SURFACE WATER MODELING SYSTEM

FESWMS Steering (Incremental Loading)

1 Introduction

This lesson will teach you how to use the steering module to perform incremental loading. Incremental loading has also been referred to as spinning down or revising the model. The process involves repetitively running the model with boundary conditions getting closer to the desired values. The steering module is used to automate the process. The methodology applies to both *FESWMS* and *RMA2*. *FESWMS* is used in this situation. The geometry has already been created and renumbered. To open the file:

1. Select *File* | *Open*.

2. Open the file *Capitol_Reef.fpr* from the Data Files Folder for this tutorial. 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 geometry data will open, as shown in Figure 1.

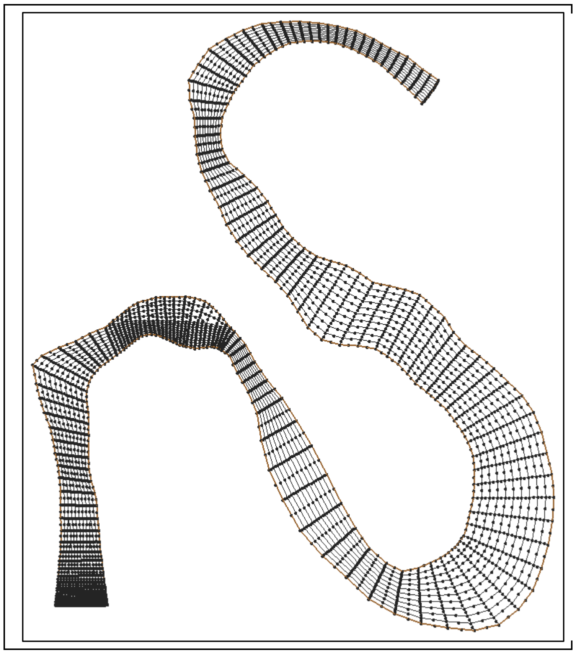

Figure 1. The mesh contained in the file Capitol_Reef.fpr.

2 Specifying Model Units

Before continuing, make sure that the units are set to Customary (feet). To do this:

1. Select *Edit | Projection*.
2. Make sure the *Horizontal System* is set to *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*.
3. Click *OK* to exit the dialog.

3 Defining Model Parameters

Several model control parameters must be assigned to define the state of the model. These model parameters include items such as the input and output files, how to handle wetting and drying, the convergence parameters, and the number of iterations to be performed by *FESWMS*. Additional information on these parameters is found in the *FESWMS Help* and the *FESWMS* documentation. To define the model parameters:

- Select *FESWMS | Model Control*.

This opens the *FESWMS Model Control* dialog, in which the model parameters are controlled. This dialog contains various items such as simulation titles that describe what is being modeled. To set a title:

1. Click in the Network Stamp edit field and type, “Capitol Reef National Park.”

2. Click in the BC Descriptor edit field and type, “50 Year Flood.”

Input and output files may also be managed in the main dialog including the option to use ini files which are used to hotstart the model. This is desirable in complicated networks that require several steps to arrive at a solution. However we will use a different method that automates this process in this tutorial.

Before continuing, make sure that the solution type is set to *Steady state*.

3.1 Iterations

The *Timing* tab contains options for defining the relaxation factor, number of iterations, and time steps in a dynamic model. The relaxation factor is an advanced option that can be adjusted to improve how fast the solution will converge. It will not affect the final results. We will use the default value, so the only applicable parameter is the number of iterations. To set this value:

1. Click the *Timing* tab.

2. Enter 25 as the number of iterations.

3.2 Parameters

The *Parameters* tab is used to set the general parameters of the model. We need to set the initial water surface elevation and the convergence parameters. To set these parameters:

1. Click the *Parameters* tab.

2. Set the *Water-surface elevation* to 5070 ft.

3. Set the *Unit flow convergence* to 0.05 and the *Water depth convergence* to 0.005.

To accept all of the above values:

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

4 Defining Boundary Conditions

For this tutorial, flowrate and water surface elevation will be defined along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where water is allowed to enter or exit. Generally for *FESWMS*, a flowrate is specified across inflow boundaries and water surface elevation is specified across outflow boundaries. Other available boundary conditions are rating curves and reflecting boundaries.

This model has one inflow boundary and one outflow boundary so two nodestrings must be created. These boundaries are highlighted in Figure 2.

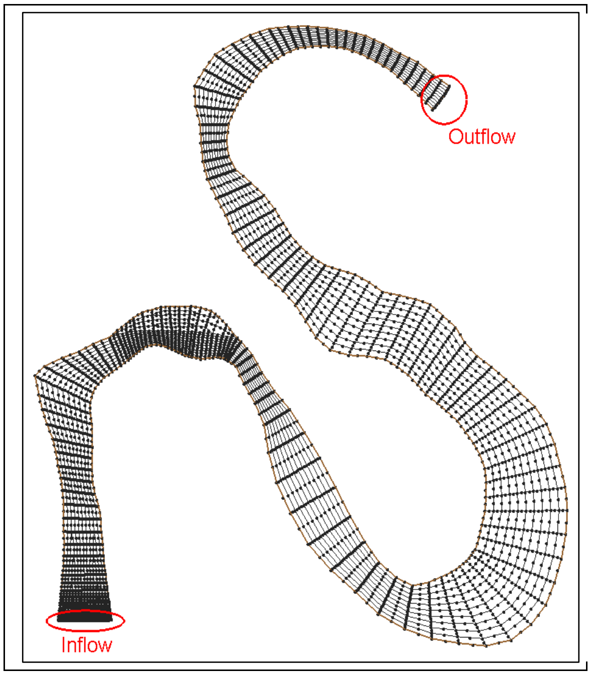

Figure 2. Position of the boundary nodestrings in the mesh.

4.2 Creating Nodestrings

Nodestrings should be created from right to left when looking downstream and the first nodestring should be that which spans the whole river section. In this case both nodestrings span the entire river section so it does not matter which nodestring is created first. To create the outflow nodestring:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the lower node at the outflow boundary (you may need to zoom in).
3. Hold the *SHIFT* key and double-click on the upper node at the outflow boundary to create and end the nodestring.
4. Create a nodestring across the inflow boundary. Make sure you create it right to left when looking downstream.

4.3 Defining Flow Boundary Conditions

To assign the flow condition:

1. Choose the *Select Nodestrings* tool from the *Toolbox*. An icon appears at the center of each nodestring.

2. Select the inflow nodestring (bottom of the mesh) by clicking on the icon.

3. Select *FESWMS | Assign BC*.

4. Make sure the *Boundary Type* is set to *Specified Flow / WSE* and select the *Flow* checkbox.

5. Assign a constant *Flowrate* of 6,550 (cfs).

6. Click the *OK* button to assign the boundary condition.

This defines the bottom nodestring to be an inflow boundary condition.

4.4 Defining Head Boundary Conditions

A water surface elevation (head) boundary condition will be assigned to the outflow boundary nodestring. To assign this boundary condition:

1. Select the outflow nodestring.

2. Select *FESWMS | Assign BC*.

3. Make sure the *Boundary Type* is set to *Specified Flow / WSE* and select the *Water surface elevation* checkbox.

4. Assign a constant value of 5,070 (feet).

5. Click the *OK* button to assign the boundary condition.

5 Defining Material Properties

Each element in the mesh is assigned a material type ID. This particular geometry has five material types. To see each of these materials:

1. Select *Display | Display Options* or click the *Display Options* macro.

2. In the *2D Mesh* tab, turn on the *Materials* option.

3. Turn off the *Nodes* and *Elements* options.

4. Click the *OK* button to close the *Display Options* dialog.

The display should look something like Figure 3. Most of the model is made of brush floodplain and the channel, but there are a few elements with other material types.

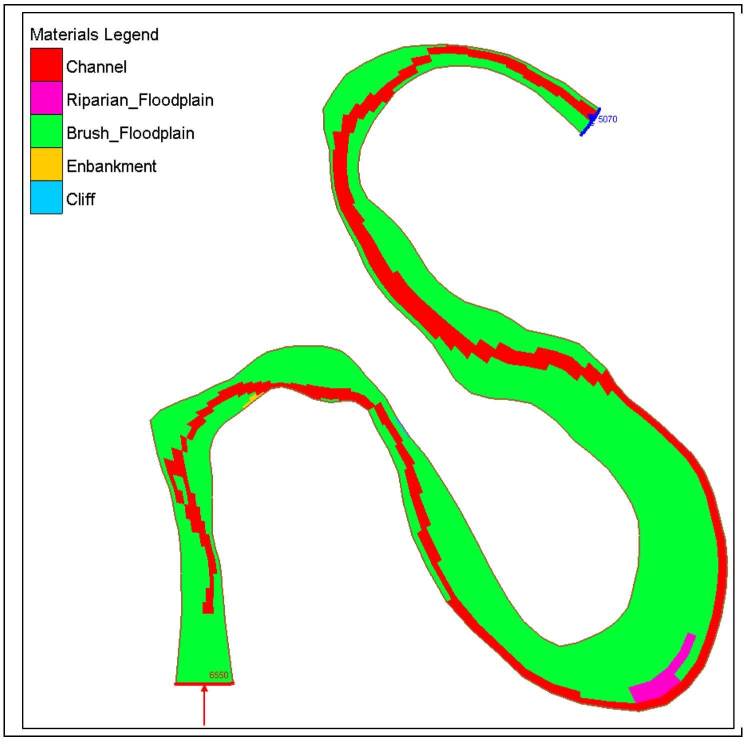

Figure 3. The display of materials.

Before continuing, turn off the material display. To do this:

- Open the *Display Options* dialog. Turn off the *Materials* option and turn the *Elements* options back on.

The materials were created with default parameters that must be changed for this particular simulation. The material properties define how water flows through the element. To edit the material parameters:

1. Select *FESWMS* | *Material Properties*.
2. In the *FESWMS Materials Properties* dialog, highlight *Brush Floodplain*.
3. Under the *Roughness Parameters* tab, set both n1 and n2 to 0.05.
4. Under the *Turbulence Parameters* tab, make sure the eddy viscosity (Vo) is set to 20 for this material.
5. Set the roughness for Channel to 0.03, Cliff to 0.05, Embankment to 0.04, and Riparian Floodplain to 0.1. Make sure you set both n1 and n2. The viscosity for each material should already be set to 20.
6. Click the *OK* button to close the *FESWMS Material Properties* dialog.

The eddy viscosity and roughness parameters have now been defined for this model.

6 Saving the Simulation

Before running the FST2DH engine, the data must be written to disk. The FST2DH files are written when saving an SMS project or when saving a FST2DH simulation directly. Saving a SMS project file is generally preferred because it will store your scatter data, map coverages, and display settings as well as the data specifically used by FST2DH.

To save the SMS project:

1. Select *File* | *Save As*.

2. Enter "CR_Sim" as the *File name*.

3. Click the *Save* button to save the project.

7 Running the Model

At this point, you are ready to try running FST2DH. To do this, choose *FESWMS* | *Run FST2DH*. Before SMS launches the model, a quick check is done on the data to make sure everything is valid. This model check will bring up the dialog shown in Figure 4 if any anomalies are detected. For this model, three warnings should be detected.

The first warning says that the elements might dry out, so the wet/dry flag should be turned on. To do this:

1. Select the *Cancel* button to leave *the Model Checker* without running *FST2DH*.

2. Select *FESWMS* | *Model Control*.

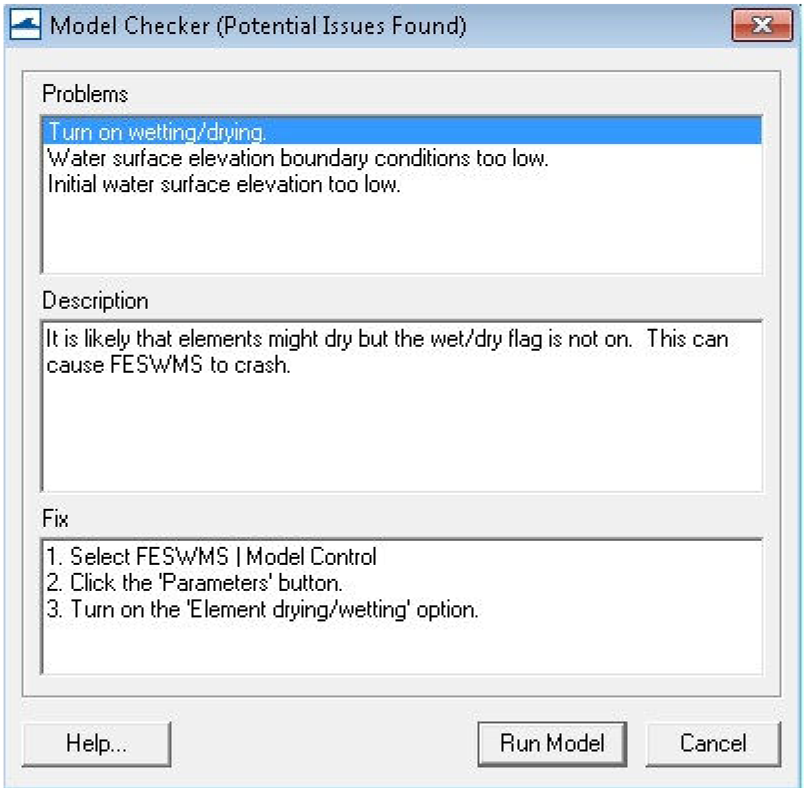

Figure 4. Warning in FESWMS data.

1. Click the *Parameters* tab.
2. Set the *Default storativity depth* to 0.1 ft.
3. Turn on *Element drying / wetting*, and set the *Depth tolerance for drying* to 0.1 ft.
4. Click the *OK* button to close the *FESWMS Model Control* dialog.
5. Resave the project, and select the *FESWMS | Run FST2DH* command again to launch the *Model Checker* and assure that the first error has gone away.

The remaining warnings say that the initial water surface elevation and the WSE boundary condition for the simulation are too low and will leave portions of the domain dry. This can lead to instabilities and cause FST2DH to crash. One option is to raise the initial water surface elevation, but if it is much higher than the outflow boundary elevation, instabilities can develop at those locations. If the simulation is not stable at the WSE boundary condition, it may be best to use incremental loading as will be demonstrated later. For now we will ignore these warnings. Because there is still a warning message, FST2DH might not converge. However, before attempting to fix this, the simulation will be tried.

- Click on *Run Model* to launch FST2DH.

FST2DH will run iteration 1 and diverge. At the bottom of the *FESWMS Output* window is the text "***Run ended abnormally because of 1 errors and 0 warnings!" as shown in Figure 5. This is how *FST2DH* declares that it has not successfully converged.

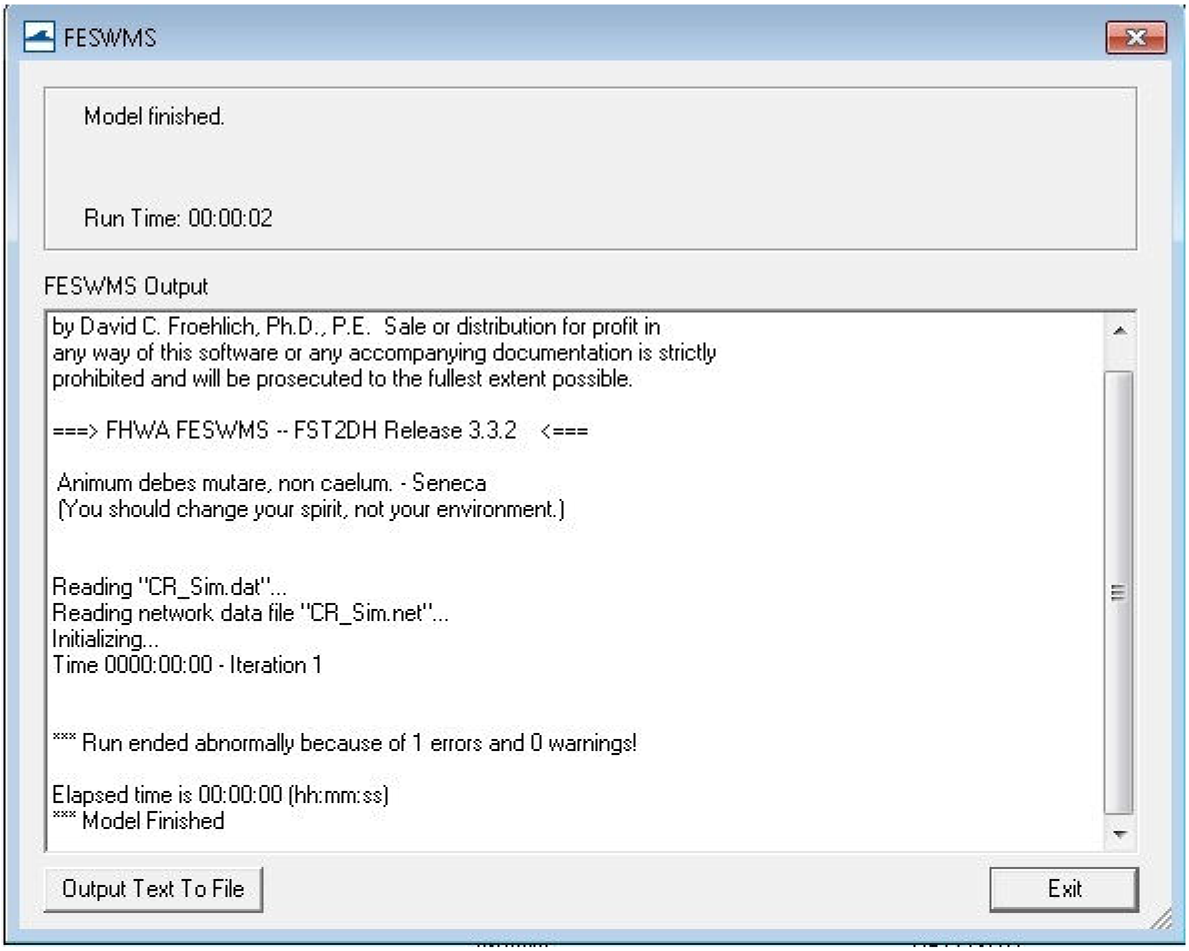

Figure 5. Output from running CR_Sim.fpr.

- Click the Exit button to close the *FESWMS Output* window.

Various things can contribute to a model not converging. In this case, *SMS* had given an error message that the initial water surface was too low. The low water surface elevation for this simulation does not allow *FST2DH* to converge from the coldstart simulation. To illustrate why this occurs, we will compare the boundary condition with the bathymetry. To do this:

1. Select *File* | *Get Info* or click the *Get Info* macro.

3. In the *Mesh Module* tab, look at the *Max Z value*, indicated in Figure 6.

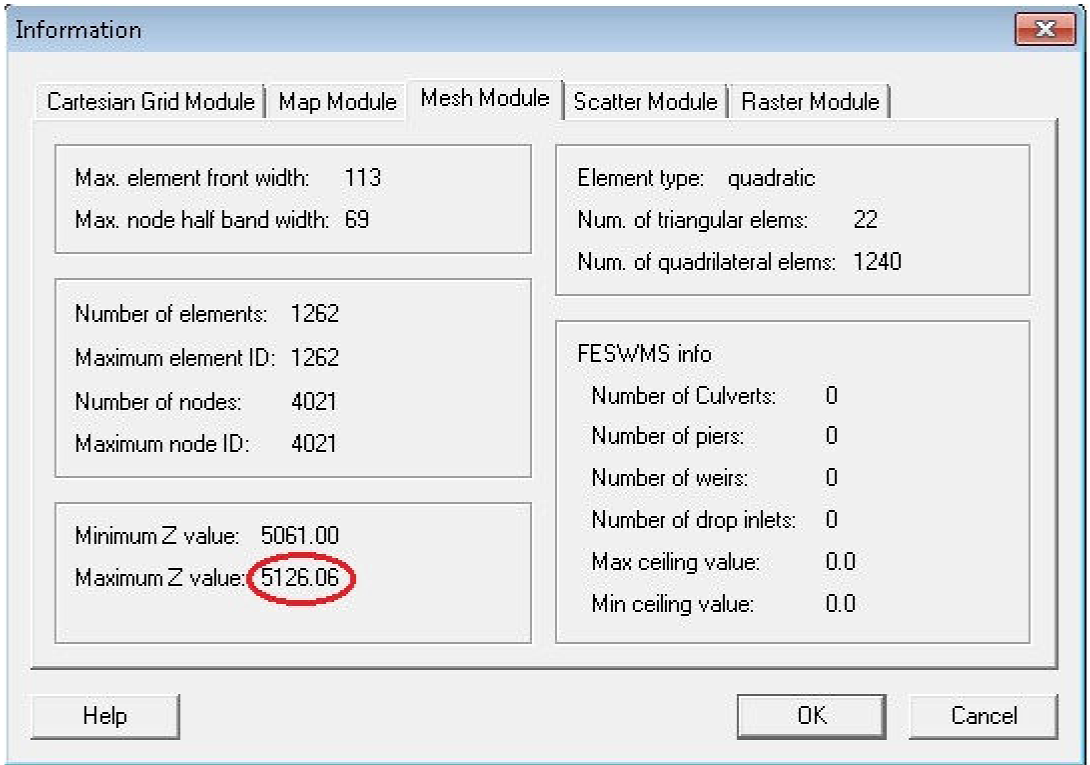

Figure 6. Mesh Information dialog.

You can clearly see that the maximum bathymetry elevation is well above our boundary condition of 5,070 ft. This means not all of the nodes are wet with our initial water surface elevation. FST2DH requires that all nodes be wet for the initial condition, or the model will not run. This is why FST2DH diverged after only running the first iteration.

- Close the *Information* dialog by clicking OK.

8 Using the Steering Module

It is possible to set the boundary water surface elevation high enough to wet all nodes. The model could then be run and an output solution file could be created. This solution file could be used to hotstart the model with a lower boundary water surface elevation. This process could then be repeated until the boundary water surface elevation is at the desired level, but that would require an enormous amount of user input and time, especially for this model. Fortunately, this process can be automated with no user-interaction by using the steering module. To run the steering module:

1. Select Data | Steering Module.
2. In the steering wizard, make sure that FESWMS Spindown is selected, then click Next.
3. Toggle on the option “Delete intermediate steering files.”

4. In the *Steering Module* dialog select *Water Surface Elevation* for the *FST2DH Spindown* method as shown in Figure 7.

5. Click *Start.*.

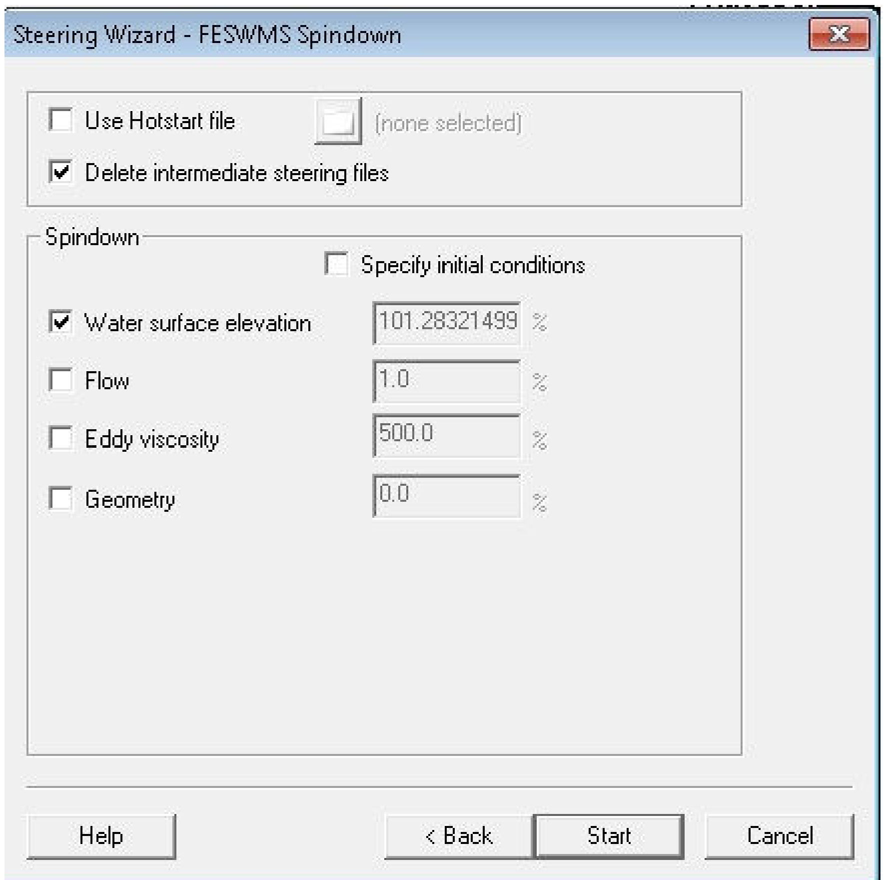

Figure 7. Steering Module dialog.

The FESWMS Spin-down dialog shown in Figure 8 will keep you updated on the progress of spinning down the model.

The top window in the dialog explains the WSE convergence of the current run. Each iteration is shown as a green point, and you can tell if the run is converging or diverging depending on if the points are moving towards or away from 0 head change. The iteration being performed is shown just above this plot.

The bottom window shows the overall spin down of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, then the model is finished. This percent is shown just above this plot. When Figure 8 was captured the model had spun down 23.8% and was on *Iteration* 10 of *Run Number* 31.

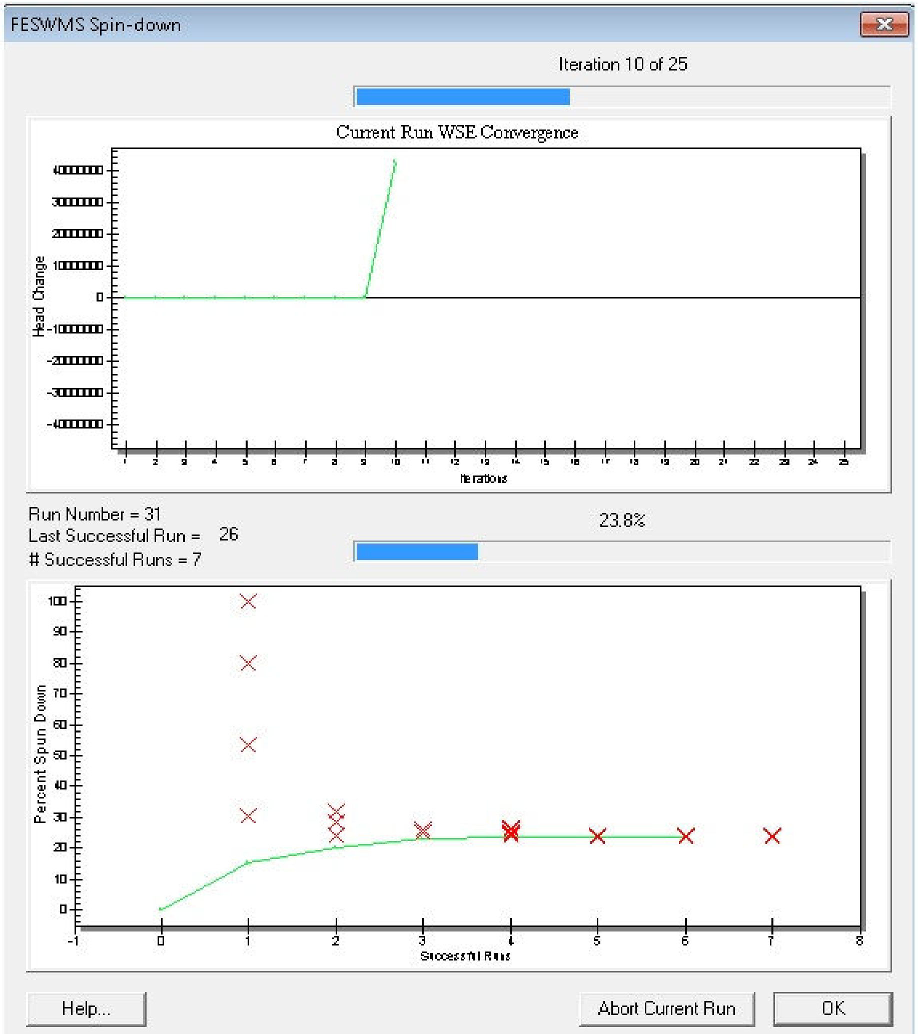

Figure 8. FESWMS Spin-down dialog.

9 Opening the Solution

The entire spindown process takes a few minutes depending on the speed of your computer. When it has finished, a window will open telling you that the "Steering process has terminated – See status file for details". This status file is named "SteeringStatus.txt" and gives a summary of the steering process. A final solution file has also been created. To view the solution of the spindown:

1. Click OK to exit the *FESWMS Spin-down* dialog.

6. Select File | Open and open the solution file named "CR_Sim.flo" from the Output file.

7. Review and post-process the results as desired.

10 Conclusion

This concludes the *FESWMS Steering (Incremental Loading)***Error! Reference source not found.** tutorial.

