-2016-09

Modeling In-Stream Structures and Internal Features with SRH-2D



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Denver, Colorado

February, 2016

Mission Statements

The mission of the Department of the Interior is to protect and provide access to our Nation's natural and cultural heritage and honor our trust responsibilities to Indian tribes and our commitments to island communities.

The mission of the Bureau of Reclamation is to manage, develop, and protect water and related resources in an environmentally and economically sound manner in the interest of the American public.

Technical Report: SRH-2016-09

Modeling In-Stream Structures and Internal Features with SRH-2D

Prepared by:

Yong G. Lai, Ph.D., Hydraulic Engineer
Sedimentation and River Hydraulics Group
Technical Service Center

Date

Report Peer Reviewed by:

Mike Sixta, P.E., M.S., Hydraulic Engineer
Sedimentation and River Hydraulics Group
Technical Service Center

Date



U.S. Department of the Interior
Bureau of Reclamation
Technical Service Center
Denver, Colorado

February, 2016

Table of Contents

	Page
Executive Summary	1
1.0 Introduction	2
2.0 About SRH-2D Model	3
3.0 Weir Module	5
3.1.1. Theory of the 1D Weir Module.....	5
3.1.2. A Sample Case Running 1D Weir Module	7
4.0 Gate Module	10
4.1.1. Theory of the 1D Gate Module	10
4.1.2. A Sample Case of 1D Gate Module	12
5.0 Culvert and Bridge Module	15
5.1.1. Theory of FST Culvert Module.....	15
5.1.2. A Culvert Flow Case without Overtop.....	20
5.1.3. A Bridge Flow Case with Overtop.....	22
5.1.4. A Sample Culvert Flow Modeling with HY-8 Module.....	25
6.0 2D Pressurized Flow Module	28
6.1.1. Theory of 2D Pressure Flow Module.....	28
6.1.2. Bridge/Culvert Parameters	30
6.1.3. A Simple 2D Pressure Flow Case without Overtop.....	30
6.1.4. A Simple 2D Pressure Flow Case with Overtop.....	33
6.1.5. A Field-Scale Test Case with Arch Bridge	35
7.0 Internal Flow Obstructions	38
7.1.1. Theory	38
7.1.2. A Sample Case	41
8.0 Internal Flow Boundaries	43
8.1.1. Theory and Usage.....	43
8.1.2. Example of Coupled Internal Boundaries	44
9.0 References	46

Index of Figures

	Page
Figure 1. Solution domain and 2D mesh of the sample weir flow case.....	8
Figure 2. Zoom-in view of the weir	8
Figure 3. Simulated flow with velocity vector and magnitude with the 1D weir flow case	8
Figure 4. The SIF file used to run the 1D weir case	9
Figure 5. Illustration of the gate inclination angle	11
Figure 6. A zoom-in view of the gate (flow is from top left to bottom right)	13
Figure 7. Simulated flow with velocity vector and magnitude with the 1D gate module.....	13
Figure 8. The SIF file used to run the 1D gate case.....	14
Figure 9. A culvert flow that is either under inlet control (A) or outlet control (B)	16
Figure 10. A zoom-in view of the culvert (flow is from top left to bottom right)	21
Figure 11. Simulated flow with velocity vector and magnitude for the sample case without overtop using the 1D FST culvert module.....	21
Figure 12. The SIF file used to run the 1D culvert case without overtop.....	22
Figure 13. Geometry and mesh of the bridge case (flow is from left to right)	23
Figure 14. Simulated flow with velocity magnitude and vector with the overtopped bridge 1D model	24
Figure 15. The SIF file used to run the overtopped bridge 1D model	24
Figure 16. Bathymetry and culvert location (flow is from top left to bottom right)	25
Figure 17. The SIF file used to run the HY-8 culvert case	26
Figure 18. Simulated flow with velocity magnitude and vector the HY-8 culvert case.....	27
Figure 19. The SIF file used to run the pressure flow over a culvert without overtop	32
Figure 20. Simulated flow with velocity magnitude and vector with the pressure flow module for the culvert flow simulation	32
Figure 21. The geometry and bed elevation of the bridge and model domain	33
Figure 22. The SIF file used to run the pressurized flow through a bridge with overtop	34
Figure 23. Simulated flow with velocity magnitude and vector with the pressure flow through a bridge with overtop	34
Figure 24. The model domain and bridge geometry (plots are from Maranzoni et al. 2015)	36
Figure 25. Predicted water depth contours at time = 10 s when the dam-break bore is under the arch bridge (grey area are pressurized)	37
Figure 26. Drag coefficient for flow over a circular cylinder as a function of the Reynolds number	39
Figure 27. Solution domain, the mesh, bathymetry and the location and size of the obstruction.....	41

Figure 28. Simulated flow with velocity magnitude and vector with the flow
obstruction case..... 41
Figure 29. The SIF file used to run the obstruction case 42
Figure 30. 2D domain, mesh and bed elevation of the internal boundary example
..... 44
Figure 31. Predicted flow velocity for the internal boundary example 45
Figure 32. The SIF file used to run the internal boundary case 45

Index of Tables

	Page
Table 1. Default Submerged Weir Coefficients ^a (FHWA, 2002)	6
Table 2. Culvert Inlet Flow Formula Coefficients (Source: FHWA 2002)	18
Table 3. Entrance loss coefficient for various culvert types and entrance conditions under the outlet control culvert flow	19
Table 4. Culvert Manning roughness coefficients for various culvert barrael materials and barrel conditions under the outlet control culvert flow	20
Table 5. Approximate drag coefficients for various flow obstructions	39

Executive Summary

The existing SRH-2D model is extended to simulate flows impacted by in-stream structures, such as weirs, gates, bridges and culverts, and in-stream features such as obstructions and internal boundaries. These new features are for flow modeling only without sediment transport. The work was partially funded by the Federal Highway Administration (FHWA) and carried out as a joint effort between the U.S. Bureau of Reclamation and FHWA. SRH-2D has been formally adopted since 2014 by FHWA as one of the FHWA's agency models.

This document serves as the User's Manual on how to use the new features with SRH-2D, and it was funded through the Reclamation Manuals and Standards program. It describes the theory, numerical implementation, user instructions and test examples for each in-stream structure and feature developed. The purpose is to document the works performed. This report may also serve as a user's guide on how to use SRH-2D to simulate flows with in-stream structures and internal features.

1.0 Introduction

This report describes additional modeling capabilities of SRH-2D developed for flow simulation with (a) in-stream structures such as weirs, gates, bridges and culverts, and (b) internal features such as in-stream obstructions which incur extra flow resistance (bridge pier, bridge deck, woody piles, etc) and internal boundaries for water diversion and inflow. These new features have been carried out under an Inter-Agency Agreement between the U.S. Bureau of Reclamation (Reclamation) and the Federal Highway Administration (FHWA). The work was partially funded by FHWA.

The SRH-2D hydraulic model has been developed and used by Reclamation and other agencies since 2004 for numerous projects. It is recognized for its ability to achieve stable and reliable two-dimensional (2D) solutions to complex hydraulic problems quickly and effectively. FHWA hydraulic engineers have independently evaluated SRH-2D by comparing SRH-2D with several other 2D models and seen a significant potential benefit for future Department of Transportation (DOT) hydraulic projects. In addition to the new feature additions to SRH-2D, Aquaveo, LLC has also been tasked under a separate contract to develop a customized SRH-2D model interface within the SMS software package, which is used for pre- and post-processing of SRH-2D.

It is noted that the in-stream structure modeling capabilities reported are limited to flow simulation, and they are not yet to be available for sediment transport modeling.

In the following, each new module (or feature) developed within SRH-2D is described. The description includes the theory used, instruction on how to use it with SRH-2D, and sample test cases using the new feature. The description in this document should be sufficient to provide users with instructions on how to simulate flows with the in-stream structures and internal features.

2.0 About SRH-2D Model

SRH-2D, Sedimentation and River Hydraulics – Two-Dimensional model, is a 2D depth-averaged hydraulic and sediment transport model for river systems developed at the Bureau of Reclamation, Technical Service Center. The hydraulic flow modeling theory was documented by Lai Yong (2008; 2010). The model adopts the arbitrarily shaped element method of Lai et al. (2003), the finite-volume discretization scheme, and an implicit integration scheme. The numerical procedure is sufficiently robust that SRH-2D can simultaneously model all flow regimes (sub-, super-, and trans-critical flows) and both steady and unsteady flows. The special wetting-drying algorithm makes the model very stable in handling flows over dry surfaces. The mobile-bed sediment transport module adopts a general sediment transport method documented by Greimann et al. (2008), Lai and Greimann (2010) and Lai et al. (2011). The mobile-bed module predicts vertical stream bed changes by tracking multi-size, non-equilibrium sediment transport for suspended, mixed, and bed loads, and for cohesive and non-cohesive sediments, and on granular, erodible rock, or non-erodible beds. The effects of gravity and secondary flows on the sediment transport are accounted for by displacing the direction of the sediment transport vector from that of the local depth-averaged flow vector.

Major capabilities of SRH-2D are listed below:

- 2D depth-averaged solution of the dynamic wave equations for flow hydraulics;
- An implicit solution scheme for solution robustness and efficiency;
- Hybrid mesh methodology which uses arbitrary mesh cell shapes. In most applications, a combination of quadrilateral and triangular meshes works the best;
- Steady or unsteady flows;
- All flow regimes simulated simultaneously: subcritical, supercritical, or transcritical flows;
- Mobile bed modeling of alluvial rivers with a steady, quasi-unsteady, or unsteady hydrograph.
- Non-cohesive or cohesive sediment transport;
- Non-equilibrium sediment transport;
- Multi-size sediment transport with bed sorting and armoring;
- A single sediment transport governing equation for both bed load, suspended load, and mixed load;
- Effects of gravity and secondary flows at curved bends; and
- Granular bed, erodible rock bed, or non-erodible bed.

SRH-2D is a 2D model, and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures such as weirs, diversion dams, release gates, coffer dams, etc.; bends and point bars; perched rivers; and multi-channel systems. 2D models may also be needed if certain hydraulic

characteristics are important such as flow recirculation and eddy patterns; lateral variations; flow overtopping banks and levees; differential flow shears on river banks; and interaction between the main channel, vegetated areas and floodplains. Some of the scenarios listed above may be modeled in 1D, but additional empirical models and input parameters are needed and extra calibration must be carried out with unknown accuracy.

3.0 Weir Module

Flows over weirs or roadways may be simulated using one of the two approaches with SRH-2D: (a) direct 2D method, and (b) empirical 1D method. With the 2D method, flows over a weir or roadway are simulated with the 2D depth-averaged approach directly by using 2D mesh cells covering the weir, along with its geometry. The 2D approach has the advantage that both free and submerged weir flows are automatically simulated without abrupt switch-over between them. The approach is straightforward and no extra model setup is required. Additional energy loss due to the presence of a weir may be added through increased Manning's coefficient in the weir zone. Calibration may be carried out to determine the extra roughness needed for modeling if measured head loss (or water surface elevation change) is available.

This section describes the new capability of simulating flow over a weir using the empirical 1D weir equations. The 1D weir module computes a total flow rate (discharge) over a weir using a set of 1D empirical equations. The need for the 1D weir module may be justified for some flow cases. Flow over weirs or weir-like structures such as roadway embankments can have significant vertical motion and might not be simulated accurately using the 2D depth-averaged flow approximations unless the extra energy loss caused by the weir is taken into account properly. The total flow through the model may be estimated more accurately with the 1D module if accurate empirical equations are available.

3.1.1. Theory of the 1D Weir Module

The zone occupied by a weir in stream needs to be represented by a 2D mesh. It is achieved as follows. The horizontal extent of the weir is represented by a quadrilateral polygon, named weir polygon in this report, and the weir polygon has 2D mesh cells within. The weir polygon is formed by creating a pair of mesh lines: the upstream side of the weir and the downstream. The two mesh lines are defined using the “nodestring” within SMS. Each mesh line may consist of a number of mesh nodes; but a minimal of two nodes is required. The 2D mesh cells within a weir polygon are automatically set up as “inactive” by the model, as 2D flows within the polygon are not simulated and results are, therefore, not displayed in the model output. Two sides of the weir polygon are assumed to be connected with straight lines and are set up as “symmetry” boundary conditions with respect to flows outside the weir zone. Bed elevation on the weir polygon is used by the 2D model and is important; but elevations inside the polygon is irrelevant as mesh inside is not used.

The 1D weir module computes the total flow rate (or discharge, Q_w in m^3/s) over a weir using the following equation:

$$Q_w = C_s C_w \sqrt{g} L_w (Z_U - Z_C)^{3/2} \quad (3.1)$$

where Z_U = average water surface elevation (WSE) along the upstream line of the weir (meter); Z_C = average crest elevation of the weir (meter); C_w = dimensionless discharge coefficient for free (unsubmerged) weir flow when downstream water elevation (Z_D) is lower than Z_C (C_w is usually 0.54; but 0.67 may be used for semicircular shaped drop inlet crests); g = gravity acceleration (meter/s²); L_w = lateral length of the weir (meter); and C_s = weir submergence factor which is non-unity if the flow over the weir is affected by tailwater (dimensionless). Submergence coefficient C_s is calculated by:

$$C_s = \left[1 - \left(\frac{Z_D - Z_C}{Z_U - Z_C} \right)^a \right]^b \quad (3.2)$$

where Z_D = average downstream water surface elevation (WSE) (meter), and a , b = dimensionless submergence factor coefficients. Note that $C_s = 1$ if $Z_D \leq Z_C$. By setting Z_D less than Z_C , users may force the weir to be free. Unless altered by users, the default values of weir coefficients C_w , a and b may be used and they are tabulated in Table 1. In addition, users have the option to use the total head ($= WSE + \frac{V^2}{2g}$) to replace Z_U in computing the discharge in Equations (3.1) and (3.2).

Table 1. Default Submerged Weir Coefficients^a (FHWA, 2002)

<i>Weir segment description</i>	C_w	a_{sub}	b_{sub}
Undefined	0.544	16.4	0.432
Paved roadway	0.544	16.4	0.432
Gravel roadway	0.544	15.4	0.608
Single railroad track	0.577	7.25	0.500
Double railroad track	0.520	7.25	0.500
Sharp-crested weir	0.544	1.5	0.385
Broad-crested weir	0.544	7.25	0.500

^aWeir coefficients are based on formulas and charts presented by King and Brater (1963, page 5-19) and Hulsing (1976, page 27).

With the Partial-Interface mode of using SRH-2D, users need to enter the following information:

- Define two nodestrings on the 2D mesh which are designated as “WEIR” when prompted for the boundary type;
- Define each weir by entering a pair of nodestring IDs; and
- Select and enter for each weir the weir type, crest elevation (Z_C), weir lateral length (L_w), and downstream water elevation (Z_D) if the downstream weir is located at an external boundary; otherwise, Z_D is computed by the model and no user input is necessary.

3.1.2. A Sample Case Running 1D Weir Module

The 1D weir module may be used to model flows over: (a) weirs; (b) roadway embankments; (c) overtopped bridge deck; and (d) drop structure when it is in one of the three modes (i.e., the “weir” mode). A simple example is described next to illustrate on how to use the 1D weir module with the Partial-Interface mode of SRH-2D.

The case has a 2D mesh shown in Figure 1 with the weir location identified. Views of the weir polygon and the geometry are shown in

Figure 2. The simulated open channel has a slope of 0.49%, Manning’s coefficient of 0.03, upstream discharge of 50 m³/s and downstream water stage of 5 m. The weir polygon has a size of 2.4 m by 2.4 m as shown in

Figure 2a (blue zone); the polygon is defined by the upstream and downstream weir nodestrings (in red). Input weir parameters include: crest elevation $Z_C = 3.0$ m; weir lateral length $L_w = 2.4$ m; and weir type “SHARP” for sharp-created weir (see Table 1). Weir downstream is internal and Z_D is not needed.

The simulated model results are shown in Figure 3. Simulated flow with velocity vector and magnitude with the 1D weir flow case are displayed.

The input file, `_SIF.dat`, to run the sample case is listed in Figure 4. The case is in the SRH-2D distribution package so that users may use it as a tutorial case.

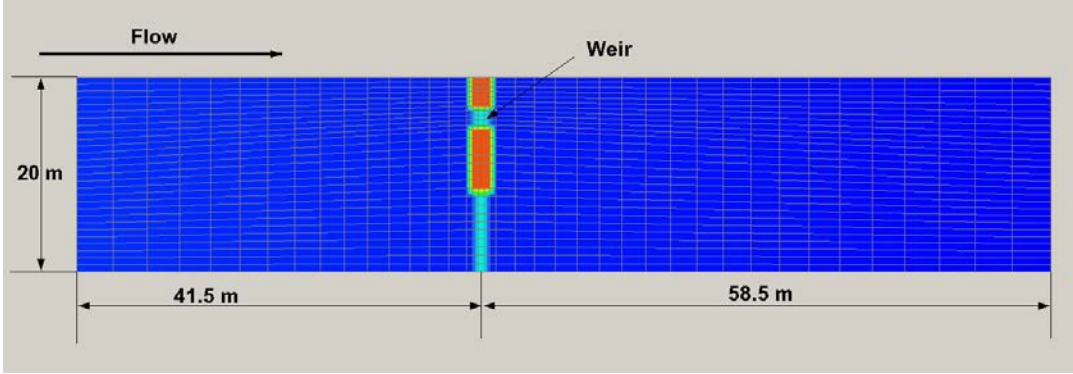


Figure 1. Solution domain and 2D mesh of the sample weir flow case

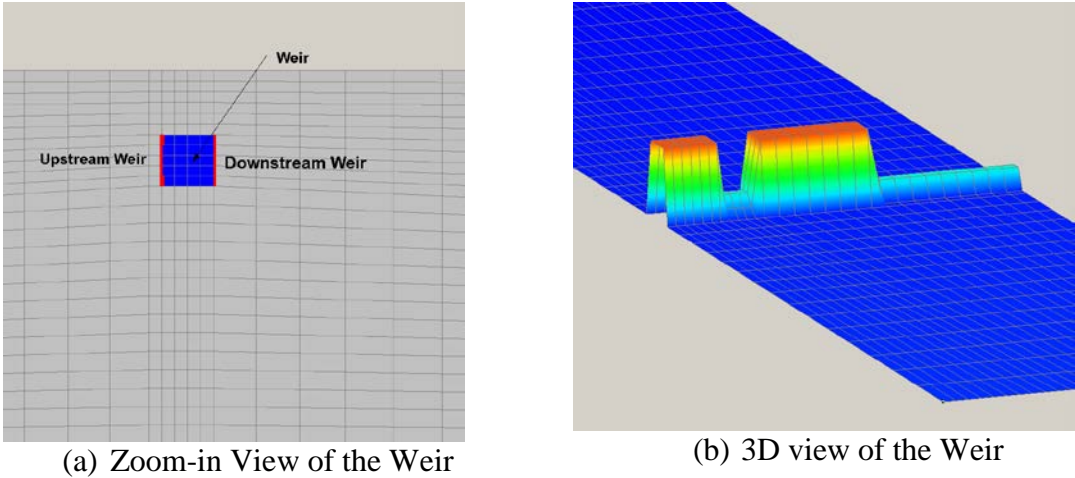


Figure 2. Zoom-in view of the weir

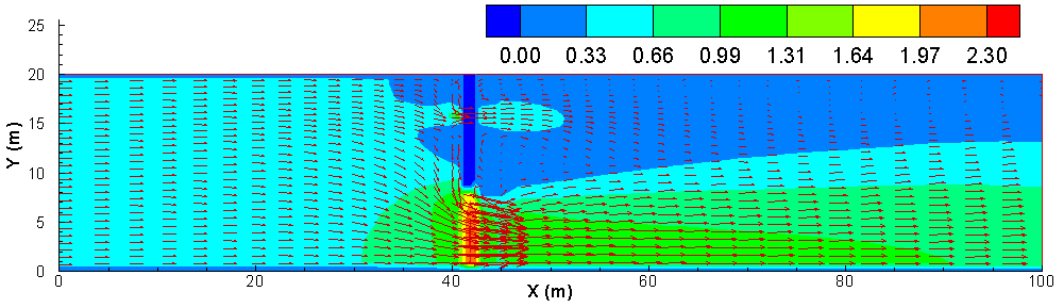


Figure 3. Simulated flow with velocity vector and magnitude with the 1D weir flow case

```

// Simulation Description (not used by SRH-2D):
test
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT
1
// Monitor Point Coordinates: x1 y1 x2 y2 ...
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
// A_TURB for the PARA Model (0.05 to 1.0)
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
// Boundary Values (Q W QS TEM H_rough etc)
50 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
// Boundary Values (Q W QS TEM H_rough etc)
5.0 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Boundary Type (INLET-Q EXIT-H etc)
weir
// Boundary Type (INLET-Q EXIT-H etc)
weir
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream WEIRS:
1
// Structure WEIR: Nodestring ID Pairs of each weir (Toe_ID Top_ID ...)
4 5
// Structure: Weir Parameters ZC ZD LW UNIT TYPE [Cw] [a] [b]
3.0 -999 2.4 SI SHARP
// Number of Obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used

// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1

```

Figure 4. The SIF file used to run the 1D weir case

4.0 Gate Module

Gate structures are movable barriers that can be used to control the flow of water through or over a dam or embankment. 1D empirical equations have been developed to compute flow rates (discharges) through gates; they are incorporated into SRH-2D. Two types of gates may be simulated with the 1D gate module: (a) underflow gates, and (2) overflow gates. Underflow (or submerged) gates act as orifices such as vertical lift gates and radial gates; while overflow (or crest) gates act as weirs such as sector gates, flap gates and roller gates.

4.1.1. Theory of the 1D Gate Module

The gate module first checks the upstream water surface elevation (Z_U) against the top elevation of the gate opening. The top elevation of the gate opening is assumed to be ($Z_C + H_g$), where Z_C is the bottom sill (crest) elevation and H_g is the height of the gate opening. If Z_U is below ($Z_C + H_g$), the gate is non-submerged and it serves as an overflow gate. The discharge of an overflow gate is computed the same way as the weir equations (3.1) and (3.2). That is, the discharge (m^3/s) is calculated by:

$$Q_g = C_s C_g \sqrt{g} W_g (Z_U - Z_C)^{3/2} \quad (4.1)$$

$$C_s = \left[1 - \left(\frac{Z_D - Z_C}{Z_U - Z_C} \right)^a \right]^b \quad (4.2)$$

where C_g = overflow gate discharge coefficient (dimensionless); W_g = width of the overflow gate (meter); Z_U = upstream water-surface elevation (meter); and Z_C = crest or sill elevation of the overflow gate (meter). The downstream submergence coefficient C_s is computed the same way as the weir equation if $Z_D > Z_C$, where Z_D = average downstream water surface elevation (meter).

If Z_U is above ($Z_C + H_g$), the gate is assumed to be underflow (submerged or orifice flow), and the discharge (m^3/s) is calculated as (Novak et al. 1996, pages 232-233):

$$Q_g = C_g W_g H_g \sqrt{2g(Z_U - Z_C)} \quad (4.3)$$

$$C_g = C_{gc} / \sqrt{1 + C_{gc} \frac{H_g}{Z_U - Z_C}} \quad (4.4)$$

$$C_{gc} = 1 - 0.75 \frac{\theta}{90} + 0.36 \left(\frac{\theta}{90} \right)^2 \quad (4.5)$$

where C_g and C_{gc} are discharge coefficient and contraction coefficient, respectively (dimensionless); W_g = width of the underflow gate opening (meter); H_g = height of the underflow gate opening (height above Z_C) (meter); Z_U = upstream water-surface elevation (meter); Z_C = bottom crest elevation of the gate opening (meter); and θ = angle in degree of the gate inclination. The contraction coefficient (C_{gc}) is due mainly to the gate inclination angle, and can either be an input by the user or computed by the above equation using the user-supplied gate inclination angle (θ). The above formulas were developed for gates having planar and cylindrically shaped skin plates (Tainter gates) with free flow conditions and a downstream horizontal apron. The gate inclination angle is defined to be the angle between the outflow edge of the gate and the flow direction ($\theta < 90^\circ$ and varies according to the gate position), as illustrated in Figure 5.

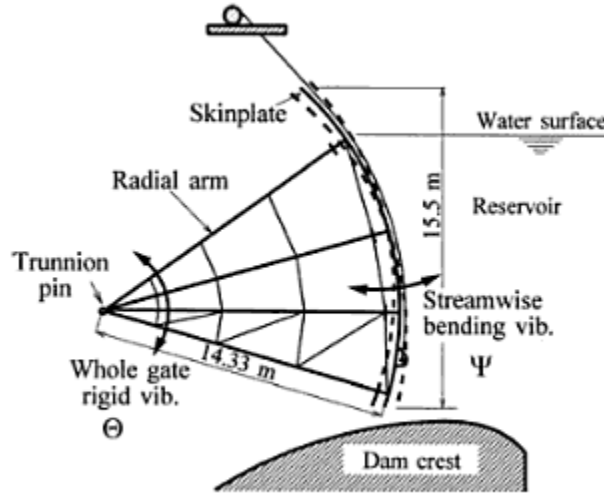


Figure 5. Illustration of the gate inclination angle

If the submerged flow is not free, i.e., downstream water elevation (Z_D) is higher than the gate crest elevation (Z_C), the above free flow discharge is multiplied by the same submergence coefficient, C_s , as the one used by the weir equation.

Users have the option to use the total head ($= WSE + \frac{V^2}{2g}$) to replace Z_U the above equations in computing the discharge through or over the gate.

Numerical representation of a gate is the same as a weir. That is, two nodestrings are used to form a polygon to define a gate structure horizontal extent: one upstream mesh line and another downstream line. The gate structure extent is represented by the mesh polygon and mesh cells within the polygon are deactivated. In SRH-2D implementation, gates are more general than weirs since the algorithm of gates recovers to weir equations if H_g is made to be infinitely high. Input parameters are similar to the weir module but more input parameters are needed follows:

- Define two nodestrings on the 2D mesh which are designated as “GATE” when prompted for the boundary type;
- Define each gater by entering a pair of nodestring IDs; and
- Select and enter for each gate the following parameters: (1) crest elevation (Z_C), (2) downstream water surface elevation (Z_D) if the downstream gate is located at an external boundary (otherwise, Z_D is computed by the model and no user input is necessary); (3) height of the gate opening (H_g); (4) width of the gate (W_g); (5) the contraction coefficient (C_{gc}); and (6) gate type when it is an overflow.

4.1.2. A Sample Case of 1D Gate Module

The gate module is illustrated using a simple underflow case; it illustrates on how to use the 1D gate module with the Partial-Interface mode of SRH-2D.

The flow case has its solution domain and 2D mesh the same as the weir case in Figure 1. The weir is simply replaced by the gate. Also, the bed elevation downstream of the gate is 2 meter lower than the weir case in order to create a free gate flow condition. A zoom-in view of the case is shown in Figure 6 (top of the gate is not shown). The average channel slope is 2.4% from upstream to downstream; the Manning’s coefficient is 0.03; the upstream discharge of 50 m³/s and downstream water stage of 2.2 meters are imposed. The gate zone is a 2.4 meters by 2.4 meters as shown in Figure 2a (blue zone); it is defined by upstream and downstream gate nodestrings (in red). The gate input parameters include: gate sill/crest elevation $Z_C = 3.0$ meters; height of gate opening $H_g = 1.0$ meter; width of the gate $W_g = 2.4$ meters; the contraction coefficient (C_{gc}) = 0.55, and weir type “SHARP” for sharp-created weirs (see Table 1). Note the weir properties are used only if the flow becomes an overflow.

The model predicted velocity field is shown in Figure 7; and the input file, _SIF.dat file, to run the model is listed in Figure 8.

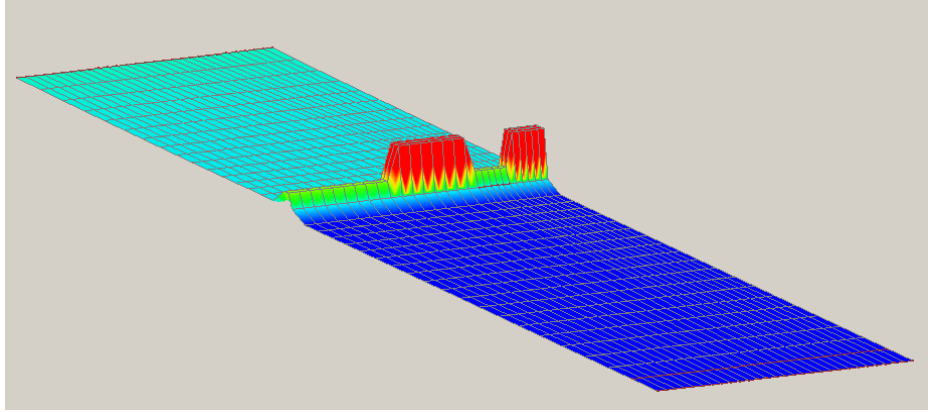


Figure 6. A zoom-in view of the gate (flow is from top left to bottom right)

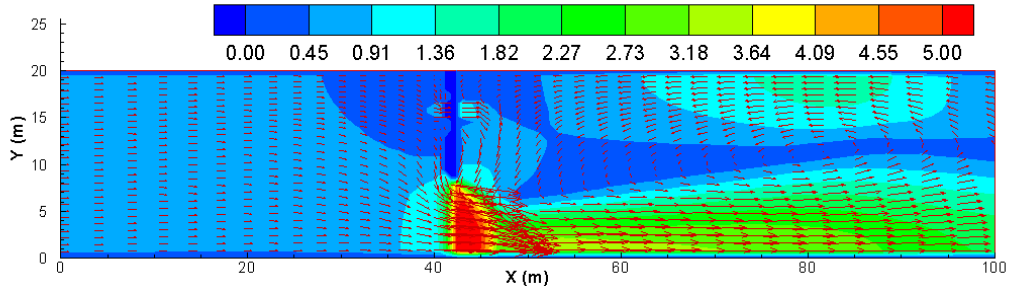


Figure 7. Simulated flow with velocity vector and magnitude with the 1D gate module

```

// Simulation Description (not used by SRH-2D):
Gate Test
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT
1
// Monitor Point Coordinates: x1 y1 x2 y2 ...
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
// A_TURB for the PARA Model (0.05 to 1.0)
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
// Boundary Values (Q W QS TEM H_rough etc)
50 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
// Boundary Values (Q W QS TEM H_rough etc)
2.2 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Boundary Type (INLET-Q EXIT-H etc)
gate
gate
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream GATES
1
// Structure GATE: Nodestring ID Pairs of each gate (Toe_ID Top_ID ...)
4 5
// Structure: Gate Parameters ZC ZD Hg Wg UNIT Cgc TYPE [CW] [a] [b]
3.0 -999 1.0 2.4 SI 0.55 SHARP
// Number of obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used
// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1

```

Figure 8. The SIF file used to run the 1D gate case

5.0 Culvert and Bridge Module

Culverts are short conduits that are used to convey water under roadways, railroads, canals, or other flow obstructions. If a detailed and accurate description of the flow within or in the vicinity of a culvert is not needed, a culvert may be modeled adequately by the 1D approach described below. Small bridges may also be modeled using the same 1D culvert module if the width of the bridge opening is relatively small. 2D approach may be used for more accurate simulation of flows within a culvert or through a bridge using the pressurized flow module described in Chapter 6.0.

Two 1D culvert modules are available from SRH-2D: FST or HY-8. The FST option follows the equations presented by FHWA (2002) and discussed in detail below. The HY-8 option links SRH-2D to the HY-8 module. HY-8 is a special program developed by FHWA to compute discharges through culverts given the upstream and downstream water elevations using a number of polynomial equations. Culvert-specific parameters are obtained using HY-8 program. The HY-8 and SMS Users' Manual should be consulted on how to use HY-8 to design culverts and to simulate flows with SRH-2D. In the following, the FST option is described in details. Use of HY-8 is only illustrated with an example.

5.1.1. Theory of FST Culvert Module

The FST culvert module adopts the approach implemented in FESWMS and FLO2DH as described in FHWA (2002). In general, a culvert flow is controlled by the characteristics of either the inlet or the outlet of the culvert, as illustrated in Figure 9. Under the inlet control condition, culverts have shallow, high-speed supercritical flow immediately downstream from their inlets. Hydraulic jumps may form within barrels depending on tailwater depth. Culverts will never flow full throughout their lengths when inlet conditions control, behaving like orifices when their inlets are submerged, or like weirs when they are not. Under the outlet control condition, culverts either flow full for their entire lengths, or flow in subcritical states without being completely filled. Outlet control flow is calculated based on a straightforward energy balance between the upstream and downstream ends of a culvert.

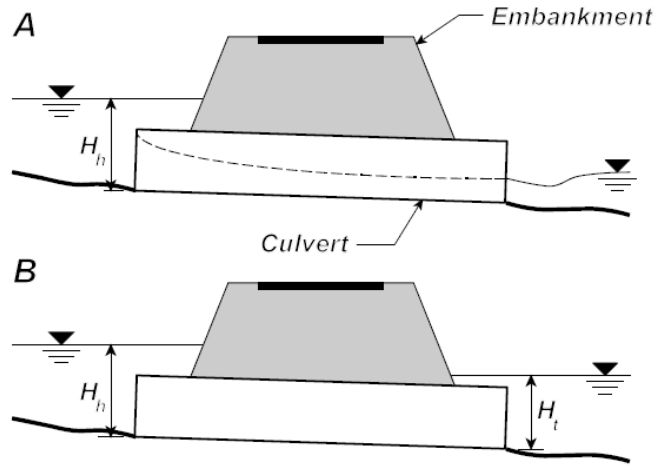


Figure 9. A culvert flow that is either under inlet control (A) or outlet control (B)

In model implementation, control condition is determined by SRH-2D automatically depending on the local flow. Both conditions are evaluated and the one that yields the smaller flow rate through the culvert is selected, giving the headwater and tailwater depths. The discharge (m^3/s) through a culvert is computed by the following equation:

$$Q_c = N_c C_c A_c \sqrt{2gH_c} \quad (5.1)$$

where N_c = number of identical barrels; C_c = discharge coefficient depending on the flow control (inlet or outlet) and other culvert characteristics; A_c = full cross-section area of the culvert barrel (m^2); g = gravitational acceleration (m/s^2); and H_c = culvert head (m).

For inlet control flow, downstream head is irrelevant and not used, and the inlet may be submerged or non-submerged. The two parameters in equation (5.1) are computed as:

$$H_c = Z_U - Z_I \quad (5.2)$$

$$C_c = \text{Min} \left\{ \sqrt{\frac{1 - (Y + mS_o) \frac{D_c}{H_c}}{2c'}}, \frac{1}{\sqrt{2K'^{1/M}}} \left(\frac{H_c}{D_c} \right)^{\frac{1-1}{2}} \right\} \quad (5.3)$$

where Z_U = water surface elevation at the upstream end of the culvert (that is, the headwater elevation); Z_I = invert elevation at the culvert entrance; D_c = interior height of the culvert barrel; S_o = culvert barrel slope; $m = 0.7$ for mitered inlets and -0.5 for all other inlets; and K' , M , c' , Y = culvert coefficients that depend on culvert material, barrel cross section shape, and inlet characteristics. Culvert coefficients, K' , M , c' , Y , for various culvert barrel material and inlet combinations are summarized in Table 2. In the

above, the first inlet control discharge coefficient is used when the culvert entrance is submerged and functions like an orifice; the second coefficient applies when the entrance is not submerged and acts as a weir.

For the outlet control flow, the downstream head impacts the flow rate and the two parameters in Equation (5.1) are computed by:

$$H_c = Z_U - Z_D \quad (5.4)$$

$$C_c = \frac{1}{\sqrt{1 + K_e + \frac{2gn_c^2 L_c}{R_c^{4/3}}}} \quad (5.5)$$

where Z_U = water-surface elevation at the upstream end of the culvert (that is, the headwater elevation) (m), Z_D = water-surface elevation at the downstream end of the culvert (that is, tailwater elevation) (m); K_e is entrance loss coefficients that may be taken from Table 3 for various culverts (dimensionless); L_c = barrel length (m); n_c = Manning's roughness coefficient of the culvert barrel that may be determined from Table 4; and R_c = the hydraulic radius of a single culvert barrel (thus is a user input at present) (m). Note that the hydraulic radius is the radius of the culvert cross sections when the flow is full. Under some circumstances that the flow is not full within the culvert, R_c is different.

The above set of equations is only the flow rate through the culvert; no overtop is considered. If flow is expected to overtop the crest of the culverts, additional input parameters are required. Specifically, top crest elevation (Z_T) of the culvert or bridge deck needed. Whenever the upstream water surface elevation (Z_U) is higher than Z_T , an additional overflow discharge is computed using the 1D weir equations discussed in Chapter 3 and the overflow discharge is added to the culvert underflow discharge to arrive at the total flow rate. The overtop scenario may occur for flood modeling through culverts and bridges.

Numerical representation of culverts/bridges is the same as weirs and gates. That is, a culvert/bridge is represented by a mesh polygon represented by two nodetsrings: one upstream mesh line and the other downstream line. The polygon such formed is used to represent one or a number of identical culvert barrels. The culvert mesh polygon is set as "inactive" by SRH-2D and no results will be displayed in the polygon.

Table 2. Culvert Inlet Flow Formula Coefficients (Source: FHWA 2002)

Barrel material	Barrel shape	Inlet description	Coefficients ^b			
			K'	M	c'	Y
Concrete	Circular	Headwall; square edge	0.3153	2.0000	1.2804	0.6700
Concrete	Circular	Headwall; gooved edge	0.2509	2.0000	0.9394	0.7400
Concrete	Circular	Projecting; grooved edge	0.1448	2.0000	1.0198	0.6900
Cor. metal	Circular	Headwall	0.2509	2.0000	1.2192	0.6900
Cor. metal	Circular	Mitered to slope	0.2112	1.3300	1.4895	0.7500
Cor. metal	Circular	Projecting	0.4593	1.5000	1.7790	0.5400
Concrete	Circular	Beveled ring; 45 deg bevels	0.1379	2.5000	0.9651	0.7400
Concrete	Circular	Beveled ring; 33.7 deg bevels	0.1379	2.5000	0.7817	0.8300
Concrete	Rectangular	Wingwalls; 30 to 75 deg flares; square edge	0.1475	1.0000	1.2385	0.8100
Concrete	Rectangular	Wingwalls; 90 and 15 deg flares; square edge	0.2242	0.7500	1.2868	0.8000
Concrete	Rectangular	Wingwalls; 0 deg flares; square edge	0.2242	0.7500	1.3608	0.8200
Concrete	Rectangular	Wingwalls; 45 deg flare; beveled edge	1.6230	0.6670	0.9941	0.8000
Concrete	Rectangular	Wingwalls; 18 to 33.7 deg flare; beveled edge	1.5466	0.6670	0.8010	0.8300
Concrete	Rectangular	Headwall; 3/4 in chamfers	1.6389	0.6670	1.2064	0.7900
Concrete	Rectangular	Headwall; 45 deg bevels	1.5752	0.6670	1.0101	0.8200
Concrete	Rectangular	Headwall; 33.7 deg bevels	1.5466	0.6670	0.8107	0.8650
Concrete	Rectangular	Headwall; 45 deg skew; 3/4 in chamfers	1.6611	0.6670	1.2932	0.7300
Concrete	Rectangular	Headwall; 30 deg skew; 3/4 in chamfers	1.6961	0.6670	1.3672	0.7050
Concrete	Rectangular	Headwall; 15 deg skew; 3/4 in chamfers	1.7343	0.6670	1.4493	0.6800
Concrete	Rectangular	Headwall; 10-45 deg skew; 45 deg bevels	1.5848	0.6670	1.0520	0.7500
Concrete	Rectangular	Wingwalls; non-offset 45° flares; 3/4 in chamfers	1.5816	0.6670	1.0906	0.8030
Concrete	Rectangular	Wingwalls; non-offset 18.4° flares; 3/4 in chamfers	1.5689	0.6670	1.1613	0.8060
Concrete	Rectangular	Wingwalls; non-offset 18.4° flares; 30° skewed barrel	1.5752	0.6670	1.2418	0.7100
Concrete	Rectangular	Wingwalls; offset 45° flares; beveled top edge	1.5816	0.6670	0.9715	0.8350
Concrete	Rectangular	Wingwalls; offset 33.7° flares; beveled top edge	1.5752	0.6670	0.8107	0.8810
Concrete	Rectangular	Wingwalls; offset 18.4° flares; top edge bevel	1.5689	0.6670	0.7303	0.8870
Cor. metal	Rectangular	Headwall	0.2670	2.0000	1.2192	0.6900
Cor. metal	Rectangular	Projecting; thick wall	0.3023	1.7500	1.3479	0.6400
Cor. metal	Rectangular	Projecting; thin wall	0.4593	1.5000	1.5956	0.5700
Concrete	Horizontal ellipse	Headwall; square edge	0.3217	2.0000	1.2804	0.6700
Concrete	Horizontal ellipse	Headwall; grooved edge	0.1379	2.5000	0.9394	0.7400
Concrete	Horizontal ellipse	Projecting; gooved edge	0.1448	2.0000	1.0198	0.6900
Concrete	Horizontal ellipse	Headwall; square edge	0.3217	2.0000	1.2804	0.6700
Concrete	Horizontal ellipse	Headwall; grooved edge	0.1379	2.5000	0.9394	0.7400
Concrete	Horizontal ellipse	Projecting; gooved edge	0.3056	2.0000	1.0198	0.6900
Cor. metal	Pipe arch (18" corner)	Headwall	0.2670	2.0000	1.5956	0.5700
Cor. metal	Pipe arch (18" corner)	Mitered to slope	0.1702	1.0000	1.4895	0.7500
Cor. metal	Pipe arch (18" corner)	Projecting	0.4593	1.5000	1.5956	0.5300
Structural plate	Pipe arch (18" corner)	Projecting	0.3998	1.5000	1.5667	0.5500
Structural plate	Pipe arch (18" corner)	Headwall; square edge	0.2799	2.0000	1.1613	0.6600
Structural plate	Pipe arch (18" corner)	Headwall; beveled edge	0.0965	2.0000	0.8493	0.7500
Structural plate	Pipe arch (31" corner)	Projecting	0.3998	1.5000	1.5667	0.5500
Structural plate	Pipe arch (31" corner)	Headwall; square edge	0.2799	2.0000	1.1613	0.6600
Structural plate	Pipe arch (31" corner)	Headwall; beveled edge	0.0965	2.0000	0.8493	0.7500
Cor. metal	Arch	Headwall	0.2670	2.0000	1.2192	0.6900
Cor. metal	Arch	Mitered to slope	0.9651	2.0000	1.4895	0.7500
Cor. metal	Arch	Projecting	0.4593	1.5000	1.5956	0.5700
Concrete	Circular	Tapered throat	1.3991	0.5550	0.6305	0.8900
Cor. metal	Circular	Tapered throat	1.5760	0.6400	0.9297	0.9000
Concrete	Rectangular	Tapered throat	1.5116	0.6670	0.5758	0.9700

^aFrom Norman et al. (1985, Table 9, pages 147-148)

^bCoefficients K' and c' have been made non-dimensional and, therefore, differ from the coefficients K and c given by Norman et al. (1985).

Table 3. Entrance loss coefficient for various culvert types and entrance conditions under the outlet control culvert flow

Type of culvert	Entrance description	Entrance loss coefficient ^a , K_e
Concrete pipe	Projecting from fill, grooved end	0.2
	Projecting from fill, square-cut end	0.5
	Headwall or headwall with wingwalls (concrete or cement sandbags)	
	Grooved pipe end	0.2
	Square-cut pipe end	0.5
	Rounded pipe end	0.1
	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
With grate	0.7	
Corrugated metal pipe or pipe-arch	Projecting from embankment (no headwall)	0.9
	Headwall with or without wingwalls (concrete or cement sandbags)	0.5
	Mitered end that conforms to embankment slope	0.7
	Manufactured end section of metal or concrete that conforms to embankment slope	
	Without grate	0.5
With grate	0.7	
Reinforced concrete box	Headwall parallel to embankment (no wingwalls)	
	Square-edged on three sides	0.5
	Rounded on three sides to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 30° to 75° to barrel	
	Square-edged at crown	0.4
	Crown edge rounded to radius of 1/12 of barrel dimension	0.2
	Wingwalls at 10° to 30° to barrel	
	Square-edged at crown	0.5
Wingwalls parallel to embankment		
Square-edged at crown	0.7	

^aSummarized from entrance loss coefficients given by Norman et al. (1985) and “Roadway and traffic design standards” (1998, Index No. 249 “Pipe end treatment application guide”).

Table 4. Culvert Manning roughness coefficients for various culvert barrel materials and barrel conditions under the outlet control culvert flow

Culvert barrel material	Entrance description	Manning roughness coefficient, n_c
Concrete	Good joints, smooth walls	0.012
	Projecting from fill, square-cut end	0.015
	Poor joints, rough walls	0.017
Corrugated metal or structural plate corrugated metal	2-2/3 inch \times 1/2 inch corrugations	0.025
	6 inch \times 1 inch corrugations	0.024
	5 inch \times 1 inch corrugations	0.026
	3 inch \times 1 inch corrugations	0.028
	6 inch \times 2 inch corrugations	0.034
	9 inch \times 2-1/2 inch corrugations	0.035

5.1.2. A Culvert Flow Case without Overtop

A simple case is used to illustrate how to use the 1D FST culvert module by working with the Partial-Interface mode of SRH-2D. The culvert flow is outlet controlled and without the overtop flow.

The solution domain and the 2D mesh for the culvert test case are the same as the weir example shown in Figure 1. The weir is simply replaced by a culvert. The bed elevation of the solution domain is the same as the gate example in Chapter 4 except that the culvert barrel entrance invert elevation is set to 0.0 meter. A zoom-in view of the case is shown in Figure 10 (the upper portion of the culvert is not shown). The open channel flow parameters are as follows: the average channel slope is 2.4% from upstream to downstream boundaries; the channel Manning's coefficient is 0.03; and an upstream discharge of 50 m³/s and downstream water stage of 2.2 meters are imposed. The culvert input parameters are as follows. There are two barrels; each culvert barrel has a length of 3.0 meters, interior height of 1.0 meter, and cross sectional area of 1.2 m². The culvert invert elevation at the culvert entrance is $Z_I = 0.0$ meter; the hydraulic radius of a single culvert barrel $R_c = 0.375$ meter; culvert barrel slope $S_o = 0.1\%$; number of identical barrels $N_c = 2$; $m = 0.7$; culvert coefficients K' , M , c' and $Y = 0.1475$, 1.0, 1.2385 and 0.81 (based on in Table 2); entrance loss coefficients $K_e = 0.4$ (from Table 3); and Manning's roughness coefficient of the culvert barrel $n_c = 0.015$ (Table 4).

The _SIF.dat file to run the model is listed in Figure 11 and sample results are shown in Figure 12.

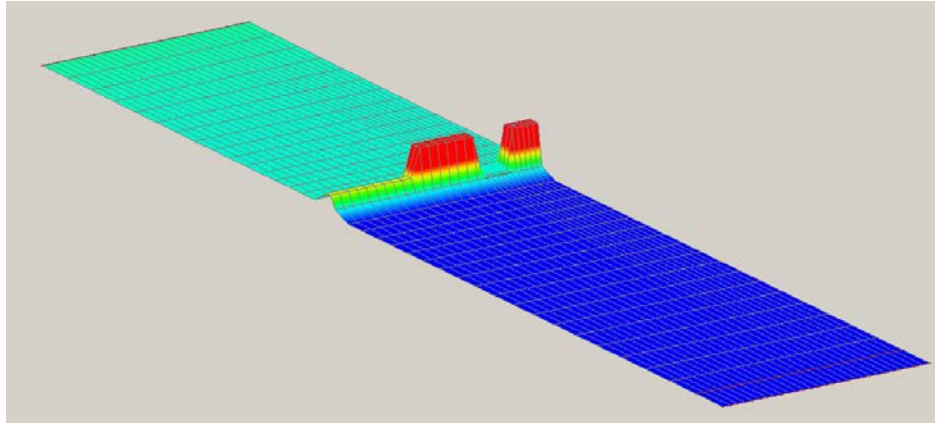


Figure 10. A zoom-in view of the culvert (flow is from top left to bottom right)

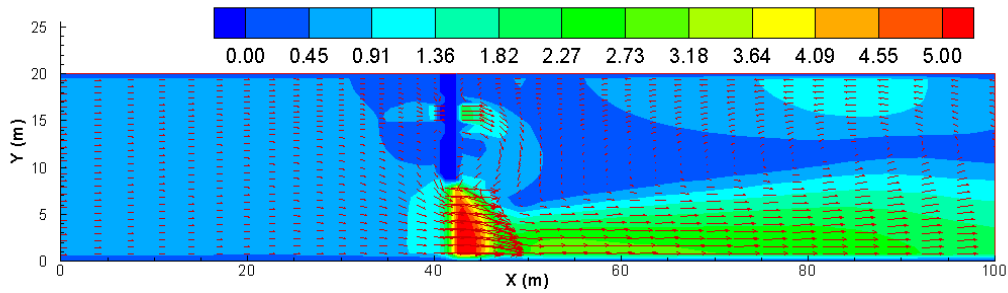


Figure 11. Simulated flow with velocity vector and magnitude for the sample case without overtop using the 1D FST culvert module

```

// Simulation Description (not used by SRH-2D):
Culvert Test
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT
1
// Monitor Point Coordinates: x1 y1 x2 y2 ...
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
// A_TURB for the PARA Model (0.05 to 1.0)
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
50 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
2.2 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Boundary Type (INLET-Q EXIT-H etc)
culvert
culvert
//
monitor
monitor
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream Culverts
1
// Structure CULVERT: Nodestring ID Pairs of each culvert (Toe_ID Top_ID ...)
4 5
// Structure: CULVERT 14 Parameters: ZI Dc Lc Ac Rc Slp UNIT Nc m_in K_p M C_p Y Ke nc
0.0 -999 1.0 3.0 1.2 0.375 0.001 SI 2 0.7 0.1475 1.0 1.2385 0.81 0.4 0.015
// Flow Obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used
// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1

```

Figure 12. The SIF file used to run the 1D culvert case without overtop

5.1.3. A Bridge Flow Case with Overtop

The 1D FST module is used to simulate the scenario of overtopped bridge flow next. The sample case illustrates how to use the 1D FST culvert module with the Partial-Interface mode of SRH-2D for bridge overtopping modeling.

The geometry, solution domain and 2D mesh are displayed in Figure 13 (bridge top deck is not shown). The bridge is located on the right side at $x=41.5$ m and has the following parameters: lateral width = 8.61 m; height from bed to ceiling = 4.0 m; longitudinal

length = 3 m; elevation of the bridge lower deck (bridge ceiling) = 5 m; elevation of the upper deck (for overtop) = 7 m; and bed elevation underneath the bridge is 1.0 m. Left side of the bridge is an embankment with elevation of 10 m. The open channel has a slope of 2.412%, Manning's coefficient of 0.03, upstream discharge of 200 m³/s and downstream water stage of 6 m. Model input parameters for the bridge are based on the culvert table in the above and they include: height of the bridge $D_c = 4.0$ m; bridge length $L_c = 3.0$; full cross-section area $A_c = 34.44$ m²; hydraulic radius of the bridge opening $R_c = 1.37$ m; bed slope underneath the bridge $S_o = 0.0$; number of identical opening $N_c = 1$; $m = 0.7$; culvert coefficients K' , M , c' and $Y = 0.2242$, 0.75, 1.2868 and 0.8 (based on Table 2); entrance loss coefficients $K_e = 0.4$ (can be from Table 3); and Manning's roughness coefficient of the culvert barrel $n_c = 0.015$ (Table 4).

The model predicted flow velocity field is shown in Figure 14, and the _SIF.dat file to run the model is listed in Figure 15. Two additional cases have also been carried out with the same mesh and terrain: one is the open channel flow without the bridge (Open Channel Case) and another is a case without overtop occurring (bridge top elevation is set high enough so that no bridge overtop occurs) (No-Overtop Bridge Case). The predicted water elevation near model inlet is predicted to be 7.561 m, 8.645 m, and 9.312 m, respectively, for the Open Channel, Overtop Bridge and No-Overtop Bridge cases. It clearly shows the backwater effect caused by the bridge. It also makes sense that the largest backwater is due to the No-Overtop scenario.

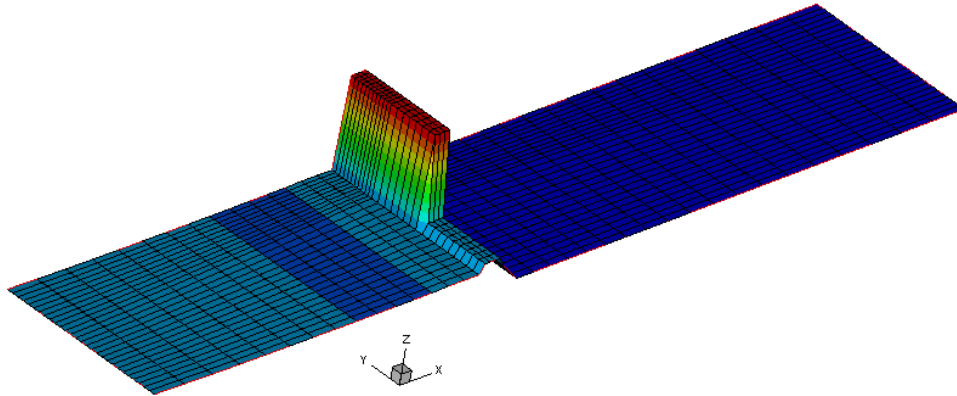


Figure 13. Geometry and mesh of the bridge case (flow is from left to right)

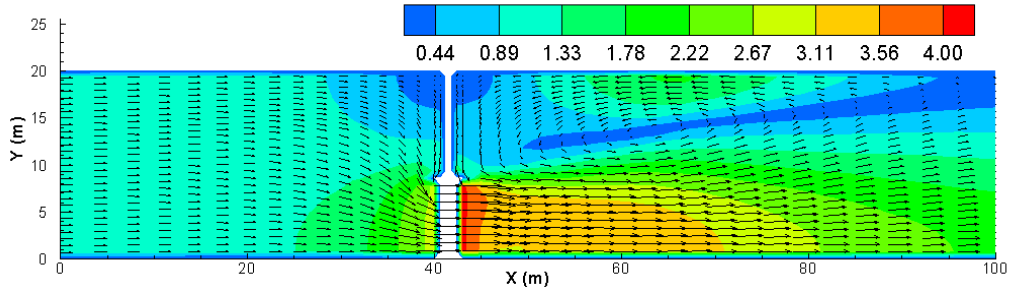


Figure 14. Simulated flow with velocity magnitude and vector with the overtopped bridge 1D model

```
// Simulation Description (not used by SRH-2D):
Bridge Test: Overtopped Flow
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT // Monitor Point Coordinates: x1 y1 x2 y2 ...
1
7.2 10.2
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 0.05 1.1
// Turbulence-Model-Selection(PARA or KE); // A_TURB for the PARA Model (0.05 to 1.0)
PARA
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
ZONAL
1
0 0 6
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)
// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
200 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
6.0 SI
// Boundary Type (INLET-Q EXIT-H etc)
culvert
culvert
//
monitor
// Wall-Roughness-Height-Specification (empty-line=DONE)
// Number of In-Stream Culverts
1
// Structure CULVERT: Nodestring ID Pairs of each culvert (Toe_ID Top_ID ...)
3 4
// Structure: CULVERT 16 Parameters: Z_invert WSE_dwn Height_culv Len_longit Area_openning
radius_hydro Slope UNIT NumberofBarrel m_in K_p M C_p Y Ke nc
1.0 -999 4.0 3.0 34.44 1.37 0.0 SI 1 0.7 0.2242 0.75 1.2868 0.8 0.4 0.015
// Overtop Weir parameter: Ztop LW TYPE or Ztop LW USER CW a b
7.0 8.6 PAVED
// Flow Obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used
// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1
```

Figure 15. The SIF file used to run the overtopped bridge 1D model

5.1.4. A Sample Culvert Flow Modeling with HY-8 Module

The 1D culvert modeling with HY-8 module is demonstrated next; the case illustrates how to use the combined modeling using HY-8 and SRH-2D with the Partial-Interface mode. Combined SRH-2D and HY-8 modeling is recommended to use the customized mode of SMS; users may consult the relevant manuals by the Aquveo, LLC.

The solution domain, bathymetry and culver location are shown in Figure 16. The average channel slope is 0.5% from upstream to downstream; the Manning's coefficient is 0.03; and the upstream discharge of 50 m³/s and downstream water stage of 5.0 m (with zero bed elevation) are imposed. The culvert zone is 2.4 m in length and 4.4 m in width; it is defined by the upstream and downstream culvert nodestrings. The culvert was setup using the HY-8 program and a separate input file, `hy8_input.hy8`, was created that contains the culvert related parameters and is used as the input file to HY-8. The `_SIF.dat` file to run the model is listed in Figure 17 while the sample results are shown in Figure 18.

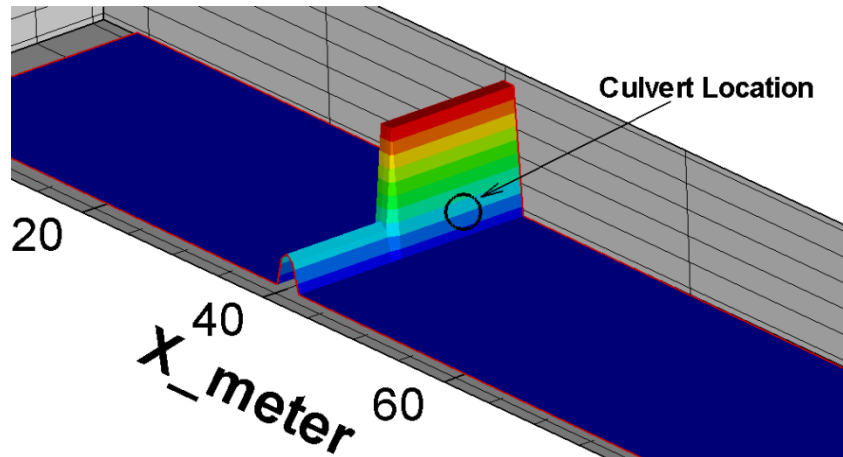


Figure 16. Bathymetry and culvert location (flow is from top left to bottom right)


```

// Simulation Description (not used by SRH-2D):
SRH-2D HY8 Test Case
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT & Then Monitor Point Coordinates: x1 y1 x2 y2 ...
1
99.0000 10.0000
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 2000
// Turbulence-Model-Selection(PARA or KE) & Then A_TURB for the PARA Model
PARABOLIC
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit & THEN // Mesh FILE_NAME and FORMAT(SMS...)
METER
case.srhgeom SRHGEOM
// Manning Roughness Input Method(1=uniform 2=2DM 3=(x y) distributed)
2
// Number of Types & Then n in each mesh zone: a real value or a WD~n file name
1
0.03
// Any-Special-Modeling-Options? (0/1=no/yes)
0
// Boundary Type (INLET-Q EXIT-H etc)
MONITORING
// Boundary Type (INLET-Q EXIT-H etc)
HY8
HY8
// Boundary Type & Then Q UNIT Type
INLET-Q
50 SI CONVEYANCE
// Boundary Type & Then WSE UNIT TYPE
EXIT-H
5 SI C
// Wall-Roughness-Height-Specification (empty-line=DONE)

// HY8 Input File Name for culvert crossings
hy8_input.hy8
// HY8 Executable Location
D:\All-Important\Code\Z_HY8\HY8_Jan27_2015\HY8
// Number of InStream HY8 culvert crossings:
1
// CULVERT Nodestring ID Pairs of each Hy8 culvert (Upstream & Downstream Nodestring IDs)
2 3
// HY8 Culvert Outlet Property: 1=All Internal Outlets; -999=there are external outlets
1
// Number of In-Stream Flow Obstructions:
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN) + Optional STL
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used

// Intermediate Result Output Control
-1

```

Figure 17. The SIF file used to run the HY-8 culvert case

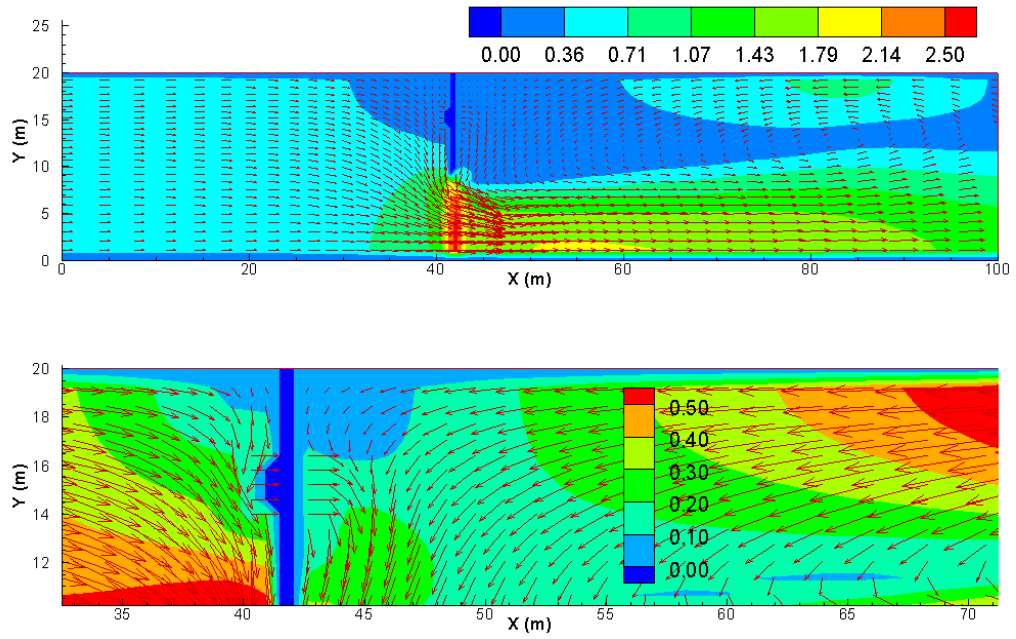


Figure 18. Simulated flow with velocity magnitude and vector the HY-8 culvert case

6.0 2D Pressurized Flow Module

Flow through a bridge or culvert can also be modeled using the 2D pressurized flow module described in this section. Within the pressure flow zone, 2D mesh is used to simulate the pressurized flow in details just like the regular 2D open channel flow modeling. If overtop over the culvert or bridge is to occur, the 1D weir equations discussed in Chapter 3 is used and the 1D weir parameters should also be supplied in the input. SRH-2D treats the pressure flow zone as free surface open channel flows when the water surface is not in contact with (or lower than) the ceiling of the bridge deck or culvert opening (unconfined flow). Pressurized flow equations are automatically turned on if local water elevation is higher than the ceiling elevation. With pressurized flow, extra resistance (energy loss) due to friction by the ceiling may be added; in addition, resistance caused by the bridge piers may also be added through the “flow obstruction” module whether it is an open channel or pressurized flow. Under the pressurized condition, depth-averaged flow equations are modified within the mesh cells where pressure flow occurs. Pressure head rather than depth is computed. The technical details for pressure flow modeling are discussed below based on the approach described in FHWA (2002).

In general, the 2D pressure flow module may be more accurate than the 1D method discussed in Chapter 5. In particular, 2D pressure flow module requires no empirical coefficients related to the flow through the culvert/bridge.

6.1.1. Theory of 2D Pressure Flow Module

With SRH-2D, the horizontal extent of a bridge or culvert opening is represented by a polygon with 2D mesh cells – called pressure flow polygon. The pressure flow through the bridge/culvert is computed with the 2D mesh in the polygon. The pressure flow polygon is defined by two mesh line segments: one represents the upstream of the bridge/culvert and another is the downstream. Each segment is defined within SMS as a “nodestring.” The pressure flow polygon is automatically formed by SRH-2D giving the two nodestrings. Two ceiling elevations are specified for each pressure flow polygon: the upstream ceiling (z_{CU}) and the downstream ceiling (z_{CD}). SRH-2D assumes that the two ceiling elevations are constant along the upstream and downstream nodestrings. The ceiling elevation of the remaining nodes in the polygon is computed through a linear interpolation. A mesh point within the pressure flow polygon is considered to be a pressure flow if the computed water surface elevation is higher than the ceiling elevation. The depth-averaged pressure flow through a bridge or culvert is simulated using the following momentum equations assuming the bed slope is small:

$$\frac{\partial hU}{\partial t} + \frac{\partial hUU}{\partial x} + \frac{\partial hVU}{\partial y} = \frac{\partial hT_{xx}}{\partial x} + \frac{\partial hT_{xy}}{\partial y} - gh \frac{\partial(z_b + P)}{\partial x} - \frac{\tau_{bx} + \tau_{cx}}{\rho} \quad (6.1)$$

$$\frac{\partial hV}{\partial t} + \frac{\partial hUV}{\partial x} + \frac{\partial hVV}{\partial y} = \frac{\partial hT_{xy}}{\partial x} + \frac{\partial hT_{yy}}{\partial y} - gh \frac{\partial(z_b + P)}{\partial y} - \frac{\tau_{by} + \tau_{cy}}{\rho} \quad (6.2)$$

where h is the water depth from bed to ceiling ($h = z_c - z_b$ and z_c is the ceiling elevation), P is the pressure head in the pressure flow zone, and τ_{cx} and τ_{cy} are the x and y components of the shear stress between flowing water and the ceiling, respectively. The ceiling shear stresses are computed by:

$$\begin{pmatrix} \tau_{cx} \\ \tau_{cy} \end{pmatrix} = \rho C_f \sqrt{1 + \left(\frac{\partial z_c}{\partial x}\right)^2 + \left(\frac{\partial z_c}{\partial y}\right)^2} \begin{pmatrix} U \\ V \end{pmatrix} \sqrt{U^2 + V^2}; \quad C_f = \frac{gn_c^2}{h^{1/3}} \quad (6.3)$$

Note that the above dimensionless friction coefficient (C_f) has the same form as that for the bed. The ceiling friction coefficient, n_c , can be made the same as the bed, or can be specified separately. The above equation set is exactly the same as the open channel free surface flow except that (a) h is a constant (from bed to ceiling), not a time varying variable; (b) pressure head P replaces h as the driving force; and (c) extra friction due to ceiling resistance is added.

A number of restrictions should be followed in the pressure flow modeling as follows:

- The pressure flow equation set is strictly valid only if the bridge/culvert bed slope is small; therefore, 1D culvert/bridge module may be used if the bed slope is high.
- Pressure flow zone should be meshed with 2D quadrilateral cells including the cells adjacent to the downstream boundary as overtop flow is added back to the domain along the mesh line parallel to the downstream boundary and the distance is assumed to be the cell size along the flow direction;
- The pressure zone should not be set as “inactive” in SMS;
- Straight side lines are assumed to form the pressure flow polygon since SRH-2D uses the four corners of the upstream and downstream nodestrings to define the polygon;
- In setting up the two nodestrings, order is important. Set up the upstream nodestring first and then the downstream. Direction of each nodestring should point to the flow direction according to the right-side rule.

6.1.2. Bridge/Culvert Parameters

The geometry of a bridge or culvert needs to be specified to simulate adequately the flow through/over the structures. At present, a constant elevation Z_C is used to specify the elevation above which flow overtop occurs. That is, Z_C represents the top elevation of a bridge deck or the crest elevation of the embankment road above the culvert. The crest or the top is assumed flat at present. When the average water surface elevation Z_U of the approaching flow is larger than Z_C along the upstream nodestring, overflow is activated. The flow rate of the overflow is computed using the weir equations discussed in Chapter 3.

The elevation of the bottom half of the bridge/culvert is represented by the 2D mesh in the pressure zone. The half plane is usually located such that the cut plan within the bridge/culvert has the largest horizontal area. The ceiling geometry, the upper half of the of the bridge/culvert may be specified using one of the two options described below. The ceiling elevation determines whether a flow is open or pressurized.

First ceiling option assumes that the ceiling is flat but has a slope from upstream to downstream nodestrings. Only two elevations are provided as user inputs, Z_U and Z_D , which are used to determine the flat ceiling elevation in the pressure zone.

The second option assumes that the ceiling has a parabolic profile across the stream; the same profile is applied along both upstream and downstream nodestrings. Therefore, zero slope is assumed from upstream to downstream. The widths of the upstream and downstream nodestrings should be equal and the pressure zone should be set up as rectangular in shape. The bridge ceiling elevation in the pressure zone is computed by the following parabolic profile:

$$Z = Z_E + \frac{4(Z_M - Z_E)}{W^2} y(W - y) \quad (6.4)$$

where Z_E is the ceiling elevation at the two end points of the upstream nodestring, Z_M is the ceiling elevation at the midpoint of the profile, W is the width of the profile (the distance between the two end points of the upstream nodestring), and y is the distance coordinate from the start point ($y=0$) to the end point ($y=W$) of the upstream nodestring. Z_E and Z_M are the two user inputs.

6.1.3. A Simple 2D Pressure Flow Case without Overtop

The 2D pressure flow module is demonstrated with the same case as the 1D culvert module case in 5.1.2, named “A Culvert Flow Case without Overtop”. The case illustrates how to use the 2D pressure flow module with the Partial-Interface mode of SRH-2D. The case has no overtop flow allowed.

The solution domain and the 2D mesh are the same as the 1D culvert case in Figure 1. The 1D culvert zone now is the pressure flow polygon which has 3.0 m in length and 2.4 m in width as shown in Figure 2a (blue zone). The zone is defined by the upstream and downstream culvert nodestrings (in red). The culvert bottom elevation is set at 0.0 meter throughout and a zoom-in view of the case is the same as shown in Figure 10 (the upper portion of the culvert is not shown). The average channel slope is 2.4% from upstream to downstream; the Manning's coefficient is 0.03; and the upstream discharge is 50 m³/s and downstream water stage is 2.2 m. The input parameters for the pressure flow are: the upstream ceiling elevation $Z_U = 1.0\text{m}$; the downstream ceiling elevation $Z_D = 1.0\text{m}$; and the Manning roughness coefficient due to culvert ceiling $n_c = 0.04$.

The _SIF.dat file to run the model is listed in Figure 19 and sample results are shown in Figure 20.

```

// Simulation Description (not used by SRH-2D):
Pressure Flow Test
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT
1
// Monitor Point Coordinates: x1 y1 x2 y2 ...
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
// A_TURB for the PARA Model (0.05 to 1.0)
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient Entry: a real value or a WD~n file name
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
50 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
2.2 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Boundary Type (INLET-Q EXIT-H etc)
PRESS
PRESS
//
monitor
monitor
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream Pressure Flow Zones:
1
// Pressure Flow ZONE: Nodestring ID Pairs of each zone (Toe_ID Top_ID ...)
4 5
// Structure: Pressure Flow ZONE Parameters: ZC_UP ZC_DN MANNING UNIT
1.0 1.0 0.04 SI
// Number of obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used
// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1

```

Figure 19. The SIF file used to run the pressure flow over a culvert without overtop

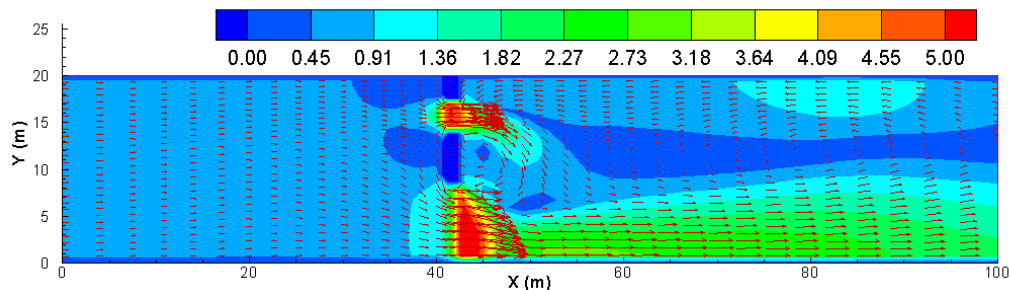


Figure 20. Simulated flow with velocity magnitude and vector with the pressure flow module for the culvert flow simulation

6.1.4. A Simple 2D Pressure Flow Case with Overtop

Next, the 2D pressure flow module is used to simulate a pressurized flow through a bridge with overtop flow. It is to demonstrate the capability to simulate simultaneous pressure flow and overtop flow through a bridge.

The solution domain, the 2D mesh and the geometry of the case is shown in Figure 12. Near the mid-section of the domain, the left opening is the bridge location, the right section is a weir, and the mid-section is a high wall. The bridge zone may experience pressurized flow and the zone has a longitudinal length of 3.0 m and lateral width of 5.4 m. The zone is defined by the upstream and downstream bridge nodestrings. The river bed elevation within the pressure zone is 0.0, while the ceiling elevation of the bridge is 1.0 m. The manning coefficient of the bridge ceiling is 0.04. For overflow simulation, the bridge deck top elevation is 3.0 m, the lateral span of the bridge is 5.5 m, and the top weir is assigned to be “PAVED”. The open channel itself has an average channel slope of 2.4% from upstream to downstream boundaries; the river bed Manning’s coefficient is 0.03; and the upstream discharge is 50 m³/s; and downstream water stage is 2.2 m.

The _SIF.dat file to run the model is listed in Figure 22 and sample results are shown in Figure 23.

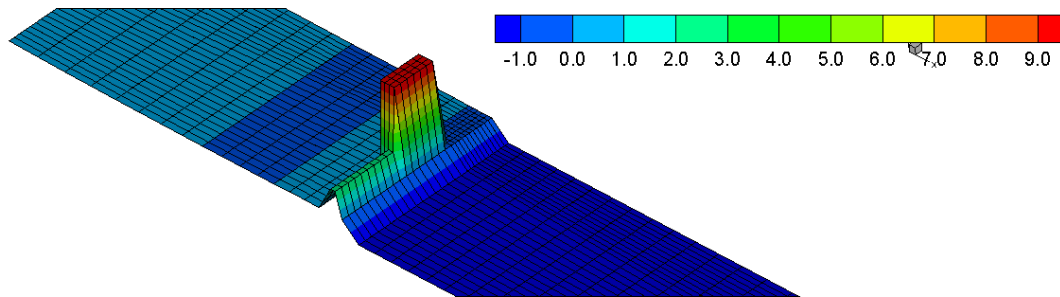


Figure 21. The geometry and bed elevation of the bridge and model domain


```

// Pressure Flow with Obstruction in the Press Zone
Pressure Flow Test #2: With Overtop Flow
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT + Monitor Point Coordinates: x1 y1 x2 y2 ...
1
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient Entry: a real value or a WD-n file name
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
50 SI
EXIT-H
2.2 SI
monitor
// Boundary Type (INLET-Q EXIT-H etc)
PRESS
PRESS
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream Pressure Flow Zones:
1
// Pressure Flow ZONE: Nodestring ID Pairs of each zone (Toe_ID Top_ID ...)
4 5
// Structure: Pressure Flow ZONE Parameters: ZC_UP ZC_DN MANNING UNIT
FLAT 1.0 1.0 0.04 SI
// Structure: Pressure Zone WEIR Parameters for Overtop Simulation: Ztop LW TYPE [Cw a b] (empty=NO_OverTop)
3.0 5.4 PAVED
// Number of In-Stream Flow Obstructions:
0
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used
// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1

```

Figure 22. The SIF file used to run the pressurized flow through a bridge with overtop

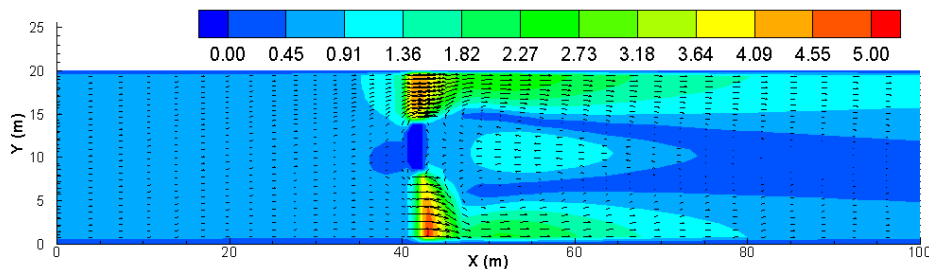


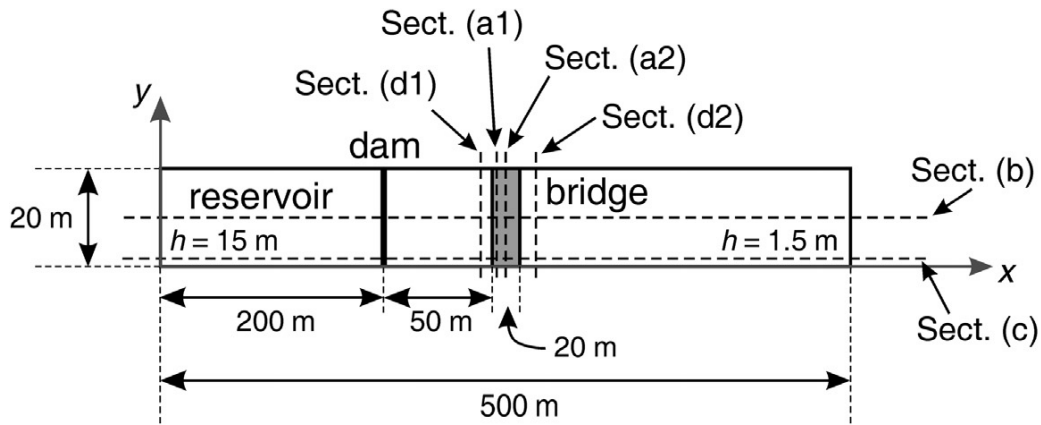
Figure 23. Simulated flow with velocity magnitude and vector with the pressure flow through a bridge with overtop

6.1.5. A Field-Scale Test Case with Arch Bridge

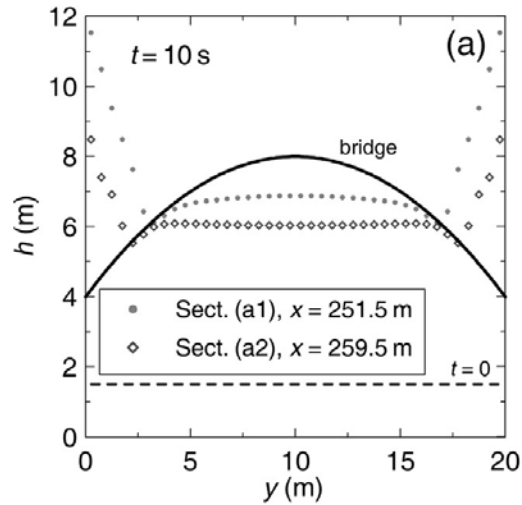
Herein, the transient process of a dam-break bore under an arch bridge is simulated in order to assess the pressure zone module for real case studies. The test was designed by Maranzoni et al. (2015). In practical applications, the presence of hydraulic structures such as bridges or culverts during a flood may strongly affect the near-field flow. The test case may test whether the module can handle unsteady process of transition from open channel flow to pressurized flow under only part of the bridge, as well as the impact of bridge to flood flows.

The geometry of the test case is shown in Figure 24. The model domain has a straight horizontal channel with length 500 m and lateral channel width 20 m. The first 200 m of the channel upstream is initially a reservoir with water depth of 15 m, while the remaining of the channel has a water depth of 1.5 m at time zero. A 20 m-wide arch bridge is located 50 m downstream of the dam face and perpendicular to the channel. The bridge ceiling has a parabolic profile described by the equation: $Z = -0.04y^2 + 0.8y + 4$ where Z is the ceiling elevation and y the transverse coordinate.

The water is initially still and a sudden failure of the dam will induce dam-break flow with the formation of a bore moving downstream towards the bridge.



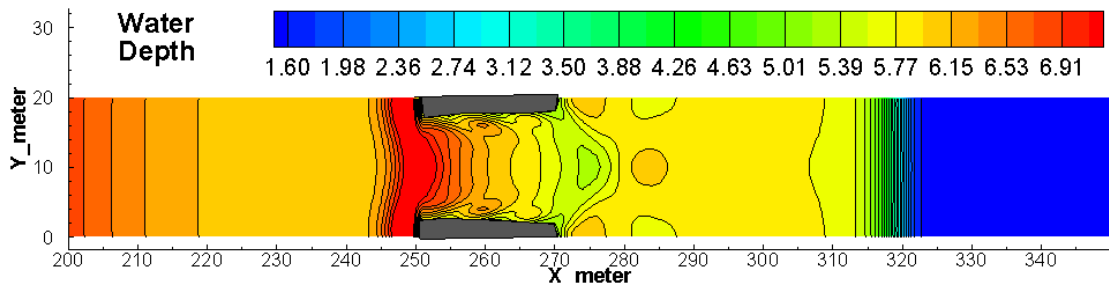
(a) Planar view of the model domain and bridge location



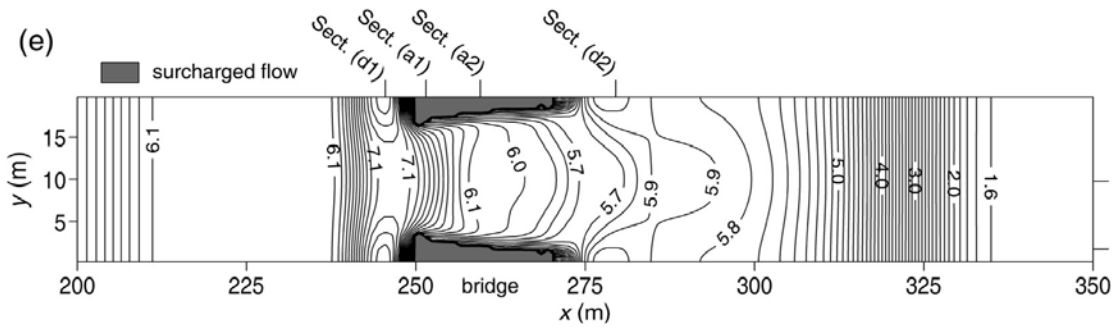
(b) Parabolic arch of the bridge ceiling

Figure 24. The model domain and bridge geometry (plots are from Maranzoni et al. 2015)

A 2D uniform mesh is generated with longitudinal mesh size of 1 m and lateral size of 0.5 m, leading to 20,000 mesh cells (500 by 40). The river bed Manning's roughness coefficient is 0.03, and no overtop flow is expected. Figure 25 shows the predicted water depth contours at time = 10s with both the SRH-2D pressurized flow module and the model developed by Maranzoni et al. (2015). At this time, the bore has already passed beyond the bridge and keeps propagating downward. The presence of the arch bridge gives origin to 2D near-field effects: a wake and a backwater effect develop behind and in front of the bridge, respectively. It is seen that the SRH-2D pressurized module predicted similar flow behaviors to the more sophisticated model of Maranzoni et al. (2015).



(a) SRH-2D results



(b) Results by Maranzoni et al. (2015)

Figure 25. Predicted water depth contours at time = 10 s when the dam-break bore is under the arch bridge (grey area are pressurized)

7.0 Internal Flow Obstructions

SRH-2D offers an option to add extra drag force (flow resistance) caused by in-stream or internal obstructions in streams such as piers, bridge decks, wood piles, etc. From hydraulic flow point of view, these internal obstructions slow down the flow within and near the obstruction locations. Additional drag forces should be added to the flow domain as these extra drag forces cannot be taken into consideration by the 2D depth-averaged approach explicitly. The intended applications include flows through bridge piers, wood piles, and other in-stream obstructions.

It is cautioned that the additional drags added to the flow are intended for the obstruction impacts on the flow farther away from the obstructions. Local flow velocity at and near the obstructions are not correct.

7.1.1. Theory

For each obstruction, users provide the location and spatial extent of the obstruction, along with the drag coefficient, and SRH-2D will then add the following total drag force to the affected zone:

$$F_d = \frac{1}{2} C_d \rho (U^2 + V^2) \beta A_p$$

where C_d = drag coefficient (dimensionless), A_p = wetted (i.e., below water) cross-sectional area of the obstruction projected normal to the direction of the approaching flow, ρ = water density, (U, V) = velocity components, and β is the porosity (percentage of projected area that is void; 0 porosity means a solid). For example, flow over an infinitely long circular cylinder (flow over a bridge pier is close to this scenario) has the drag coefficient shown in Figure 26. Approximately, users may use the values of various flow obstructions as in Table 5. However, these drag coefficients are developed for 3D flows. Therefore, in the 2D depth-averaged modeling such as SRH-2D, we recommend to adopt the drag coefficient as a calibration parameter. Also, porosity=1 may be used for all cases as it can be combined together with the drag coefficient in input.

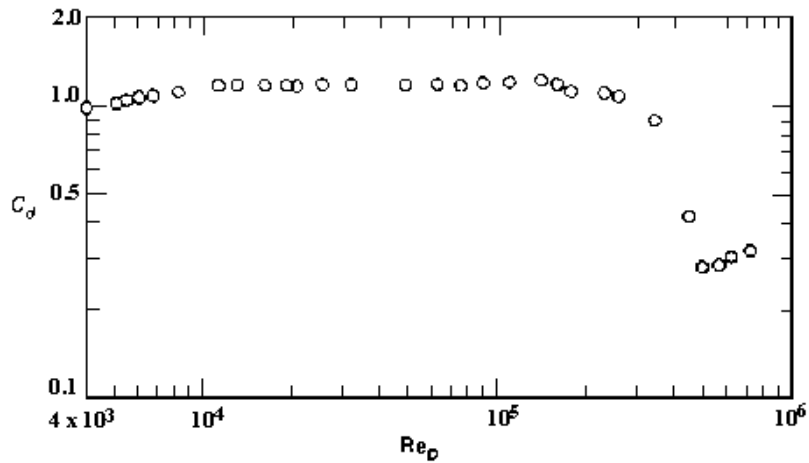


Figure 26. Drag coefficient for flow over a circular cylinder as a function of the Reynolds number

Table 5. Approximate drag coefficients for various flow obstructions

Type of Object	Drag Coefficient	Frontal Area - A
Long stream-lined body	0.1	
Modern Car like Toyota Prius	0.26	frontal area
Hollow semi-sphere facing stream	0.38	
Solid Hemisphere	0.42	$\pi / 4 d^2$
Sphere	0.5	
Cube	0.8	
Thin Disk	1.1	$\pi / 4 d^2$
Solid Hemisphere flow normal to flat side	1.17	$\pi / 4 d^2$
Squared flat plate at 90 deg	1.17	

Type of Object	Drag Coefficient	Frontal Area - A
Circular Cylinder	1.0 - 1.3	
Hollow semi-cylinder opposite stream	1.2	
Long flat plate at 90 deg	1.98	

Within SRH-2D, each obstruction is represented approximately by a hexagonal or a circular cylinder in geometry. The geometry of the obstructions themselves are not represented by the 2D mesh; instead, their geometry are used to determine which mesh cells are to have the additional drags due to the obstruction, as well as the amount of drag forces to be added. Users need to provide the following obstruction information: (a) the horizontal location and the extent of each obstruction (center coordinates and radius for a circular cylinder and horizontal coordinates of four vertices for a hexagon); (b) bottom elevation and vertical height of each obstruction. Each obstruction has a flat bottom and flat top. SRH-2D first finds all mesh cells who either contain an obstruction or are located within an obstruction; these cells are assigned extra flow resistance. The total resistance force due to the obstruction is distributed evenly among all impacted mesh cells by altering the bed shear stress as follows:

$$\tau_{total} = \tau_b + \frac{1}{2} C_d \rho (U^2 + V^2) \frac{\beta A_p}{\sum_{i=all_obstructin_cells} A_i}$$

In the above, A_i is the mesh cell area and the summation is over all cells containing the obstruction.

Note that if an obstruction extends from stream bed to above the water, it can be taken into account by simply using mesh cells to represent the obstruction and then either removing these cells to create a hole or setting the mesh cells as inactive (material ID equals zero within SMS). This approach requires no extra drag computation discussed herein.

There is a limitation that only up to two obstructions may be specified which overlay each other vertically. For example, beneath a bridge deck, piers may be added.

7.1.2. A Sample Case

A simple open channel flow with a flow obstruction is simulated. The solution domain, the 2D mesh generated, the bathymetry, and the location and size of the obstruction are shown in Figure 27. The channel has a slope of 0.49%, Manning's coefficient of 0.03, upstream discharge of $50 \text{ m}^3/\text{s}$ and downstream water stage of 5 m. The obstruction zone is a rectangle with its four corner coordinates given, the bottom elevation of the obstruction is 1.0 m and the height is 2.0 m, and the drag coefficient is 1.0. Simulated results are shown in Figure 28. The `_SIF.dat` file to run the model is listed in Figure 29.

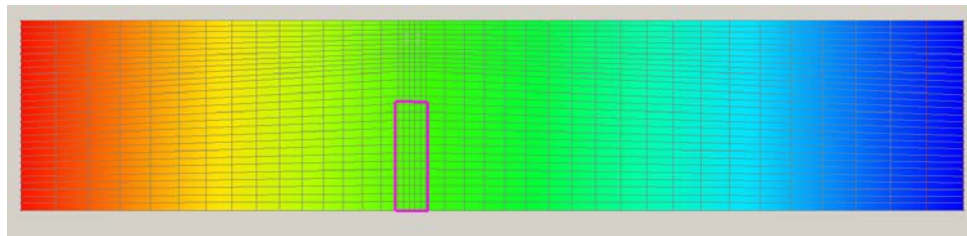


Figure 27. Solution domain, the mesh, bathymetry and the location and size of the obstruction

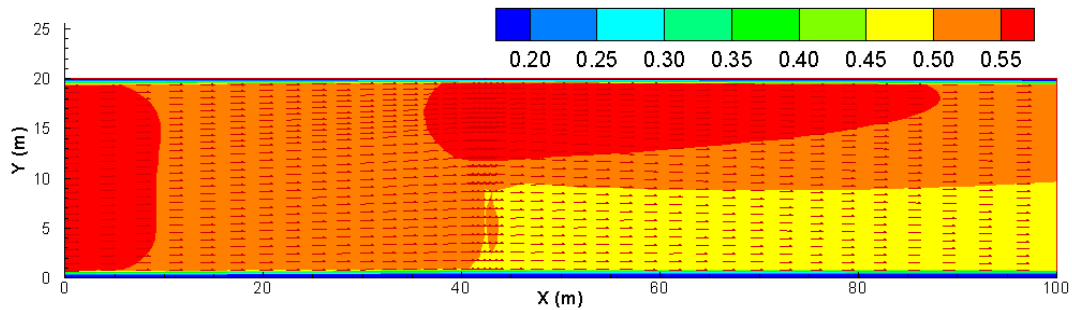


Figure 28. Simulated flow with velocity magnitude and vector with the flow obstruction case


```

// Simulation Description (not used by SRH-2D):
test
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT
1
// Monitor Point Coordinates: x1 y1 x2 y2 ...
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE)
PARA
// A_TURB for the PARA Model (0.05 to 1.0)
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM SMS
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
// Boundary Values (Q W QS TEM H_rough etc)
50 SI
// Boundary Type (INLET-Q EXIT-H etc)
EXIT-H
// Boundary Values (Q W QS TEM H_rough etc)
5.0 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of In-Stream Flow Obstructions:
1
// Obstruction parameters: HEXAGON X1 Y1 X2 Y2 X3 Y3 X4 Y4 Z_BOT HEIGHT UNIT CD Porosity
HEXAGON 39.56 11.5 39.56 0.0 43.52 0.0 43.52 11.5 1.0 2.0 SI 1.1 0.0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used

// Intermediate Result Output Control
-1

```

Figure 29. The SIF file used to run the obstruction case

8.0 Internal Flow Boundaries

SRH-2D offers an “internal boundary” option that allows water withdrawal or removal from or addition to the model domain. The internal boundary option is in addition to the external boundary along the model domain exterior. The internal boundary can be used alone for water withdrawal or water addition or it can be used in a pair to emulate the scenario that water is diverted from one arc and added back to the stream at a downstream arc. One application of the paired internal boundaries is a drainage detention pond, where flow is collected from one location and conveyed by a conduit for discharge at another location that can be a considerable distance downstream.

8.1.1. Theory and Usage

The location of an internal boundary is specified by a polyline along the 2D mesh lines consisting of a list of nodes. With SMS, it is represented by a “Nodestring” internal to the model domain; and with the Partial-Interface mode, each internal boundary is identified as “INTERNAL” (versus INLET-Q, EXIT-H, etc. for external boundaries). At each “internal” boundary, users have the following option to specify the flow rate (volume per unit time) through the boundary:

- The flow rate can be a constant or a time series if users know the flow rate;
- The rate is computed with a weir equation;
- The rate is linked to another “internal boundary” where the flow rate is known.

If the flow rate Q , constant or a time series, is provided, user specified flow rate is taken out of the boundary if Q is positive and added to the domain if Q is negative. If the bed upstream of the internal boundary is dry and Q is positive, no flow will be out of the domain as it is physically impossible. Users should be aware that unreasonable velocity solution may be obtained for the scenario that the water depth is small upstream of the internal boundary but Q is large. Such scenario may also lead to numerical instability. A sample case is discussed in 8.1.2 with this boundary condition.

The flow rate Q may be computed using a weir equation; i.e., Q (in m^3/s) is computed using the average water surface elevation (Z_U) along the internal boundary as follows:

$$Q = C_w \sqrt{g} L_w (Z_U - Z_B)^{3/2}$$

where Z_U = average water surface elevation along the internal boundary upstream cells; Z_B = average bed elevation along the boundary; C_w = dimensionless discharge coefficient; g = gravity acceleration; and L_w = length of the boundary. Users need to supply the parameters: C_w , Z_B and L_w . Within SRH-2D, Z_U is computed along the internal boundary upstream cells through averaging. Users have the option to use Z_U directly or the total head $[= Z_U + \frac{V_a^2}{2g}]$ with V_a the average total velocity along the boundary.

Flow rate Q into an internal boundary (say Q_2 at IB#2) may be linked to another internal boundary (say Q_1 at IB#1) where flow is taken out of the domain. Q_2 equals $-Q_1$ instantly if there is no time lag. Time lag may be used to account for long conveyance delays from IB#1 to IB#2. Users may use one of the two options in computing the time lag: a constant time lag or a conduit option. With the conduit option, the time lag is computed assuming full conduit flow using the user-giving conduit parameters. The conduit parameters include conduit length (L), radius (R), friction slope (S), and Manning's roughness coefficient (n). First, the conduit flow velocity is computed by:

$$V_{conduit} = \frac{R^{2/3} S^{1/2}}{n}. \text{ Then, the time lag is computed by } t_{lag} = \frac{L}{V_{Conduit}}.$$

There is a required that the internal boundary IB#1 should be specified first and IB#2 later in the list of nodestrings. The time lag computation assumes that the conduit is full.

8.1.2. Example of Coupled Internal Boundaries

A coupled internal boundary test case is presented to illustrate the use of the internal boundary module.

The sample case has a 2D domain, mesh and bed elevation contours shown in Figure 30. Two internal boundaries are specified. The upstream one is located on the left at $X=40$ m and width of 10.5 m; water is diverted from this boundary. The downstream boundary is located just 3 m downstream of the upstream boundary. The flow diversion rate of the upstream boundary is computed by the weir equation with the following parameters: discharge coefficient of 0.0448, bed sill elevation of $Z_B = 0.3$ m, and boundary lateral length of $L_w = 10.5$ m. The flow rate of the downstream boundary is linked to the upstream boundary with zero time lag.

The channel itself has a slope of about 0.5%, the Manning's coefficient of 0.03, upstream flow rate of $50 \text{ m}^3/\text{s}$, and downstream water elevation of 5 m.

The simulated flow velocity is shown in Figure 31 while the input SIF file is shown in Figure 32.

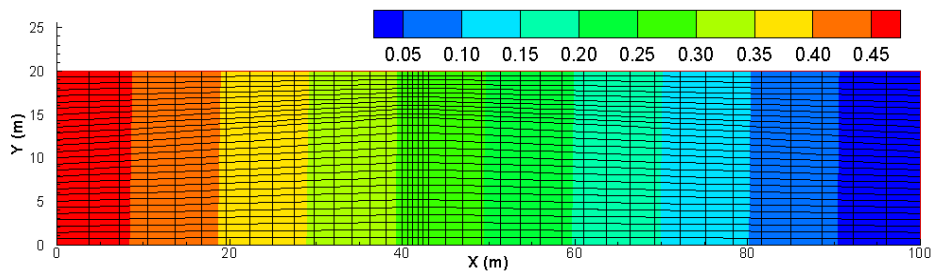


Figure 30. 2D domain, mesh and bed elevation of the internal boundary example

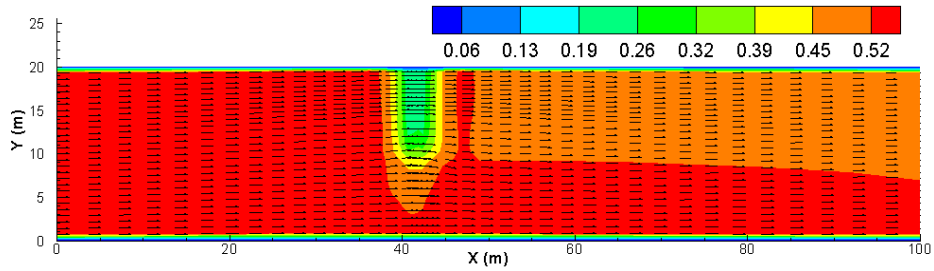


Figure 31. Predicted flow velocity for the internal boundary example

```
// Simulation Description (not used by SRH-2D):
Internal Boundary Case with weir eqn
// Module/Solver Selected (FLOW MORP MOB TEM TC)
FLOW
// Monitor-Point-Info: NPOINT; & Monitor Point Coordinates: x1 y1 x2 y2 ...
1
41.3 5.16
// Steady-or-Unsteady (STEADY/UNS)
STEADY
// Tstart Time_Step and Total_Simulation_Time: TSTART DT T_SIMU [FLAG]
0 1 10
// Turbulence-Model-Selection(PARA or KE) & A_TURB for the PARA Model (0.05 to 1.0)
PARA
0.7
// Initial Condition Method (DRY RST AUTO ZONAL)
DRY
// Mesh-Unit (FOOT METER INCH MM MILE KM GSCALE)
meter
// Mesh FILE_NAME and FORMAT(SMS...)
mesh1.2DM
// Manning Roughness Input Method(1=constant 2=material-type 3=(x y) distributed)
1
// Constant Manning Coefficient
0.03
// Any-Special-Treatments? (0 or empty = NO; 1=YES)

// Boundary Type (INLET-Q EXIT-H etc)
INLET-Q
50 SI
EXIT-H
5.0 SI
// Boundary Type (INLET-Q EXIT-H etc)
monitor
// Positive Q means water is taken out of the domain: WEIR CW ZB LW [HEAD]
// CW=0.0448 is to make Q=15cms
INTERNAL
WEIR 0.0448 0.3 10.5
// qout.dat SI
INTERNAL
LINK_TO PREVIOUS 0.0
// Wall-Roughness-Height-Specification (empty-line=DONE)

// Number of Obstructions
0
// Results-Output-Format-and-Unit(SRHC/TEC/SRHN/XMDF;SI/EN)
TEC SI
// Headers of Output Variables specified by the User: EMPTY line means default is used

// Intermediate Result Output Control: INTERVAL(hour) OR List of T1 T2 ... EMPTY means the end
-1
```

Figure 32. The SIF file used to run the internal boundary case

9.0 References

- FHWA, 2002, User's Manual for FESWMS Flo2DH, Publication No. FHWA-RD-03-053, Federal Highway Administration, U.S. Department of Transportation.
- Greimann, B.P., Lai, Y.G., and Huang, J. 2008. Two-Dimensional Total Sediment Load Model Equations. *J. Hydraul. Eng., ASCE*, 134(8): 1142-1146.
- Lai, Y.G., Weber, L.J., and Patel, V.C. (2003). Non-hydrostatic three-dimensional method for hydraulic flow simulation - Part I: formulation and verification. *J. Hydraul. Eng., ASCE*, 129(3), 196-205.
- Lai, Y.G. (2008). SRH-2D Theory and User's Manual version 2.0, Technical Service Center, Bureau of Reclamation, Denver, CO.
- Lai, Y.G. (2010). Two-Dimensional Depth-Averaged Flow Modeling with an Unstructured Hybrid Mesh. *J. Hydraul. Eng., ASCE*, 136(1): 12-23.
- Lai, Y.G., and Greimann, B.P. (2010). Predicting contraction scour with a two-dimensional depth-averaged model. *J. Hydraulic Research, IAHR*, 48(3): 383-387.
- Lai, Y.G., Greimann, B.P., and Wu, K. (2011). Soft bedrock erosion modeling with a two-dimensional depth-averaged model," *J. Hydraul. Eng., ASCE*, 137(8): 804-814.
- Maranzoni, A., Dazzi, S., Aureli, F, and Mignosa, P. (2015). Extension and application of the Preissmann slot model to 2D transient mixed flows. *Advances in Water Resources* 82 (2015) 70-82.
- Novak, P., Moffat, A. I. B., Nalluri, C., and Narayanan, R. (1996). *Hydraulic structures*, (2nd ed.), E & FN Spon, London, United Kingdom.