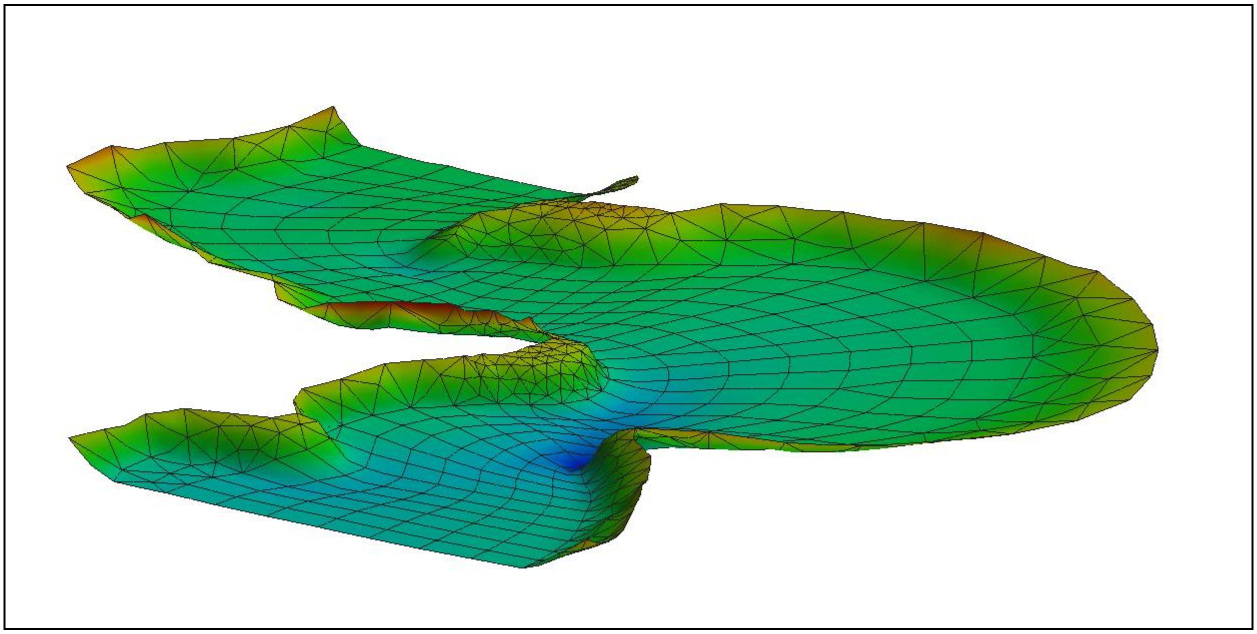
v. 12.1

SMS 12.1 Tutorial

Basic FESWMS Analysis



Objectives

This tutorial instructs on how to prepare a mesh for a FESWMS simulation.

Prerequisites

- Overview Tutorial

Requirements

- FESWMS
- Fst2dh
- Mesh Module

Time

- 15-30 minutes

1 Introduction

The project file "stmary.sms", included in the *data files* folder, is used for this tutorial. This project file includes a FESWMS project (*.fpr) file containing a list of filenames used by FESWMS. The actual input data is stored in the files named in the project file.

To open the SMS project file:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select "stmary.sms" in the *data files* folder for this tutorial and click **Open**.
3. If geometry is still open from a previous tutorial, SMS will ask if it should delete existing data. Click the **Yes** button.
4. If asked, click **Yes** at the warning prompt to overwrite current (default) materials.

The display will refresh with the mesh as shown in Figure 1. The mesh that is read in includes geometry (nodes and elements from a NET file), as well as material properties and boundary conditions (from a DAT file).

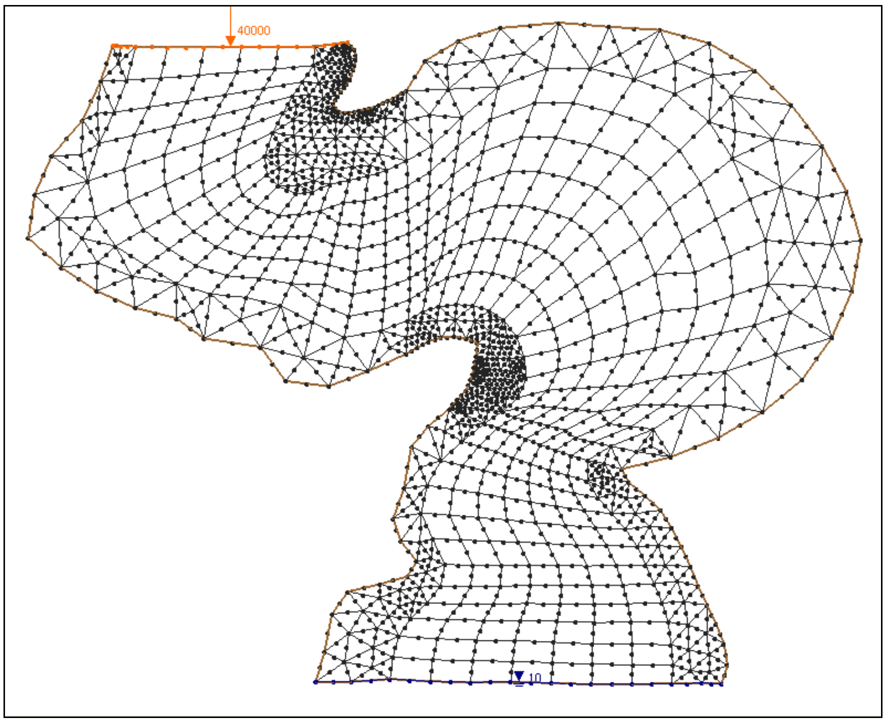


Figure 1 The mesh contained in "stmary.sms"

2 Converting Elements

For FESWMS, it is best to use 9-noded quadrilateral elements (quads) even though both 8-noded and 9-noded quads are supported. The mesh generation process from the conceptual model generates 8-noded quads to increase compatibility.

To convert these to 9-noded quads:

1. Select *Elements* | **QUAD8↔QUAD9**.

The screen will refresh and the quadrilateral elements will have 9 nodes. Since there was a change in the number of nodes, the mesh should be renumbered, even though it was renumbered before being saved.

2. Choose the **Select Nodestring** tool.
3. Click in the selection box at the downstream boundary condition (at the bottom of the screen).
4. Select *Nodestrings* | **Renumber Nodestrings**.
5. Click **OK** in the message that indicates the nodestrings have been renumbered.

3 Defining Material Properties

Each element in the mesh is assigned a material type. Each material type includes parameters for roughness, turbulence, and wetting/drying. The materials properties define how water flows through the element. These material properties must be changed for this analysis.

To edit the material parameters:

1. Select *FESWMS* | **Material Properties** to bring up the *FESWMS Material Properties* dialog.
2. Select the material "left_bank".
3. On the *Turbulence Parameters* tab, enter "50" for the kinematic eddy viscosity (*Vo)*, and "0.045" for both *Cu1* and *Cu2*.
4. Select the material "main_channel" from the list on the left.
5. On the *Roughness Parameters* tab, enter "0.03" as the roughness value (*n1* and *n2*). Scour is not a part of this tutorial, so ignore the other roughness values.
6. On the *Turbulence Parameters* tab, enter "50" for (*Vo*).
7. Select the material "right_bank".
8. On the *Roughness Parameters* tab, enter "0.04" for both *n1* and *n2*.
9. On the *Turbulence Parameters* tab, enter "100" for *Vo* (higher turbulence requires a higher viscosity value).
10. Click **OK** to close the *FESWMS Material Properties* dialog.

The kinematic eddy viscosity and Manning's roughness values should always be set. Other material properties can also be set for more advanced problems. See the FESWMS documentation for more information on these other material properties.

Optional: The materials can be displayed by opening the *Display Options* dialog and toggling on the *Materials* option. Be sure to turn the option back off before continuing with this lesson.

4 Setting Model Parameters

Before running an analysis, model controls and parameters must be set. The parameters and files used are specified in the *FESWMS Model Control* dialog.

To change the global display parameters:

1. Right-click on "Area Property" and select **Projection…** to bring up the *Object Projection* dialog.
2. In the *Horizontal* section, select *No Projection* and turn on *Units.*
3. Select "U.S. Survey Feet" from the *Units* drop-down.
4. In the *Vertical* section, select "U.S. Survey Feet" from the *Units* drop-down.
5. Click **OK** to close the *Object Projection* dialog.
6. Next, go to *Display* | **Projection** to open the *Display Projection* dialog.
7. In the *Horizontal* section, select *No Projection* and select "U.S. Survey Feet" from the drop-down.
8. In the *Vertical* section, select "U.S. Survey Feet" from the *Units* drop-down.
9. Click **OK** to close the *Display Projection* dialog.

To set the model controls:

1. Select *FESWMS* | **Model Control** to open the *FESWMS Model Control* dialog.
2. On the *General* tab in the *FESWMS Version* section, select *FESWMS 3.**.
3. In the *FST2DH Input* section, make sure the *NET file* option is the only box selected.
4. In the *Solution Type* section, select *Steady state*.
5. On the *Timing* tab, set *Iterations* to "10".
6. On the *Parameters* tab, set:
    - *Water-surface elevation* to "10.0".
    - *Unit flow convergence* to "0.005".
    - *Water depth convergence* to "0.001".
    - Turn on *Element drying / wetting*.
    - Leave all other options at the default settings.
7. On the *Print* tab, turn on *ECHO to screen*.

8. Click **OK** to close the *FESWMS Model Control* dialog.

5 Saving the Simulation

The boundary conditions (inflow rate and head at the outflow) were previously defined using the conceptual model. These were read in with the simulation. The entire simulation can now be saved.

To do this:

1. Select *File*| **Save As…** to open the *Save* As dialog.
2. In the *File name* field, enter "stmary_ready.sms" and click **Save**.

The model control options and boundary conditions are saved to the file "stmary_ready.dat", and the finite element network is saved to the file "stmary_ready.net". If desired, look at the file "stmary_ready.fpr" to see these filenames.

6 Running the Simulation

The analysis is now ready to be run. The analysis module of FESWMS is called "FST2DH" and it can be launched from inside SMS.

To launch the FST2DH program:

1. Select *FESWMS* | **Run FST2DH**, which performs two basic tasks:
   - Performing a model check to detect missed components.
   - Running the simulation.

If no problems are detected in the model check, this step produces no visible effects. If the model is missing a required component (e.g., no boundary conditions exist), or if there is an error in the simulation (e.g., an invalid mesh domain), a list of problems is posted for the user.

Once the check is complete, SMS launches the FST2DH executable. For this simulation, FST2DH should finish quickly.

2. Turn on *Load solution*. This will automatically load the solution file upon exiting the *FESWMS* dialog.
3. Click the **Exit** button to close the *FESWMS* model wrapper.

If running in Demo Mode, the solution "stmary_ready.flo" is found in the *data files/output* directory and can be opened with the *File* | **Open** command.

With the solution loaded, now evaluate the results by doing the following:

1. Click on the **Display Options** macro to open the *Display Option* dialog.
2. Select *2D Mesh* from the list on the left.
3. Turn on *Contours* and *Vectors* and turn off *Nodes*.
4. On the *Contours* tab in the *Contour method* section, select "Color Fill" from the drop-down.

5. On the *Vectors* tab, in the *Arrow Options* section, select “Scale length to magnitude” from the *Shaft Length* drop-down.
6. Click **OK** to close the *Display Options* dialog.

The Main Graphics Window should appear as similar to Figure 2.

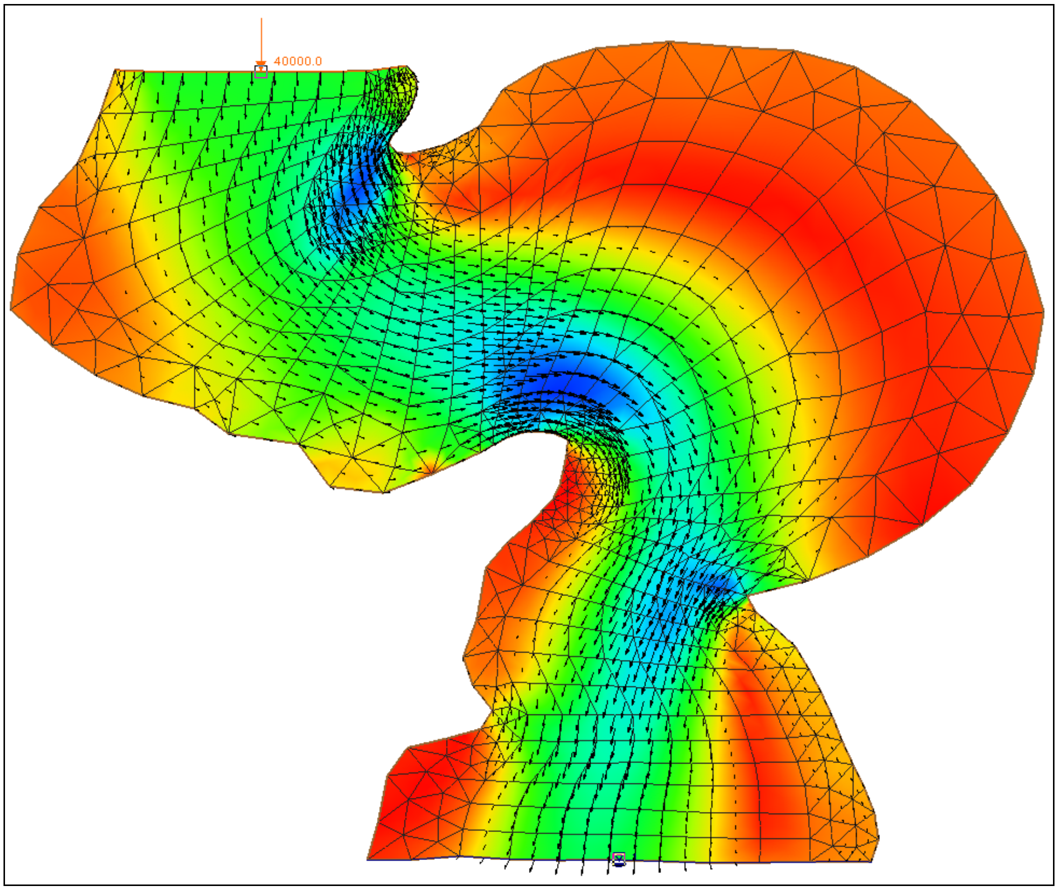


Figure 2 The final appearance of the FST2DH solution

The FST2DH solutions for velocity magnitude, water depth and water surface elevation can be viewed by selecting the desired dataset in the Project Explorer.

7 Conclusion

This concludes the *Basic FESWMS Analysis* tutorial.