

SURFACE WATER MODELING SYSTEM

Basic FESWMS Analysis

1 Introduction

This lesson will teach you how to prepare a mesh for a *FESWMS* simulation. You will be using the project file *stmary.sms*. The input file is included in the Data Files Folder for this tutorial. This project file includes a *FESWMS* project (*.*fpr*) file. It contains a list of filenames that are used by *FESWMS*. The actual input data is stored in the files named in the project file. To open the file:

1. Select *File / Open*.
2. Open the file *stmary.sms* supplied in the Data Files Folder. 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. The display will refresh with the mesh as shown in Figure 1.
3. If asked, click *Yes* at the warning prompt to overwrite current (default) materials.

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).

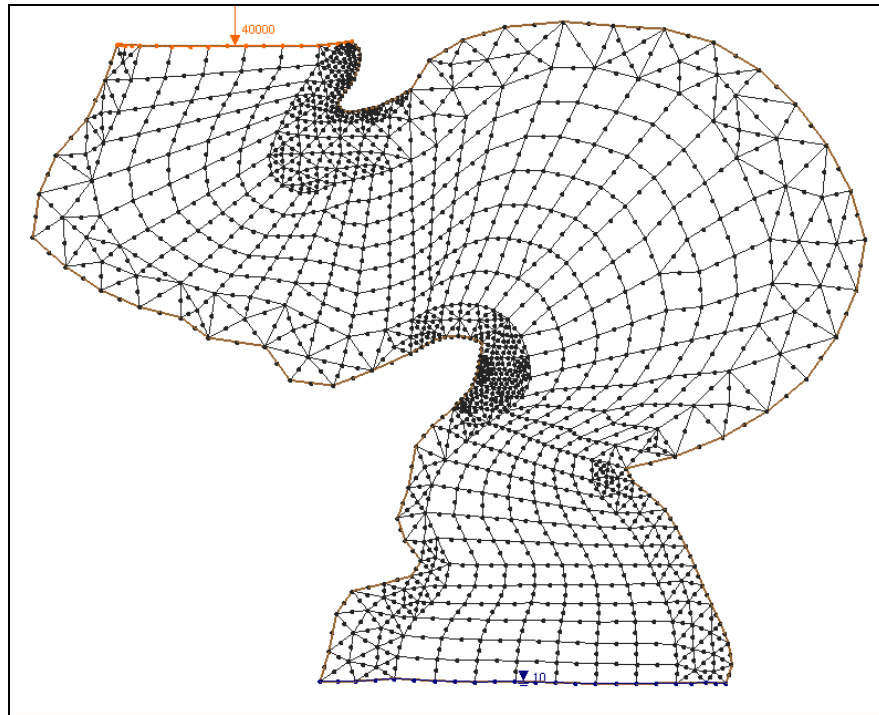



Figure 1. The mesh contained in *stmary.sms*.

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:

- 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. To do this:

1. Choose the *Select Nodestring*  tool from the *Toolbox*.
2. Click in the selection box at the downstream boundary condition (at the bottom of the screen).
3. Select *Nodestrings / Renumber*.

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. These material properties must be changed for this analysis. The materials properties define how water flows through the element. To edit the material parameters:

1. Select *FESWMS* | *Material Properties*.
2. In the *FESWMS Material Properties* dialog, select the material *main_channel*.
3. In the *Roughness Parameters* tab of the *FESWMS Material Properties* dialog, an image shows what the Manning's coefficients are for different depths. Enter 0.03 as the roughness value (n) for both depths. We are not dealing with scour in this problem, so we can ignore the other roughness values.
4. In the *Turbulence Parameters* tab of the *FESWMS Material Properties* dialog, enter a value of 50 for the kinematic eddy viscosity (V_o).
5. Highlight the material named *left_bank*. Set V_o to 50 and set $Cu1$ and $Cu2$ to 0.045 for both depths.
6. For the material named *right_bank*. Set V_o to 100 (higher turbulence requires a higher viscosity value) and set n in the *Roughness Parameters* tab to 0.04 for both depths.
7. Click the *OK* button 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 the *Materials* option on. If you do this, 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 parameters:

1. First, to set the units to English, go to *Edit* | *Projection*.

-
2. Make sure the *Horizontal System* is *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*. Press *OK* to exit the *Current Coordinates* dialog.
 3. Select *FESWMS | Model Control*.
 4. In the *FESWMS Version* section (under the *General* tab), make sure *FESWMS 3.** is selected.
 5. In the *FST2DH Input* section, make sure the *NET file* option is the only box selected.
 6. Make sure the *Solution Type* is set to *Steady state*.
 7. Click the *Parameters* tab and set the values shown below:
 - Water-surface elevation = 10.0
 - Unit flow convergence = 0.005
 - Water depth convergence = 0.001
 - Element drying / wetting = ON
- Leave the other defaults.
8. Click the *Timing* tab and change the number of *Iterations* to 10.
 9. Click the *Print* tab and make sure the *ECHO to screen* option is turned on.
 10. Click the *OK* button to exit 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...*
2. Enter the name *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

You are now ready to run the analysis. 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*.

This command performs two basic tasks. These are:

1. Performing a model check to detect missed components. If no problems are detected, this step produces no visible effects. If the model is missing a required component (for example, if no boundary conditions exist), or if there is an error in the simulation (such as an invalid mesh domain), a list of problems is posted for the user.
2. Running the simulation. Once the check is complete, *SMS* launches the *FST2DH* executable. The location of the executable is stored as a model preference. The progress of the model is displayed in the *Model Wrapper*.

For this simulation, *FST2DH* should finish quickly. The *Model Wrapper* dialog waits for the user to acknowledge the completion of the model run. Make sure that *Load solution* is turned on. It will automatically load the solution file when the *Exit* button is clicked. (If you are running in *Demo Mode*, the solution *stmary_ready.flo* is found in the *tutorials/SMS_FESWMS/output* directory and can be opened with the *File/Open* command.)

With the solution loaded, you are ready to evaluate the results. To do this:

1. Open the *Display Options*  dialog.
2. Under the *2D Mesh* tab, turn on *Contours* and *Vectors* and turn off *Nodes*.
3. Under the *Contours* tab, select *Color Fill* as the *Contour Method*.
4. Under the *Vectors* tab, select *Scale length to magnitude* as the option for *Shaft Length*. Click *OK* to close the *Display Options* dialog.

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

7 Conclusion

This concludes the *Basic FESWMS Analysis* tutorial.