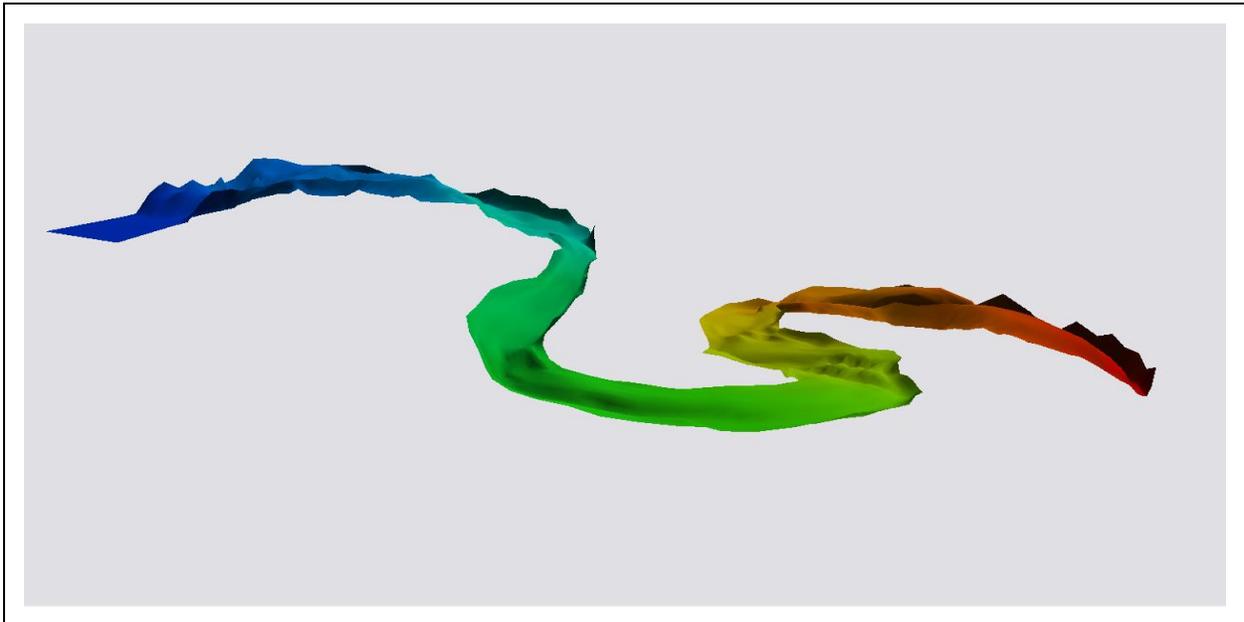*SMS 12.1 Tutorial*
FESWMS – Steering (Incremental Loading)

Objectives

This lesson will teach 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.

Prerequisites
- Overview Tutorial

Requirements
- FESWMS
- Fst2dh
- Mesh Module

Time
- 30-45 minutes

AQUAVEO™

1 Getting Started

Start by opening the file:

1. Select *File* / **Open** to bring up the *Open* dialog.

2. Select the file "Capitol_Reef.fpr" from the *data files* folder for this tutorial and click **Open**.

3. If geometry is still open from a previous tutorial, SMS will ask existing data should be deleted. If this happens, click the **Yes** button.

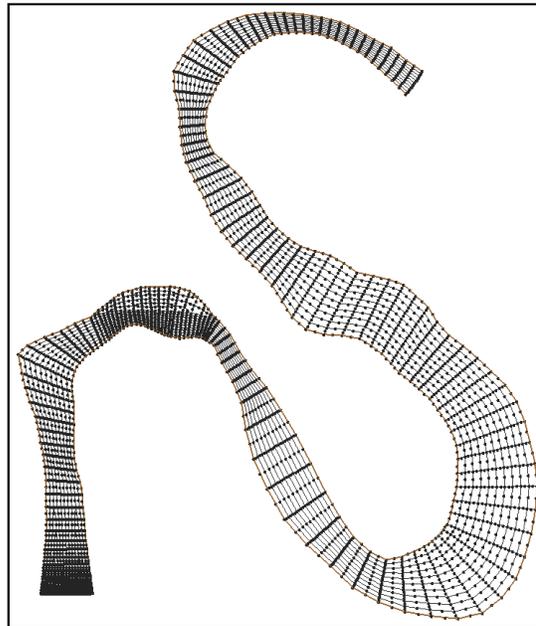The geometry data will open, as shown in Figure 1.

Figure 1   The mesh contained in Capitol_Reef.fpr

2   Specifying Model Units

Set the units and projection as follows:

1. Right-click on "Capitol_Reef" in the Project Explorer and select **Projection…** to bring up the *Object Projection* dialog.

2. Turn on *No projection.*

3. Turn on *Units under Horizontal* and set the units to "U.S. Survey Feet" in both the *Horizontal* and *Vertical* sections.

4. Click **OK** to close the *Object Projection* dialog when done.

5. Follow steps 1–4 for the "Area Property" coverage.

6. Select *Display* | **Projection** to bring up the *Display Projection* dialog.

7. Set *Units* under both *Horizontal* and *Vertical* to "U.S. Survey Feet".

8. Click **OK** to exit the *Display Projection* 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.

Input and output files may also be managed in the *FESWMS Model Control* dialog, including the option to use INI files which are used to hot start the model. This is desirable in complicated networks that require several steps to arrive at a solution. This tutorial uses a different method that automates this process.

Additional information on these parameters and on all of the tabs in the *FESWMS Model Control* dialog is found in the *FESWMS Help* and the FESWMS documentation.

3.1   General Settings

To define the general model parameters:

1. Select *FESWMS* / **Model Control** to open the *FESWMS Model Control* dialog.

2. On the *General* tab, click in the *Network Stamp* field and type, "Capitol Reef National Park."

3. In the *BC Descriptor* field and type, "50 Year Flood."

4. In the *Solution Type* section, select the *Steady state* radio button.

5. Continue to the next section, below.

3.2   Iterations

The *Timing* tab contains options for defining the relaxation factor, number of iterations, and time steps in a dynamic model. The *Relaxation factor* field is an advanced option

that can be adjusted to improve how fast the solution will converge. It will not affect the final results. Use the default value, so the only applicable parameter is the number of iterations.

To set the iterations:

6. Click on the *Timing* tab.

7. Enter "25" in the *Iterations* field.

8. Continue to the next section, below.

3.3   Parameters

The *Parameters* tab is used to set the general parameters of the model, the initial water surface elevation, and the convergence parameters.

To set these parameters:

9. Click the *Parameters* tab.

10. Set *Water-surface elevation* to "5070".

11. Set *Average water density* to "1.937".

12. Set *Unit flow convergence* to "0.05".

13. Set *Water depth convergence* to "0.005".

14. 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. For FESWMS, a flowrate is generally 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.
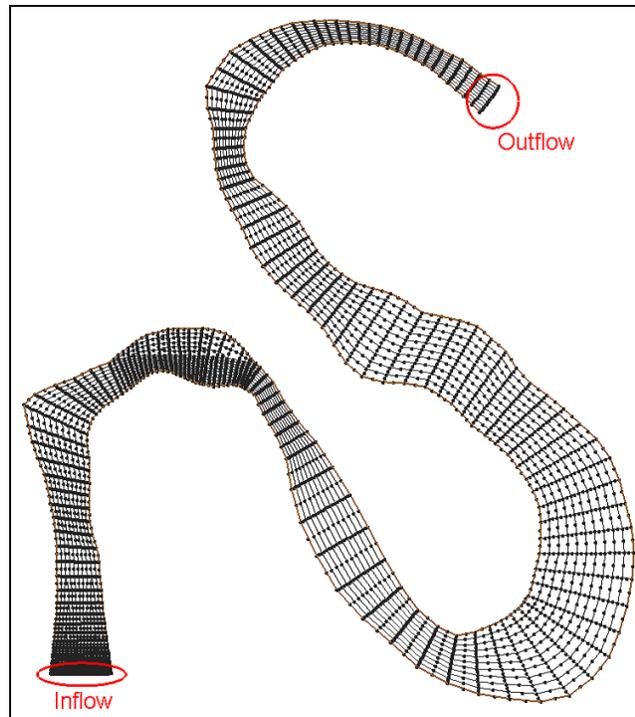
Figure 2   Position of the boundary nodestrings in the mesh

4.1   Creating Nodestrings

Nodestrings should be created from right to left when looking downstream (as if standing in the river and facing 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. Select the **Create Nodestring** ⌒ tool.

2. Start the nodestring by clicking on the lower node at the outflow boundary (since it would be on the right if facing downstream). Use the **Zoom** ⌕ tool if needed.

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 to create it right to left when looking downstream.

4.2   Defining Flow Boundary Conditions

Define the bottom nodestring with an inflow boundary condition as follows:

1. Choose the **Select Nodestrings** ⌖ tool. 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** to bring up the *FESWMS Nodestring Boundary Conditions* dialog.

4. Set *Boundary type* to "Specified Flow / WSE".

5. Turn on *Flow*.

6. Select *Constant* and *Normal* in the two drop-down boxes.

7. Set *Flow rate* to "6550" (cfs).

8. Click **OK** to close the *FESWMS Nodestring Boundary Conditions* dialog.

4.3   Defining Head Boundary Conditions

A water surface elevation (head) boundary condition will be assigned to the outflow boundary nodestring by doing the following:

1. Select the outflow nodestring.

2. Right-click and select **Assign BC** to bring up the *FESWMS Nodestring Boundary Conditions* dialog.

3. Set *Boundary type* to "Specified Flow / WSE".

4. Turn on *Water surface elevation*.

5. Select *Constant* and *Essential* in the two drop-down boxes.

6. Set WSE to "5070" (feet).

7. Click **OK** to close the *FESWMS Nodestring Boundary Conditions* dialog.

5   Defining Material Properties

Each element in the mesh is assigned a material type ID. This particular geometry has five material types that are viewable by doing the following:

1. Select *Display* | **Display Options** or click the **Display Options** 🔹 macro to open the *Display Options* dialog.

2. Select "2D Mesh" from the list on the left.

3. Turn on the *Materials* option.

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

5. Click **OK** 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.
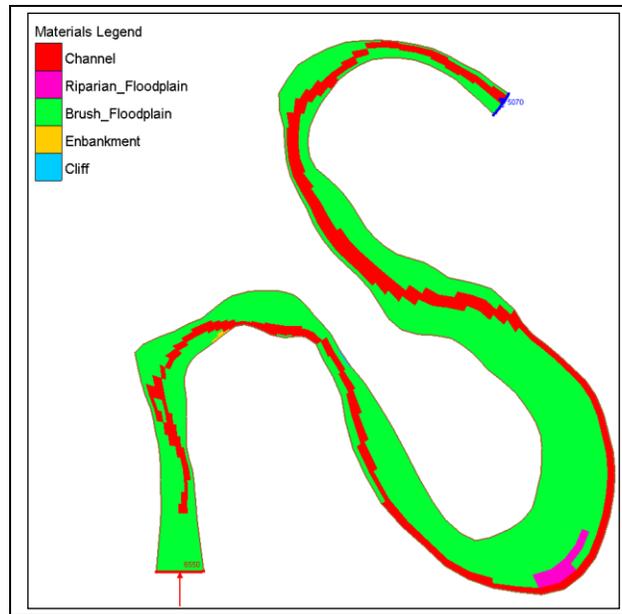
Figure 3   The display of materials

Before continuing, turn off the material display by doing the following:

6. Select *Display* | **Display Options** to open the *Display Options* dialog.

7. Select "2D Mesh" from the list on the left then turn off *Materials* and turn on *Elements*.

8. Click **OK** to close the *Display Options* dialog.

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

To define the eddy viscosity and roughness parameters for this model:

1. Select *FESWMS* | **Material Properties** to bring up the *FESWMS Materials Properties* dialog.

2. Select "Brush Floodplain" from the list on the left.

3. Set the roughness (both *n1* and *n2*) on the *Roughness Parameters* tab, and the eddy viscosity (*Vo*) on the *Turbulence Parameters* tab, as indicated in the table below.

4. Repeat for each of the other material types in the *Material* column in the table below.

5. Click **OK** to close the *FESWMS Materials Properties* dialog.

Material	*Roughness Parameters, n1* **and** *n2*	*Turbulence Parameters, Vo*
Brush_Floodplain	0.05	20
Channel	0.03	20
Cliff	0.05	20
Embankment	0.04	20
Riparian_Floodplain	0.1	20

6   Saving the Simulation

Before running the FST2DH engine, the data must be saved by saving an SMS project or when saving a FST2DH simulation directly. Saving an SMS project file is generally preferred because it will store the 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** to open the *Save As* dialog.

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

3. Select "Project Files (*.sms)" for *Save as type*.

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

7   Running the Model

To run FST2DH:

1. Select *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 *Model Checker* 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 (Figure 4), so the wet/dry flag should be turned on by doing the following:

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

2. Select *FESWMS* / **Model Control** to open the *FESWMS Model Control* dialog.

3. Click the *Parameters* tab.

4. Set the in the *General Parameters* section, set *Default storativity depth (ft)* to "0.1".

5. Turn on the *Element drying / wetting* option.

6. Set *Depth tolerance for drying (ft)* to "0.1".

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

8. Resave the project (*Ctrl-S*).

9. Select *FESWMS* / **Run FST2DH** to launch the *Model Checker* and verify the first error has gone away.
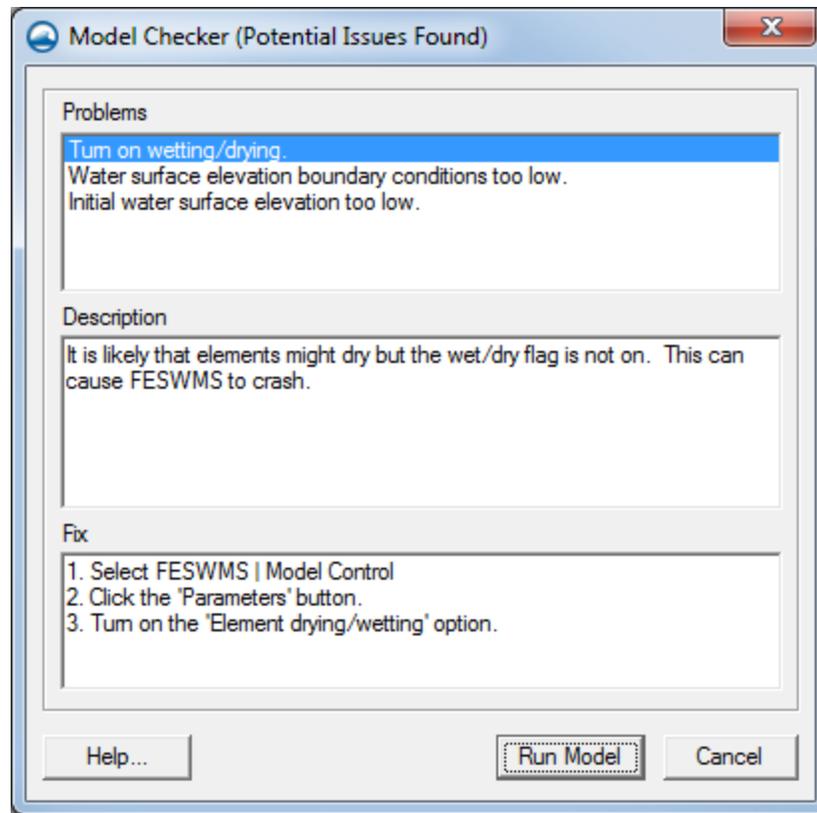
Figure 4   Warning in FESWMS data

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, 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.

10. Click **Run Model** to launch FST2DH.

FST2DH will run iteration 1 and diverge. Near the bottom of the *FESWMS Output* section in the FESWMS dialog 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.

11. Click the **Exit** button to close the *FESWMS* 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 cold start simulation.
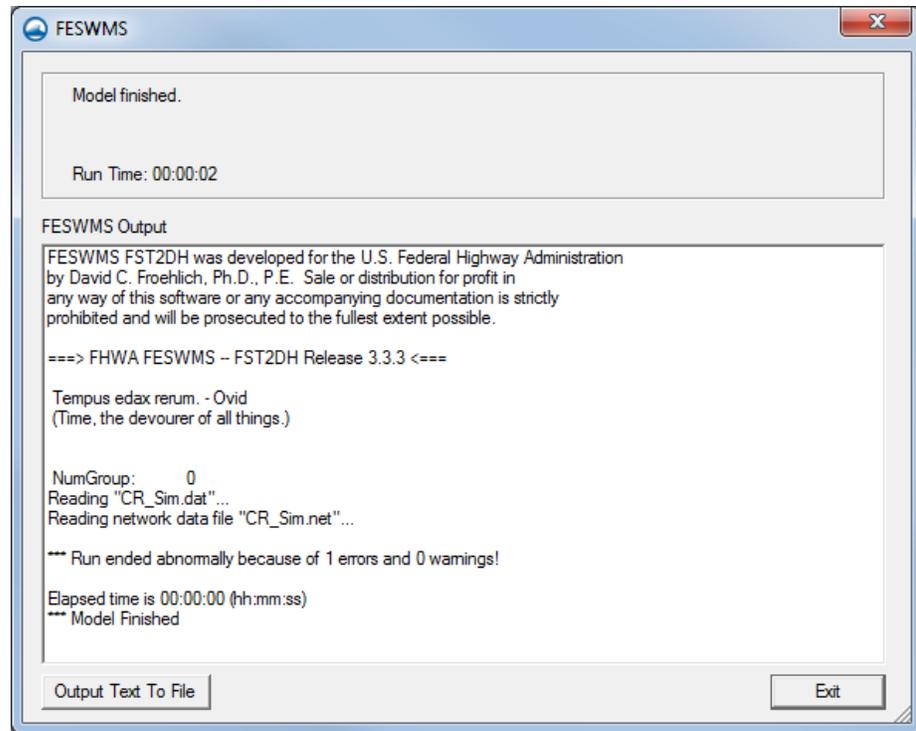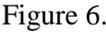
Figure 5   Output from running CR_Sim.fpr

To illustrate why this occurs, compare the boundary condition with the bathymetry:

1. Select *File* / **Get Info,** or click the **Get Module Info** macro, to bring up the *Information* dialog.

2. On the *Mesh Module* tab, look at the *Maximum Z value*, indicated in   Figure 6.
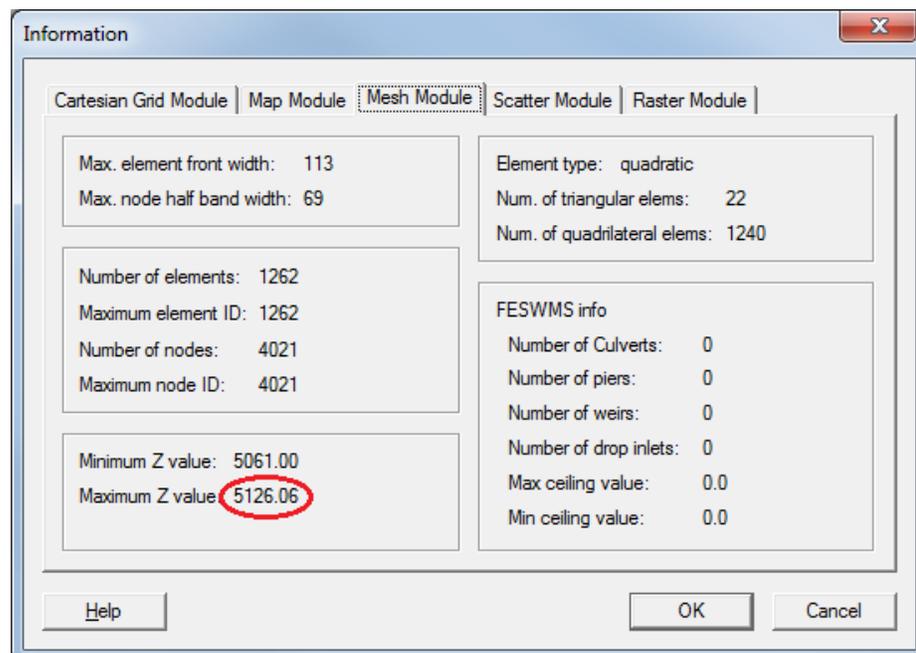

Figure 6   Mesh Information dialog with Maximum Z value highlighted

The maximum bathymetry elevation is well above the boundary condition of 5,070 ft. This means not all of the nodes are wet with the 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.

3. 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 hot start 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 using the following steps:

1. Select *Data* | **Steering Module** to bring up the *Steering Wizard* dialog.

2. Select *FESWMS Spindown* and click **Next**.

3. Turn on *Delete intermediate steering files*.

4. In the *Spindown* section, turn on *Water Surface Elevation* (Figure 7).

5. Turn off any other options in the *Spindown* section.

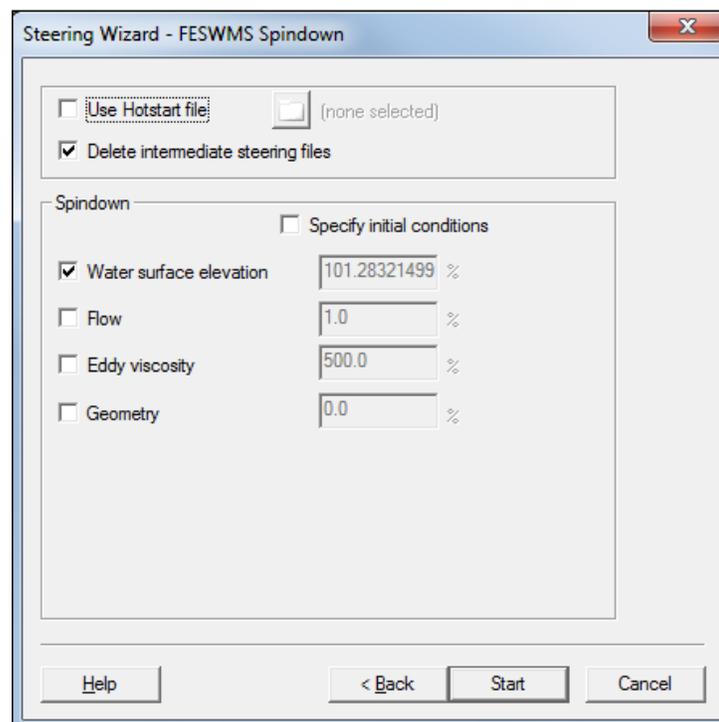6. Click **Start** to close the *Steering Wizard* dialog.

Figure 7   Steering Module dialog

The *FESWMS Spindown* dialog (Figure 8) updates with the progress of spinning down the model. The top window explains the WSE convergence of the current run. Each iteration shows as a green point, allowing determination of if the run is converging or diverging (moving toward or away from 0 head change). The iteration being performed is shown just above this plot.

The bottom window shows the overall spindown of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, the model is finished. This percent is shown just above this plot.

This process can take several minutes to complete. When the spindown has finished, a window appears advising that the "Steering process has terminated – See status file for details".

7. Click **OK** to close the completion message.

8. Click **OK** to close the *FESWMS Spindown* dialog.

This status file is named "SteeringStatus.txt" and gives a summary of the steering process. A final solution file has also been created.
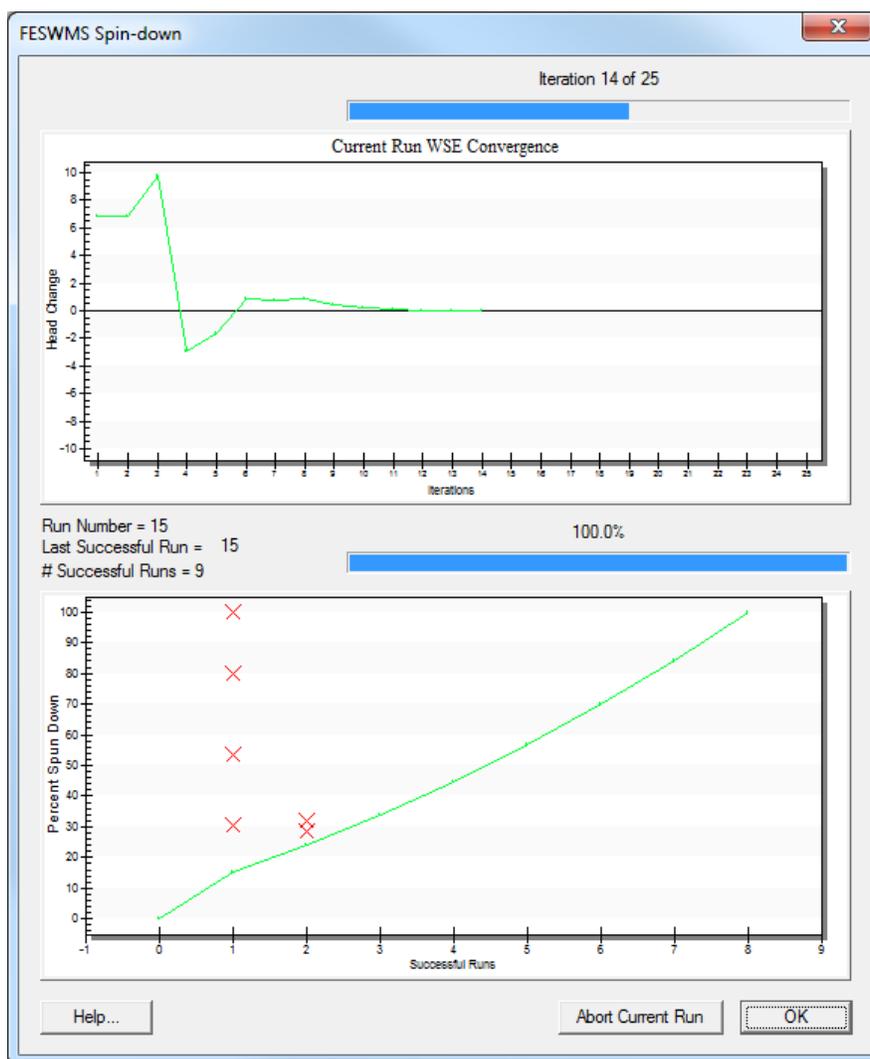

Figure 8   FESWMS Spindown dialog

9   Opening the Solution

To view the solution file:

1. Select *File* | **Open** and go to the *data files* folder.

2. Double-click on "CR_Sim.flo". This adds a new mesh folder named "CR_Sim.flo (FESWMS)" under the Capitol Reef mesh in the Project Explorer.

3. Select *Display* | **Display Options** to open the *Display Options* dialog.

4. Select "2D Mesh" from the list on the left.

5. Turn on *Contours* and *Vectors.*

6. On the *Contours* tab in the *Contour method* section, select "Color Fill" from the drop-down list.

7. On the *Vectors* tab in the *Arrow Options* section, set *Shaft length* to "Define min. and max. length".

8. Set the *Minimum* to "10" and *Maximum* to "30".

9. Click **OK** to exit the *Display Options* dialog.

10   Conclusion

This concludes the *FESWMS Steering (Incremental Loading)* tutorial. Review and post-process the results as desired.