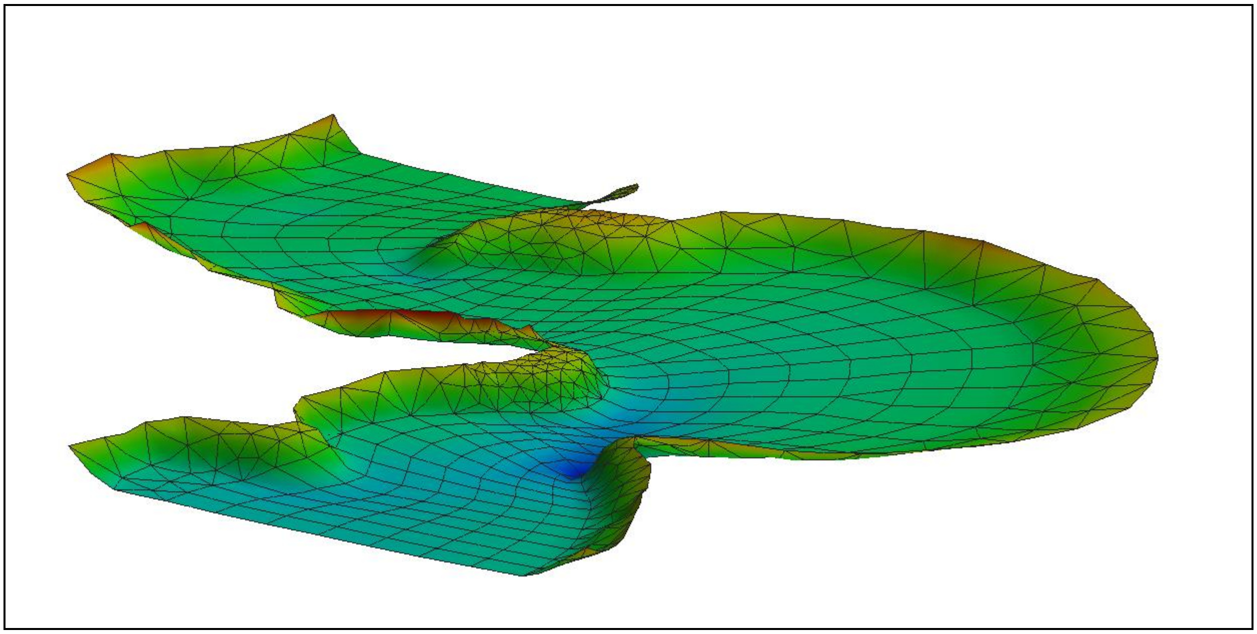
SMS 12.2 Tutorial

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 2
2 Converting Elements 3
3 Defining Material Properties 3
4 Setting Model Parameters 4
5 Saving the Simulation 5
6 Running the Simulation 6
7 Conclusion 7

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 “Project Files (*.sms)” from the *Files of type* drop-down.
3. Browse to the *data files* folder for this tutorial and select “stmary.sms”.
4. Click **Open** to import the project file and exit the *Open* dialog.
5. If geometry is still open from a previous tutorial, SMS will ask if it should delete existing data. Click **Yes**.
6. 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 imported mesh includes geometry (nodes and elements from a NET file), as well as material properties and boundary conditions (from a DAT file).

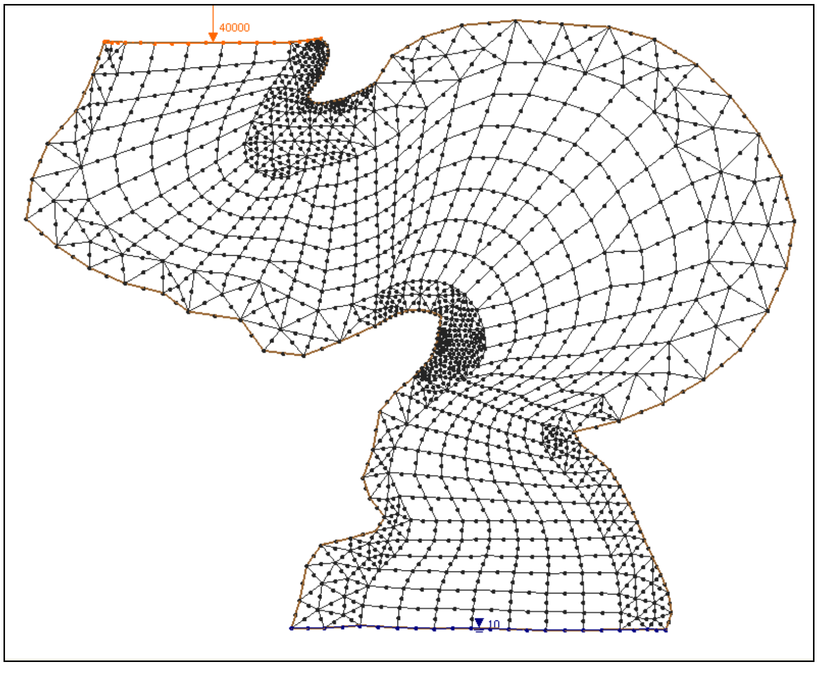


Figure 1 The mesh contained in “stmary.sms”

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 now 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. Using the **Select Nodestring** tool, click in the selection box at the downstream boundary condition (at the bottom of the screen, see Figure 2).
3. Select *Nodestrings* | **Renumber Nodestrings**.
4. Click **OK** when advised the nodestrings have been renumbered.

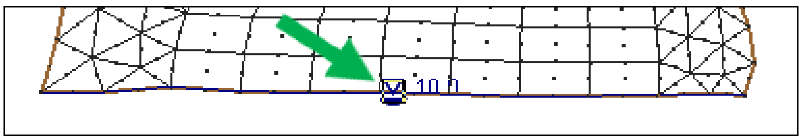


Figure 2 Selection box at the downstream boundary condition

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" from the list on the left.
3. On the *Turbulence Parameters* tab, enter "50.0" for *Vo* and "0.045" for both *Cu1* and *Cu2*.
4. Select "main_channel" from the list on the left.
5. On the *Roughness Parameters* tab, enter "0.03" for both *n1* and *n2*.

Scour is not a part of this tutorial, so ignore the other roughness values.

6. On the *Turbulence Parameters* tab, enter "50.0" for *Vo*.
7. Select "right_bank" from the list on the left.
8. On the *Roughness Parameters* tab, enter "0.04" for both *n1* and *n2*.

9. On the *Turbulence Parameters* tab, enter "100.0" 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 and other material properties.

The materials can optionally be displayed by opening the *Display Options* dialog and turning on the *Materials* option (Figure 3). Turn the option off before continuing with this tutorial.

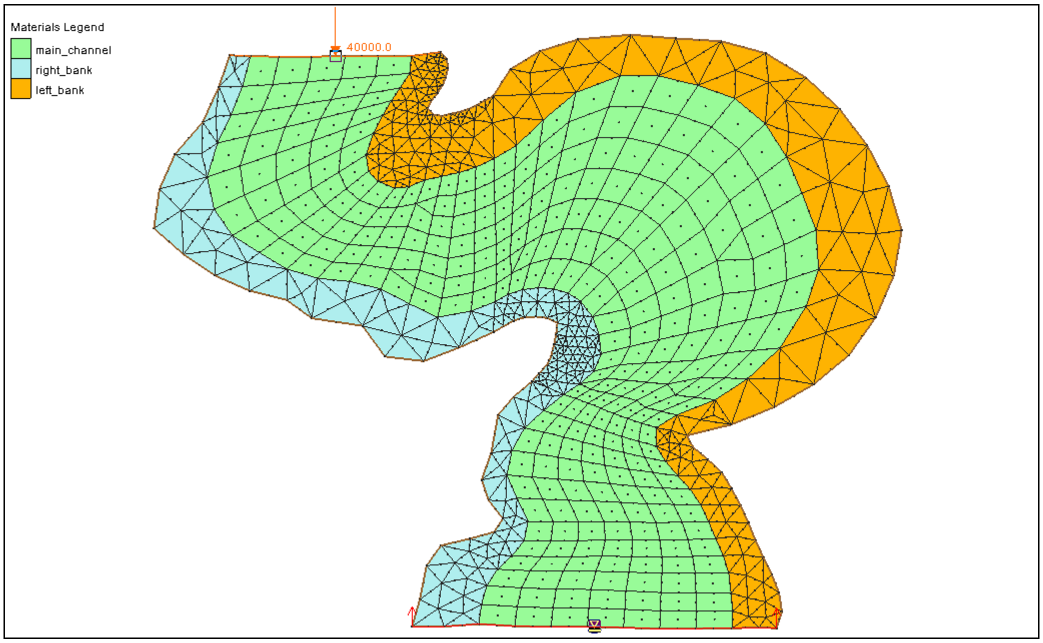


Figure 3 Project with Materials display option turned on

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 " 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 "Feet (U.S. Survey)" from the *Units* drop-down.
4. In the *Vertical* section, select "Feet (U.S. Survey)" from the *Units* drop-down.
5. Click **OK** to close the *Object Projection* dialog.
6. Select *Display* | **Projection…** to open the *Display Projection* dialog.

7. In the *Horizontal* section, select *No Projection* and select “Feet (U.S. Survey)” from the drop-down.
8. In the *Vertical* section, select “Feet (U.S. Survey)” 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, turn on *NET file* and turn off all other options in the section.
4. In the *Solution Type* section, select *Steady state*.
5. On the *Timing* tab, enter “10” as the *Iterations*.
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, in the *Extras* section, 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 imported with the simulation. The entire simulation can now be saved. This will generate an SMS project file, and SMS materials file, and a number of FESWMS files.

To save the project:

1. Select *File*| **Save As…** to open the *Save* As dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.
3. Enter “stmary_ready.sms” as the *File name.*
4. Click **Save** to save the project under the new name and close the *Save As* dialog.

These files are saved within a folder named for the mesh within a folder named for the project file name. For example, because the project is now named “stmary_ready.sms” and the mesh is named “stmary”, these files are saved within the *data files\stmary_ready\FESWMS\stmary* folder.

The FESWMS file names are based on the name of the mesh—in this case, “stmary”. For example, the model control options and boundary conditions file is “stmary.dat” and the finite element network is saved as "stmary.net". The file "stmary.fpr" contains a list of all the related FESWMS files.

6 Running the Simulation

The analysis is now ready to be run. The FESWMS analysis module—FST2DH—can be launched from inside SMS.

To launch the FST2DH program:

1. Select *FESWMS* | **Run FST2DH** to bring up the *Model Checker* dialog.

The model checker 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 and the *Model Checker* dialog will close automatically and open the *FESWMS* model wrapper dialog. 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 and suggested solutions is presented.

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

2. When the model is finished, turn on *Load solution*.

This automatically imports the solution file upon exiting the *FESWMS* model wrapper dialog.

3. Click **Exit** to close the *FESWMS* model wrapper dialog.
4. 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, evaluate the results by doing the following:

5. Click **Display Options** to open the *Display Option* dialog.
6. Select "2D Mesh" from the list on the left.
7. On the *2D Mesh* tab, turn on *Contours* and *Vectors* and turn off *Nodes*.
8. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the drop-down.
9. On the *Vectors* tab, in the *Arrow Options* section, select "Scale length to magnitude" from the *Shaft Length* drop-down.
10. Click **OK** to close the *Display Options* dialog.

The Main Graphics Window should appear similar to Figure 2.

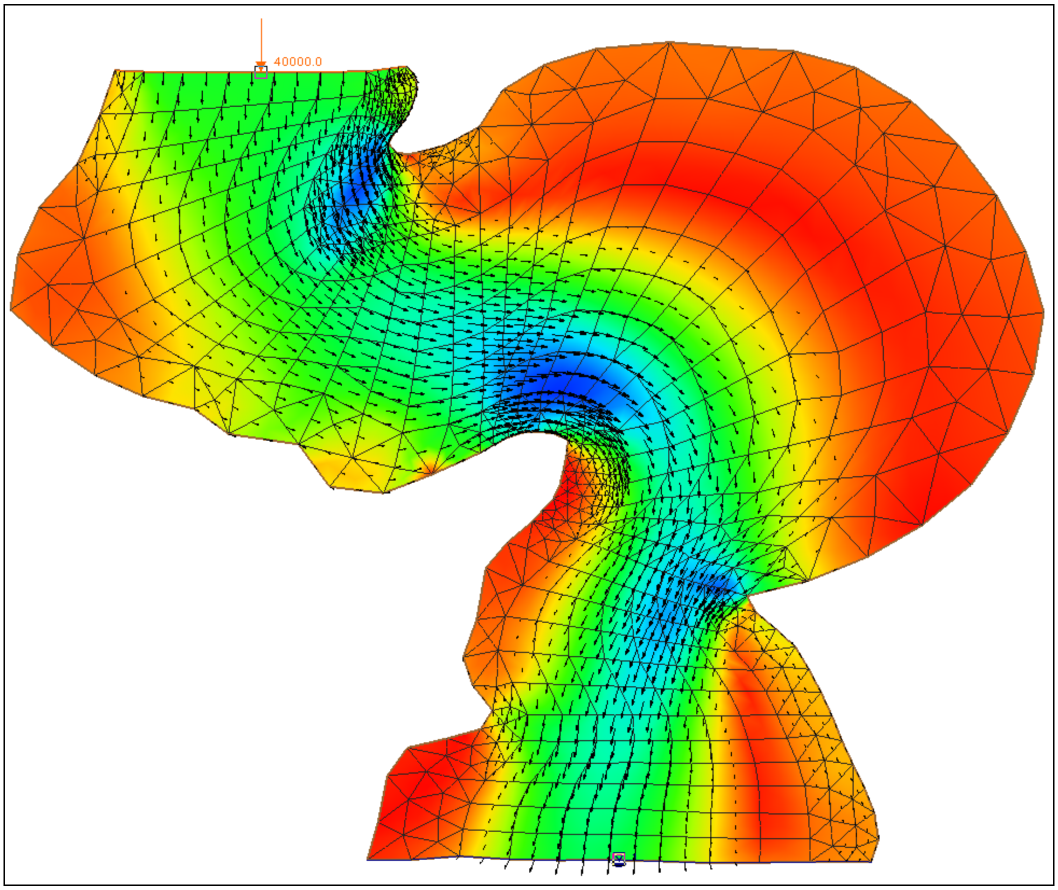


Figure 4 The final appearance of the FST2DH solution

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. Feel free to continue experimenting, or exit the program.

