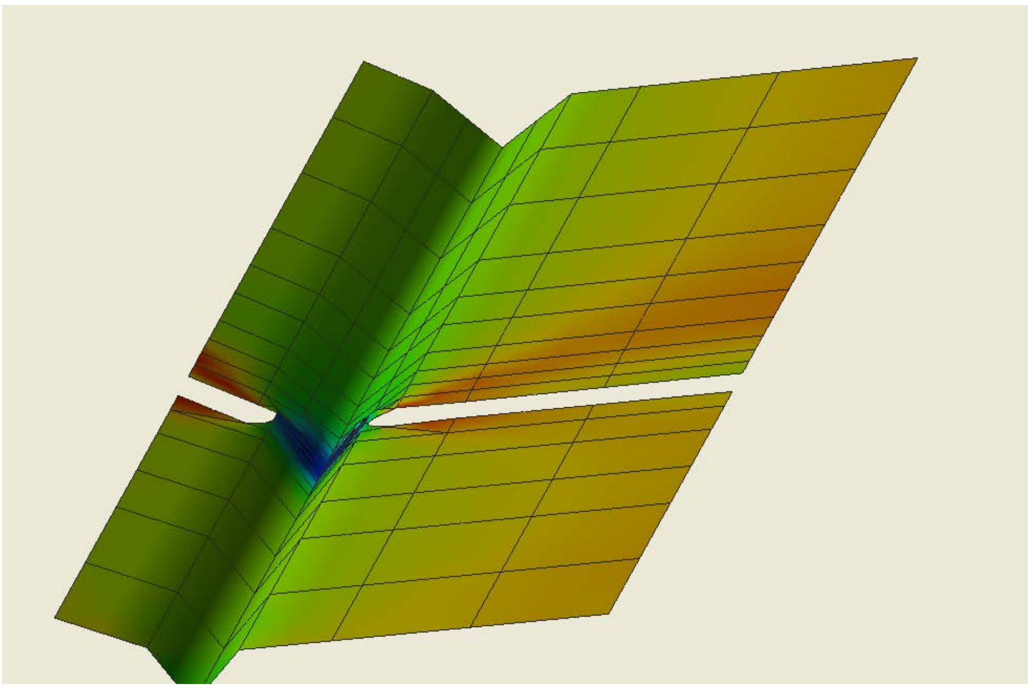
SMS

v. 11.2

SMS 11.2 Tutorial

FESWMS Analysis with Weirs



Objectives

This lesson will teach you how to prepare an *FST2DH* simulation which includes the use of weirs.

Prerequisites

- Overview Tutorial
- FESWMS Tutorial

Requirements

- FESWMS
- Fst2dh
- Mesh Module

Time

- 30-60 minutes

AQUAVEO™

1 Introduction

You will be using the file *suecreek.sms* as shown in Figure 1. This mesh has been created and renumbered. To open the mesh data:

1. Select *File* | *Open.*
2. Highlight the file *suecreek.sms* in the Data Files Folder for this tutorial and click the *Open* button. If you still have geometry open from a previous tutorial, you will be asked if you want to delete existing data. If this happens, click the *Yes* button.

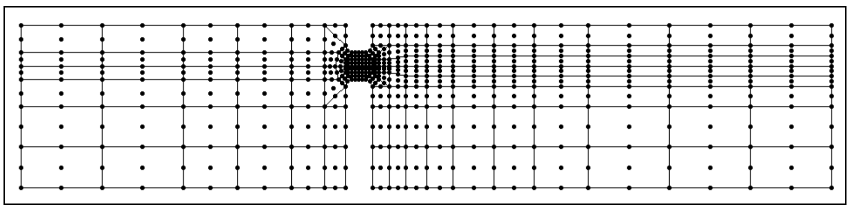


Figure 1. The suecreek.sms geometry

2 Defining Material Properties

Each element of the mesh is assigned a material ID. The material ID tells *FST2DH* which material properties should be assigned to the element. We want to view the materials used in our mesh, let’s turn them on:

1. Go to *Display* | *Display Options*.
2. Make sure that 2D Mesh is selected on the left and the 2D Mesh tab is active.
3. Turn off nodes and elements and turn on Materials.
4. Click OK to return to the main SMS screen.

There are four different material types in this mesh, but the material properties have not been defined. When *SMS* opens a mesh with undefined materials, the materials are assigned default properties. See the *FESWMS* documentation for a definition of individual material parameters. To change the material values:

1. Select *FESWMS* | *Material Properties*. A graphical image at the top of the *Roughness Parameters* tab of the *FESWMS Material Properties* dialog shows the Manning’s n value as a function of water depth.
2. Highlight *material_01*, and enter the following values:

- On the *Roughness Parameters* tab enter 0.035 for both *n1* and *n2*
- On the *Turbulence Parameters* tab enter 20.0 for *Vo* and 0.6 for *Cu1*

3. Select *material_02* and enter the same values that were entered for *material_01*.
4. For *material_03* and *material_04*, enter the same values as above, except that *n1* and *n2* are both 0.055.
5. Click the *OK* button to accept these changes and close the *FESWMS Material Properties* dialog.

You have just assigned values for the four materials in this mesh. Notice that there are only two distinct material regions because materials 1 and 2 have the same values, as do materials 3 and 4. Now that we are finished with the materials, let’s turn off their display:

1. Go to *Display | Display Options.*
2. Make sure that 2D Mesh is selected on the left and the 2D Mesh tab is active.
3. Turn off Materials and turn on elements.
4. Click OK to return to the main SMS screen.

2.1 Assigning Boundary Conditions

Boundary conditions such as flow and head define how water enters and leaves the finite element network. Without proper boundary conditions, instability of the model and inaccuracy of the solution will result.
A steady state model such as this can only have constant boundary conditions. The flow and head boundary conditions will be defined at nodestrings on opposite sides of the model as shown in Figure 2. To create the two boundary nodestrings:

1. Choose the *Create Nodestrings* tool from the *Toolbox.*
2. Click on the lower node of the left boundary.
3. Hold the *SHIFT* key and double-click on the upper node of the left boundary. This creates the nodestring all the way across the left boundary of the mesh. If you did not hold the *SHIFT* key, it would not be a valid boundary nodestring because it would not include all nodes across the boundary.
4. Repeat this procedure to create a nodestring across the right boundary.

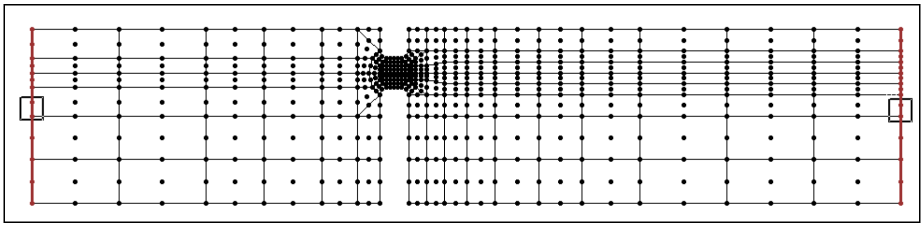


Figure 2.   Location of the boundary condition nodestrings.

Boundary conditions can now be assigned to the nodestrings. To assign the flow to the left boundary:

1. Choose the *Select Nodestrings* tool from the *Toolbox*. An icon appears at the center of each nodestring, as shown in Figure 2.
2. Select the nodestring on the left boundary by clicking inside its icon.
3. Select *FESWMS | Assign BC*.
4. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Flow* option and assign a value of 9000 (cfs). Be sure that the *Normal* option is selected.
5. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a flow nodestring and its color changes. An arrow appears at the center of the nodestring to indicate the flow direction and the flow value is shown next to the arrow (see Figure 3).

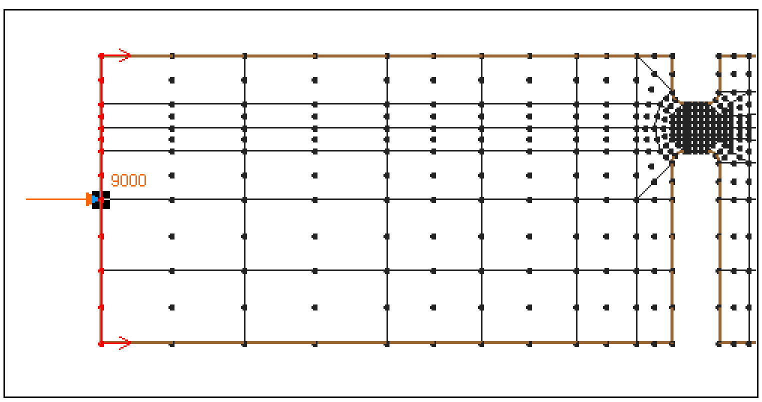


Figure 3   The inflow boundary condition.

To assign the head to the right boundary:

1. Select the right nodestring.
2. Select *FESWMS | Assign BC*.

3. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Water surface elevation* option and assign a value of 812.9 (feet). Be sure that the *Essential* option is selected.
4. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a head nodestring and its color changes. A head symbol appears at the center of the nodestring and the head value is shown next to the symbol (see Figure 4).

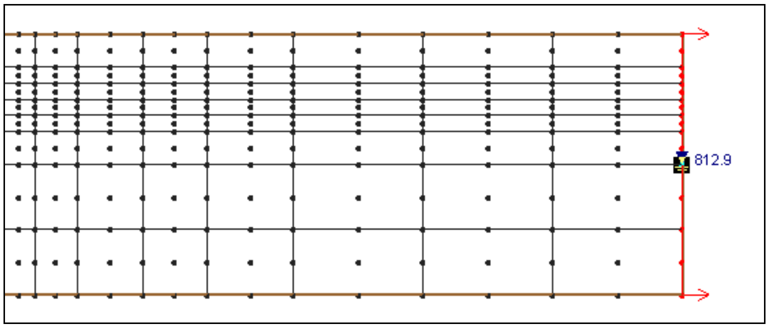


Figure 4 The outflow boundary condition.

3 Creating Weirs

With *FESWMS*, flow control structures such as weirs, piers, culverts, and drop inlets are easily added to the mesh. Weirs, culverts, and drop inlets are created between pairs of nodes. Wide structures can be created between strings of node pairs. For this model, a weir will be defined along seven node pairs across the abutment at the bottom middle of the mesh, as shown in Figure 5.

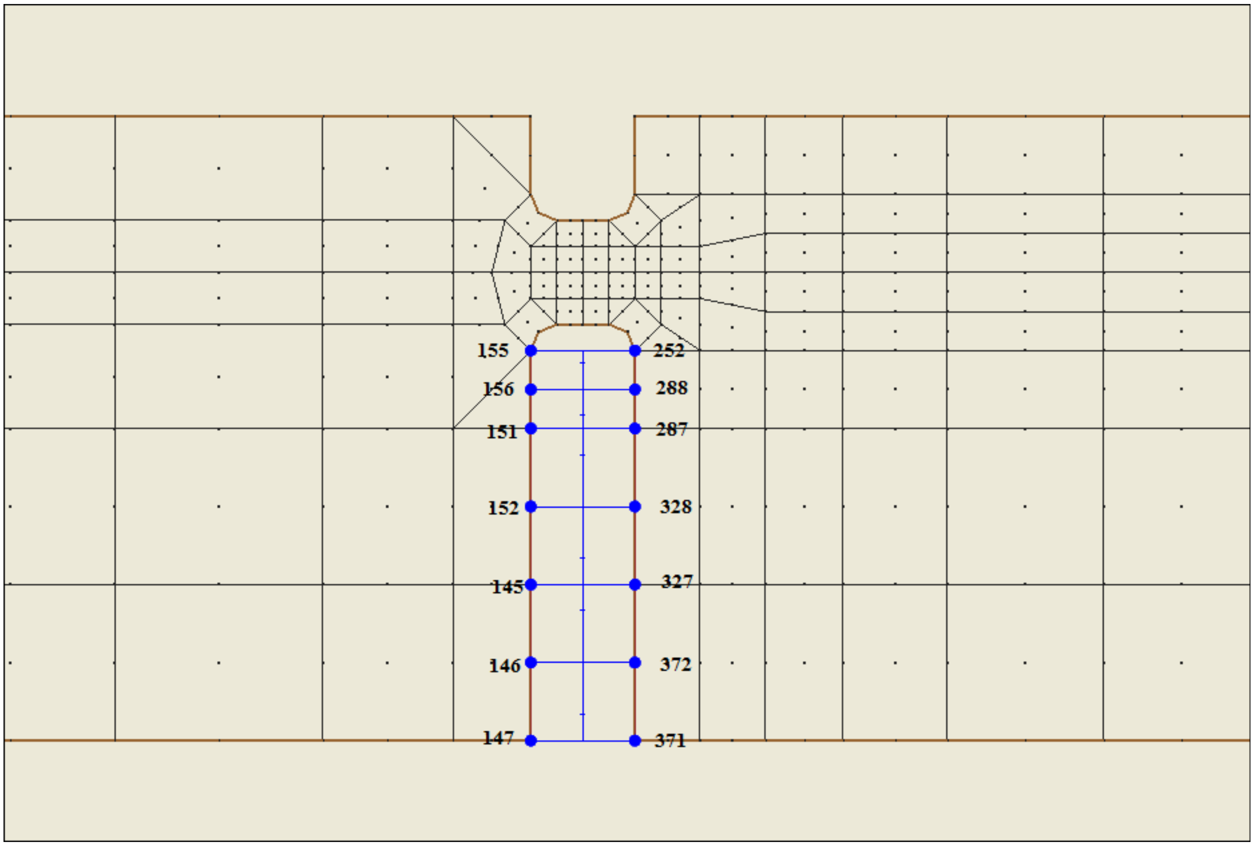


Figure 5   Area where weirs will be added.

This image highlights the nodes across which weir segments will be created. To see these nodes more clearly:

1. *Zoom* in to the area shown in Figure 5.
2. Open the *Display Options* dialog and turn on the *Node Numbers* option in the *2D Mesh* tab then click *OK*.

The seven node pairs for the weir are: 155<->252, 156<->288, 151<->287, 152<->328, 145<->327, 146<->372, and 147<->371. A set of weir segments can be created across pairs of adjacent nodes by using nodestrings. To create the weir segments:

1. Choose the *Create Nodestrings* tool from the *Toolbox*.
2. Hold SHIFT and create one nodestring from node 155 to 147 and a second from node 252 to 371.

3. Choose the *Select Nodestrings* tool and select both nodestrings by selecting one, and holding down shift while selecting the other.
4. Select *FESWMS* | *Weir*. Make sure the *Upstream* nodestring is that across nodes 155 to 147 (click the *Switch* button if not).
5. Enter the following values:
   - Set the *Weir type* to “Paved roadway”.
   - 825 for the *Zc* – Crest Elevation.
6. Click the *OK* button to close the dialog.

A set of five consecutive weir segments spanning the two bottom elements has just been defined. Together, these segments define a 300-foot-long broad crested weir with a crest elevation of 825 feet, and discharge coefficient of 0.544. SMS displays tick marks to show the break between weir segments. There is a specific formula for determining the length of each pair of nodes. Each midside node has 2/3 of the element width in its crest length while each corner node has 1/6 of each element width that is involved in the weir. See the *SMS Help* for more information on weirs and other flow control structures. After creating the weir, reset the display to the way it was before starting the weir creation, as shown in Figure 6. To do this:

1. Turn off the display of node numbers using the *Display Options* dialog.
2. Click the *Frame* macro in the *Toolbox* to frame the image.

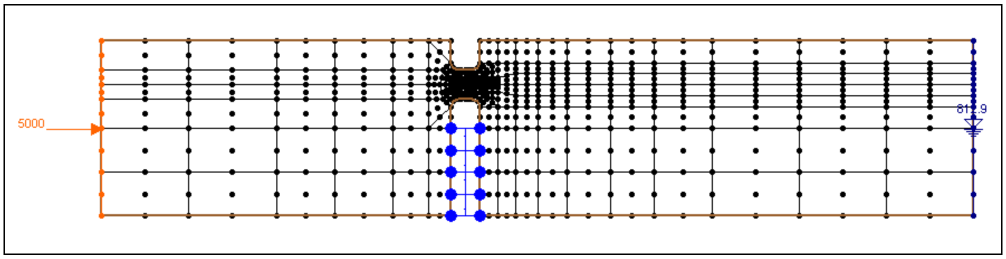


Figure 6. The bridge geometry with weirs

4 Saving the Data

For the most part everything was setup for us. However, we need to modify the FESMWS model parameters:

1. Select *FESWMS* | *Model Control.*
2. Click the *Parameters* tab and turn on *Element drying / wetting*.

3. Click the *OK* button to close the dialog.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *File* | *Save As*.
2. Make sure the *Save as type* is *Project Files* and enter the name *suecreek2.*
3. Click the *Save* button.

5 Using FST2DH

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FST2DH*. It uses either a previous solution or a default value as initial conditions to compute the solution. The default conditions correspond to still water at the *Water surface elevation* specified in the *Parameters* section of the *Model Control*. In this case we will use the default condition. To run *FST2DH*:

1. Select *FESWMS* | *Run FST2DH.*
2. The model wrapper will display the progress of the iterations. After it finishes, click the *Exit* button.

The solution is stored in a file called *suecreek2.flo*. This file contains the velocity and water surface elevation for each node in the mesh. It is automatically read into *SMS* upon clicking the *Exit* button so long as the *Load solution* option is checked. (The solution is also provided in the Data Files Folder.)

6 Editing Weir Data

In the solution just computed, there is no flow over the weir. The water surface at the weir is much lower than the weir crest elevation so no overtopping occurred. This was purposely done to add model stability. With an initial solution, the weir's crest elevation can be lowered to allow overtopping. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox* and select the two weir nodestrings.
2. Choose *FESWMS* | *Weir*.
3. Change the *Crest Elevation* to *812.5 feet* and click the *OK* button.

6.1 Using the Hotstart file

SMS needs to tell *FST2DH* to use the previous solution file as an input *hotstart*, or *initial condition*, file. To do this:

1. Select *FESWMS | Model Control*.
2. In the *FESWMS Model Control* dialog, turn on the *INI file* option.
3. Click the *File Browser* button to the right of this option. Find and open the file *suecreek2.flo* that was created when *FST2DH* ran. If you were not able to run the model, you can use the solution file that is provided in the *output* directory.
4. Click the *OK* button in the *FESWMS Model Control* dialog.

6.2 Computing a New Solution File Using a Hotstart File

To run the new simulation:

1. Select *File | Save As*, make sure the "Save as type:" is *Project Files (*.sms)* and save the simulation as *suecreek3.sms*.
3. Select *FESWMS | Run FST2DH*.

After *FST2DH* finishes, the solution file should automatically be opened into *SMS* for post processing (as long as the *Load solution* option is on). See the visualization tutorial for post processing operations.

7 Checking Flow over Weirs

When fst2dh runs, it saves information for each weir segment. To view this:

1. Select *File | View Data File* and open the file *suecreek3.prt*.
2. Choose to open the file using *Notepad*.
4. From within notepad, select *Edit | Find* and search for the text "WEIR REPORT".
3. Hold the F3 key to keep finding weir reports until a message appears that the text cannot be found. This will be the last weir report in the file.

The final weir report should look like the following:

```
                             *** SUMMARY WEIR REPORT ***
================================================================================
      Weir -------- Node 1 -------- -------- Node 2 -------- ------ Flow -------
        id   Node  WS elev   Energy   Node  WS elev   Energy  Flow rate Submerge
              no.     (ft)     (ft)    no.     (ft)     (ft) (ft^3/sec)   factor
```

weir-1-5	147	813.966	813.984	371	813.083	813.106	136.980	1.000
weir-1-4	146	813.974	813.989	372	813.074	813.095	552.475	1.000
weir-1-3	145	813.983	813.994	327	813.066	813.083	278.505	1.000
weir-1-2	152	813.985	813.995	328	813.053	813.068	558.662	1.000
weir-1-1	151	813.988	814.003	287	813.040	813.048	140.087	1.000

The second-to-last column shows the flowrate over each weir segment. To see which segment corresponds with which pair of nodes, turn on node numbers inside SMS.