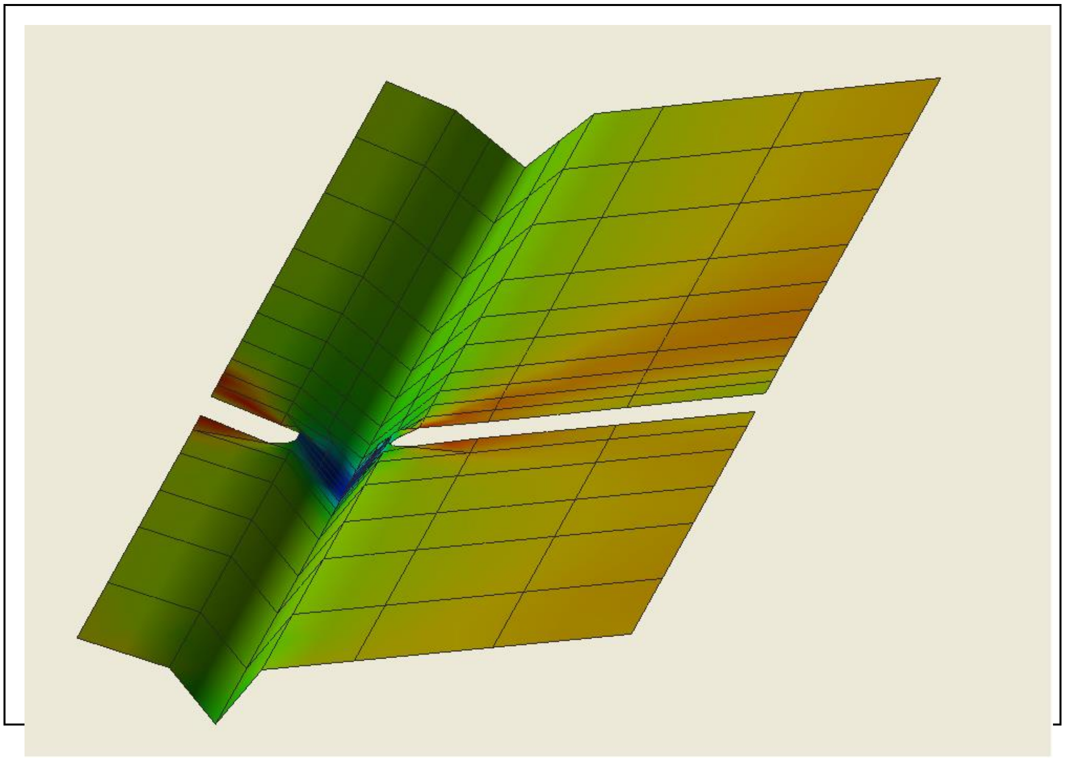
SMS

v. 12.1

*SMS 12.1 Tutorial*

# ***FESWMS Analysis with Weirs***



## Objectives

This lesson will teaches how to prepare an FESWMS simulation, including the use of weirs.

## Prerequisites

- Overview Tutorial
- FESWMS Tutorial

## Requirements

- FESWMS
- FST2DH
- Mesh Module

## Time

- 15-30 minutes

AQUAVEO™

# 1 Introduction

This tutorial uses the project file "suecreek.sms". It contains a mesh created and renumbered specifically for this tutorial.

To open the mesh data:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Highlight the file "suecreek.sms" in the *data files* folder for this tutorial and click the **Open** button.
3. If geometry is still open from a previous tutorial, a dialog will ask if existing data should be deleted. If this happens, click the **Yes** button.

The project file will open and should appear similar to Figure 1.

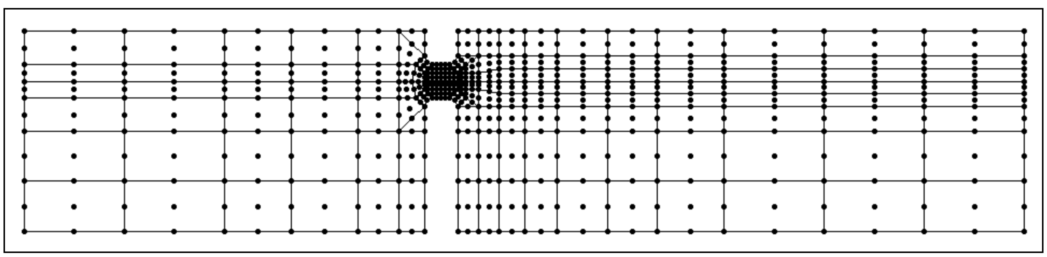


*Figure 1 The "suecreek.sms" geometry*

# 2 Defining Material Properties

Each element of the mesh is assigned a material ID. The material ID tells FESWMS which material properties should be assigned to the element.

To view the materials used in the mesh:

1. Go to *Display* | **Display Options** to open the *Display Options* dialog.
2. Select "2D Mesh" from the list on the left.
3. Turn off *Nodes* and *Elements* then turn on *Materials*.
4. Click **OK** to close the *Display Options* dialog.

The mesh should now appear as in Figure 2.

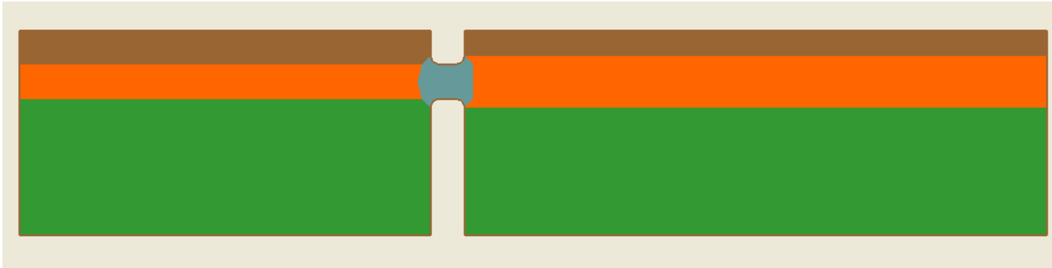


*Figure 2 The mesh with only Materials being displayed*

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** to bring up the *FESWMS Material Properties* dialog.

A graphical image at the top of the *Roughness Parameters* tab shows the Manning's *n* value as a function of water depth.

2. Select "material_01" from the list on the left.
3. Enter values shown in the table below for *n1* and *n2* on the *Roughness Parameters* tab, and for *Vo* and *Cu1* on the *Turbulence Parameters* tab.
4. Repeat steps 2-3 for "material_02", "material_03", and "material_04".
5. Click the **OK** to close the *FESWMS Material Properties* dialog.

<table>
<tr><th rowspan="2">Material</th><th colspan="2">Roughness Parameters</th><th colspan="2">Turbulence Parameters</th></tr>
<tr><th>n1</th><th>n2</th><th>Vo</th><th>Cu1</th></tr>
<tr><td>material_01</td><td colspan="2" rowspan="2">0.035</td><td rowspan="4">20.0</td><td rowspan="4">0.6</td></tr>
<tr><td>material_02</td></tr>
<tr><td>material_03</td><td colspan="2" rowspan="2">0.055</td></tr>
<tr><td>material_04</td></tr>
</table>

Values have now been assigned 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 the material values have been assigned, turn off their display:

1. Go to *Display* | **Display Options** to open the *Display Options* dialog again.
2. Select "2D Mesh" from the list on the left.
3. Turn off *Materials* and turn on *Elements*.
4. Click **OK** to close the *Display Options* dialog.

## 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 3.

To create the two boundary nodestrings:

1. Select the **Create Nodestrings** tool.
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 the *Shift* key is not held down when this is done, a valid boundary nodestring is not created because it does not include all nodes across the boundary.

4. Repeat this procedure to create a nodestring across the right boundary.

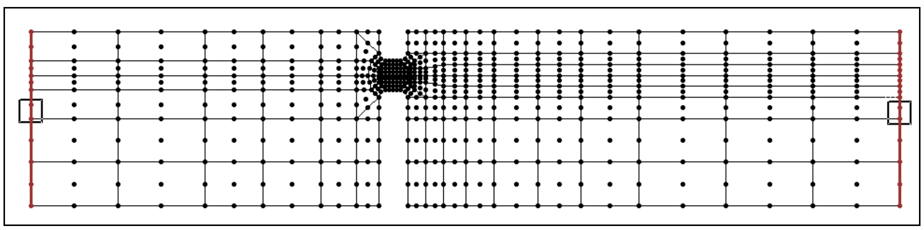


*Figure 3 Location of the boundary condition nodestrings*

Boundary conditions can now be assigned to the left nodestring by doing the following:

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

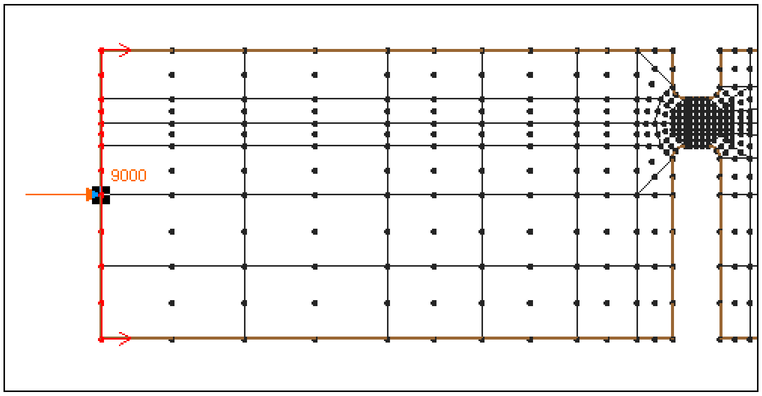



*Figure 4 The inflow boundary condition*

To assign the head to the right boundary:

1. Select the right nodestring.
2. Right-click and select **Assign BC…** to bring up the *FESWMS Nodestring Boundary Conditions* dialog.
3. Turn on *Water surface elevation* and enter "812.9" in the *WSE* field. Be sure that the "Essential" option is selected.
4. Click **OK** to close the *FESWMS Nodestring Boundary Conditions* 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 5).

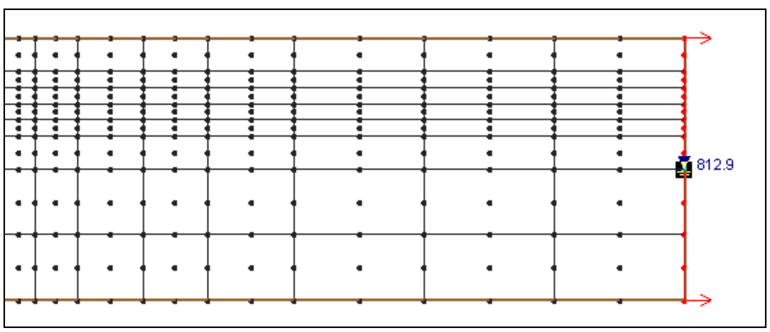



*Figure 5 The outflow boundary condition*

# 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 6.

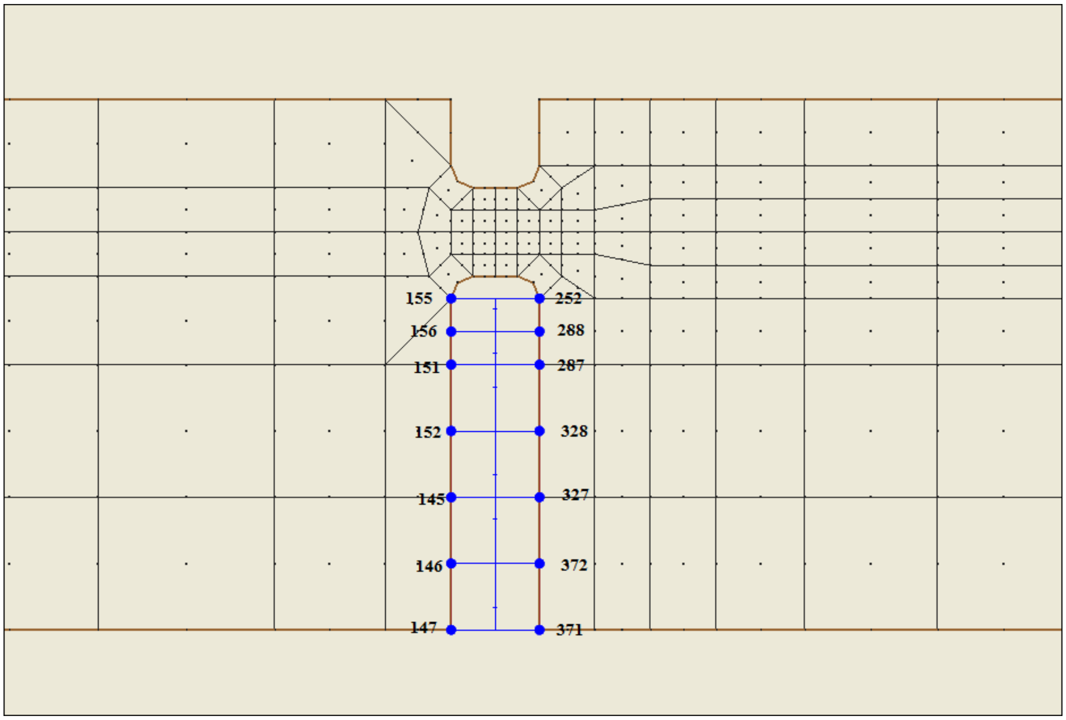



*Figure 6 Area where weirs will be added*

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 6.
2. Click the **Display Options** macro to open the *Display Options* dialog.
3. Select "2D Mesh" from the list on the left.
4. Turn on *Node Numbers*.
5. Click **OK** to close the *Display Options* dialog.

The seven node pairs for the weir are:

- 155↔252.
- 156↔288.
- 151↔287.
- 152↔328.
- 145↔327.
- 146↔372.
- 147↔371.

A set of weir segments can be created across pairs of adjacent nodes by using nodestrings by doing the following:

1. Select the **Create Nodestrings** tool.

2. Hold the *Shift* key and create a nodestring from node 147 to 155 (right to left, as if facing downstream).
3. Hold the *Shift* key and create a second from node 371 to 252.
4. Using the **Select Nodestrings** tool, select both nodestrings by selecting one, and holding down the *Shift* key while selecting the other.
5. Select *FESWMS* | **Weir** to bring up the *FESWMS Weir* dialog.
6. Note that each nodestring is labeled: one is labeled "B", and the other is labeled "A". In the dialog, make sure the value to the right of *Upstream* matches the letter corresponding to the nodestring going from node 147 to 155. If it does not, click on the **Switch** button (see Figure 7, below).
7. In the *Weir Coefficients* section, set *Weir type* to "Paved roadway".
8. In the *Weir Geometry* section, set *Zc – Crest Elevation* to "825".
9. Click **OK** to close the *FESWMS Weir* 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 along the centerline of the weir to show the breaks between weir segments.

There is a specific formula for determining the length of each pair of nodes. Each midside node has two-thirds of the element width in its crest length while each corner node has one-sixth of each element width that is involved in the weir. See *SMS Help* for more information on weirs and other flow control structures.

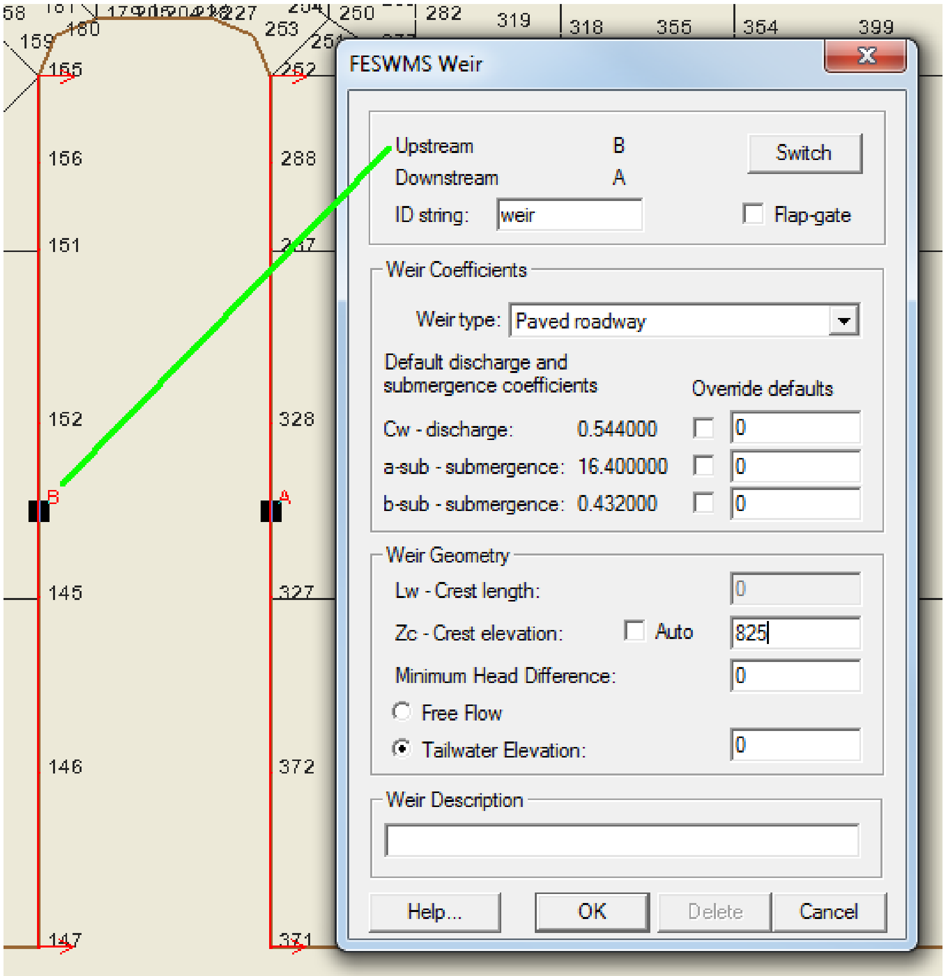



*Figure 7 FESWMS Weir dialog*

After creating the weir, reset the display to the way it was before starting the weir creation, as shown in Figure 8.

To do this:

1. Click the **Display Options** macro to open the *Display Options* dialog.
2. Select "2D Mesh" from the list on the left then turn off *Node Numbers*.
3. Click **OK** to close the *Display Options* dialog..
4. Click the **Frame** macro to frame the image.

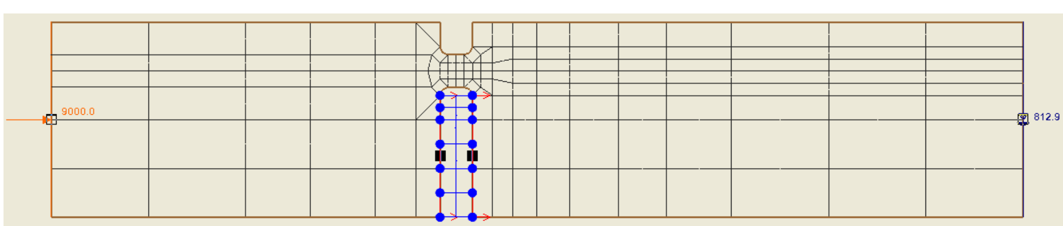


*Figure 8 The bridge geometry with weirs*

# 4 Assigning Model Parameters

For the most part everything was setup for the tutorial. However, the FESMWS model parameters need to be modified:

1. Select *FESWMS* | **Model Control** to bring up the *FESWMS Model Control* dialog.
2. On the *Parameters* tab in the *General Parameters* section, turn on *Element drying / wetting*.
3. Click **OK** to close the *FESWMS Model Control* dialog.
4. Right-click on the “suecreek” mesh and select **Rename**.
5. Enter the name as “suecreek2”.

# 5 Saving the Data

The data is now ready to be saved by doing the following:

1. Select *File* | **Save As...** to bring the *Save As* dialog.
2. Set *Save as type* to “Project Files (*.sms)”
3. Enter “suecreek2.sms” in the *File name* field.
4. Click the **Save** button to save the project.

# 6 Using FST2DH

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 tutorial, the default conditions are used.

To run FST2DH:

1. Select *FESWMS* | **Run FST2DH** to launch the *FESWMS* model wrapper. The model wrapper will display the progress of the iterations.
2. When the model run finishes, turn on *Load solution* and click **Exit** to close the *FESWMS* model wrapper.

The solution is stored in a file named “suecreek2.flo”. This file contains the velocity and water surface elevation for each node in the mesh. SMS automatically loads the solution by following the directions above. The solution file is also provided in the *data files\output* folder.

# 7 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 by doing the following:

1. Using the **Select Nodestrings** tool, select the two weir nodestrings by using the *Shift* key.
2. Select *FESWMS* | **Weir** to open the *FESWMS Weir* dialog.
3. In the *Weir Geometry* section, set *Zc – Crest Elevation* to "812.5".
4. Click **OK** to close the *FESWMS Weir* dialog.

## 7.1 Using the Hot Start File

SMS needs to tell FST2DH to use the previous solution file as an input "hot start", or "initial condition", file.

To do this:

1. Select *FESWMS* | **Model Control** to bring up the *FESWMS Model Control* dialog.
2. In the *FST2DH Input* section, turn on the *INI file* option.
3. Click the **File Browser** button to the right of this option to bring up an *Open* dialog.
4. Browse to the *data files\suecreek2\FESWMS\suecreek2* folder and select "suecreek2.flo" (created when FST2DH ran). If unable to run the model, use the "suecreek2.flo" solution file provided in the *data files\output* directory.
5. Click on **Open** to close the *Open* dialog.
6. Click **OK** to close the *FESWMS Model Control* dialog.

## 7.2 Computing a New Solution File Using a Hot Start File

To run the new simulation:

1. Right-click on the "suecreek2" mesh and select **Rename**.
2. Enter the name as "suecreek3".
3. Select *File* | **Save As...** to bring up the *Save As* dialog.
4. Set *Save as type* to "Project Files (*.sms)"
5. Enter "suecreek3.sms" in the *File name* field.
6. Click the **Save** button.
7. Select *FESWMS* | **Run FST2DH** to start the *FESWMS* model wrapper.
8. When the model run finishes, turn on *Load solution* and click **Exit** to close the *FESWMS* model wrapper.

The solution is stored in a file named "suecreek3.flo". This file contains the velocity and water surface elevation for each node in the mesh. SMS automatically loads the solution by following the directions above. The solution file is also provided in the *data files\output* folder. See the *Data Visualization* tutorial for post-processing operations.

# 8 Checking Flow over Weirs

When FST2DH runs, it saves information for each weir segment. To view this:

1. Select *File* | **View Data File** to bring up the *Open* dialog.
2. Browse to the *data files\suecreek2\FESWMS\suecreek3* folder.
3. Double-click on "suecreek3.prt" to open it and bring up the *View Data File* dialog.
4. Choose to open the file using "Notepad" (or any other text editor) and click OK to close the *View Data File* dialog.
5. From within Notepad, select *Edit* | **Find**.
6. Search for the text "WEIR REPORT".
7. Tap the **Find Next** button repeatedly until a message appears stating 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 id | Node 1 Node no. | Node 1 WS elev (ft) | Node 1 Energy (ft) | Node 2 Node no. | Node 2 WS elev (ft) | Node 2 Energy (ft) | Flow rate (ft^3/sec) | Flow Submerge factor |
|---|---|---|---|---|---|---|---|---|
| weir-7 | 155 | 813.914 | 813.998 | 252 | 813.089 | 813.092 | 64.875 | 1.000 |
| weir-6 | 156 | 813.949 | 813.990 | 288 | 813.116 | 813.132 | 269.133 | 1.000 |
| weir-5 | 151 | 813.984 | 814.005 | 287 | 813.143 | 813.156 | 209.162 | 1.000 |
| weir-4 | 152 | 813.990 | 814.002 | 328 | 813.148 | 813.161 | 561.004 | 1.000 |
| weir-3 | 145 | 813.995 | 814.008 | 327 | 813.153 | 813.169 | 282.142 | 1.000 |
| weir-2 | 146 | 813.986 | 814.001 | 372 | 813.162 | 813.182 | 558.937 | 1.000 |
| weir-1 | 147 | 813.977 | 813.995 | 371 | 813.171 | 813.194 | 138.410 | 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.

# 9 Conclusion

This concludes the *FESWMS Analysis with Weirs* tutorial. Continue experimenting with FESWMS weirs or exit the program.