SMS 12.3 Tutorial
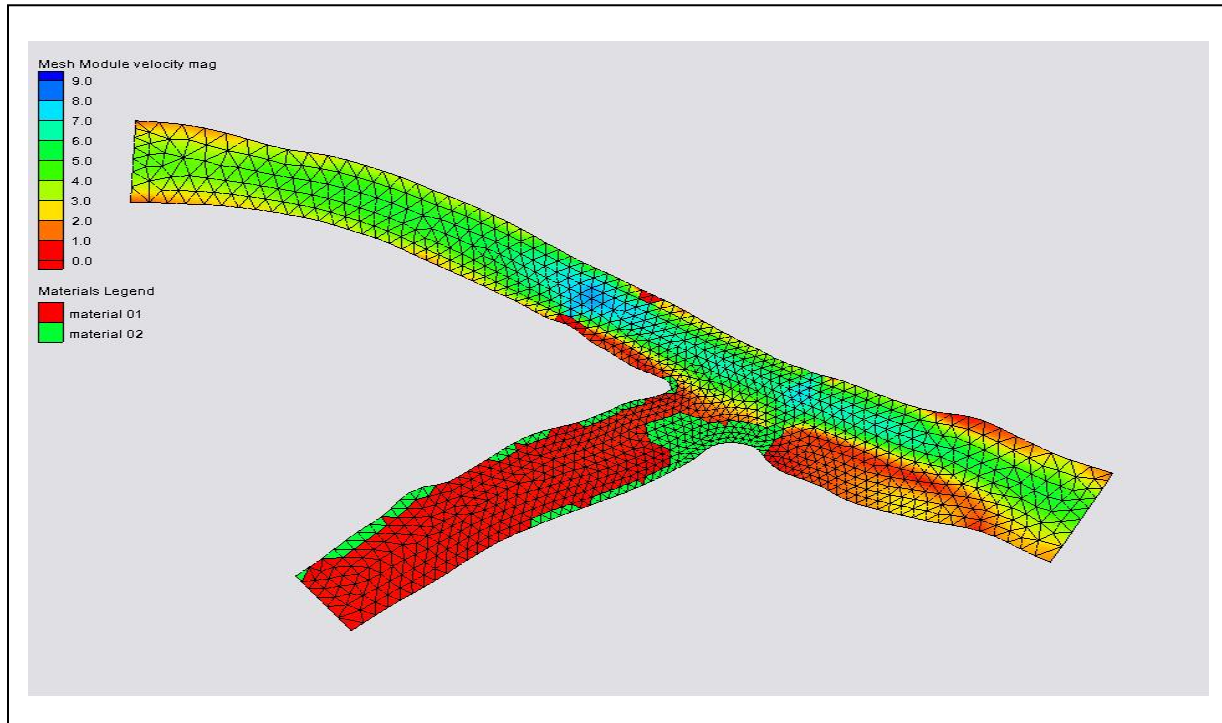
RMA2 Steering Module

Objectives

Learn how to use revision cards for a spindown simulation in RMA2.

Prerequisites

• Overview Tutorial
• RMA2 Tutorial

Requirements

• RMA2
• GFGEN
• Mesh Module

Time

• 30–60 minutes

1   Introduction

The geometry has already been created, renumbered, and saved as a geometry file. Import this file by doing the following:

1. Select *File |* **Open…** to bring up the *Open* dialog.

2. Browse to the *data files* folder for this tutorial and select "tribbase.geo".

3. Click **Open** to import the geometry file and exit the *Open* dialog.

4. If there is geometry open from a previous tutorial, click **Yes** when asked if existing data should be deleted.

5. If asked to specify the type of feet used in the file, click **U.S. Survey**.

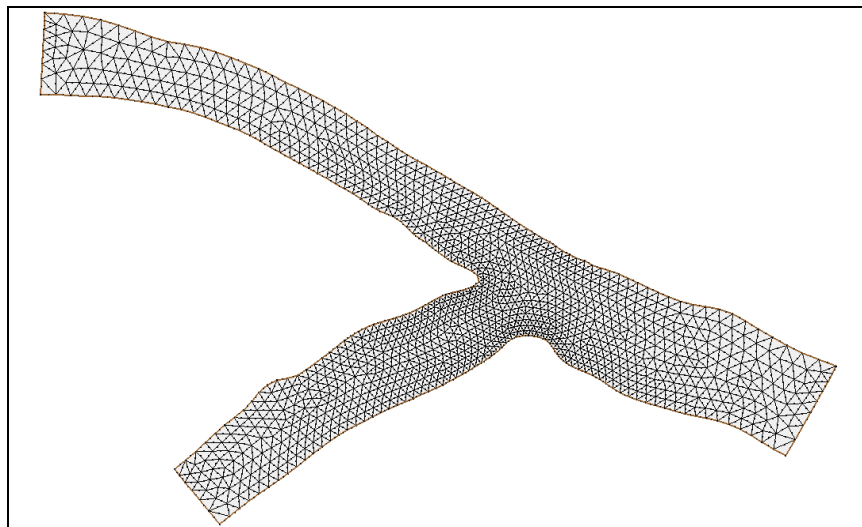The geometry should appear similar to Figure 1.

Figure 1   Mesh contained in "tribbase.geo"

2   Specifying Model Units

Before continuing, make sure the units are set correctly by doing the following:

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

2. In the *Horizontal* section, select *No Projection* and turn on *Units*.

3. In both the *Horizontal* and *Vertical* sections, select "Feet (U.S. Survey)" from the *Units* drop-down.

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

5. Right-click "🔷 tribbase" in the Project Explorer and select **Projection…** to bring up the *Object Projection* dialog.

6. Repeat steps 2–4.

7. Select *Display* | **Projection…** to bring up the *Display Projection* dialog.

8. In the *Horizontal* section, select *No projection*.

9. In both the *Horizontal* and *Vertical* sections, select "Feet (U.S. Survey)" from the *Units* drop-down.

10. 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 how to handle wetting and drying, the model units, simulation time, and the number of iterations to be performed by RMA2.

To define the model parameters:

1. Select *RMA2* / **Model Control…** to bring up the *RMA2 Model Control* dialog.

This is where model parameters are specified. When the dialog opens, it is on the *General* tab by default. This tab contains various items such as simulation titles that describe what is being modeled.

2. Enter "RMA2 Steering Tutorial" into the *Title 1* field.

The remaining options on the *General* tab can be left at their default values. The machine type is used by RMA2 to set numeric precision. The temperature and density parameters are fairly self-explanatory. The initial water surface defaults to the elevation of the highest node in the mesh. The *Scale Factors* section tells RMA2 how to scale the geometry, but is not used much since the geometry can be scaled inside of SMS.

Additional information on these parameters is found in the *SMS Help* and the RMA2 documentation.

3.1   Timing

The *Timing* tab contains options for defining model run time, iterations and convergence. This run involves a steady state (non-transient) simulation, so only the applicable parameters are discussed here.

To set these values:

1. On the *Timing* tab, in the *Simulation Type* section, select *Steady state*.

2. In the *Iterations For Flow Calculations* section, enter "20" as the *Initial solution*.

The model will run this number of iterations unless it converges first.

3. In the *Depth Convergence Parameters* section, enter "0.0002" as the *Steady state depth convergence*.

The model has converged when the maximum change in water depth between iterations at each node is less than this value.

3.2   Files

The *Files* tab is used to specify various file options. To set the options used for this tutorial, do the following:

1. On the *Files* tab, in the *RMA2 Input Files* section, turn on *Specify geometry file* to bring up the *Save* dialog.

2. Enter "tribmesh.geo" as the *File name* and click **Save** to close the *Save* dialog.

This allows a single geometry file to be used with multiple RMA2 simulations to avoid re-writing copies of the geometry in each simulation.

3. In the *RMA2 Solution Files* section, turn on *Write hotstart file*.

This instructs RMA2 to save a hot start output file from this simulation which can be used in other simulations.

3.3   Global Methods

The *Global Methods* tab allows for setting options relating to material properties such as default roughness, eddy viscosity method and wetting/drying.

To set these parameters:

1. On the *Global Methods* tab, in the *Global Roughness Assignment* section, set the *Default roughness value* to "0.03".

2. Be sure the *Global Eddy Viscosity Assignment* section, select *Traditional eddy viscosity approach*.

With this method, exact eddy viscosity values need to be assigned for each material.

3. Click **OK** to accept the remaining default options and close the *RMA2 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 RMA2, 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 two inflow boundaries and one outflow boundary (Figure 2), so three nodestrings must be created. The upper tributary is the main river and has much more water than the lower tributary.
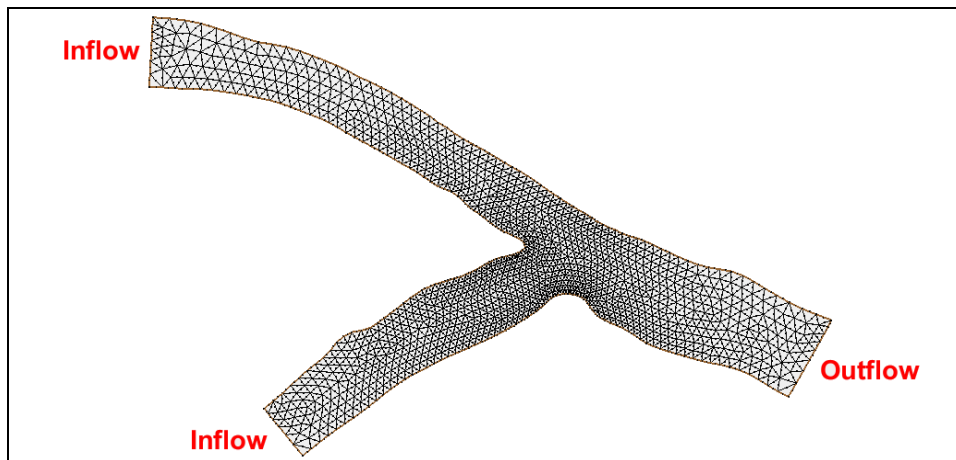

Figure 2   Position of the boundary nodestrings in the mesh

4.1 Creating Nodestrings

Nodestrings should be created from right to left when looking downstream and the first nodestring created span across the whole river section. Because the inflow comes from two tributaries, the outflow nodestring should be created first.

To create the nodestrings, do the following:

1. **Zoom** in on the outflow boundary.

2. Using the **Create Nodestring** tool, create a nodestring at the outflow boundary by clicking on the bottom node of the outflow, holding down the *Shift* key, and double-clicking on the top node of the outflow.

3. **Frame** the project, then **Zoom** in on the upper tributary inflow boundary.

4. Repeat step 2 to create the inflow boundary nodestring across the upper tributary. Remember to create the nodestring from right to left when looking downstream.

5. **Frame** the project, then **Zoom** in on the lower tributary inflow boundary.

6. Repeat step 2 to create the inflow boundary nodestring across the lower tributary, again making sure to create it from right to left when looking downstream.

4.2 Defining Flow Boundary Conditions

To assign the first flow condition:

1. Using the **Select Nodestring** tool, click on the selection box for the upper inflow nodestring.

2. Select *RMA2* / **Assign BC…** to bring up the *RMA2 Assign Boundary Conditions* dialog.

3. In the *Boundary Condition Type* section, select *Specified Flowrate*.

4. In the *Flowrate* section, select *Constant* and enter "55000.0" in the ft^3/s field.

5. In the *Flow Direction* section, turn on *Perpendicular to boundary*.

6. In the *Flow Distribution* section, enter "0.750".

This allows more flow to come into the mesh in the deeper areas.

7. Click **OK** to assign the boundary condition and exit the *RMA2 Assign Boundary Conditions* dialog.

This defines the top left nodestring to be an inflow boundary condition. To define the second inflow boundary condition:

8. Using the **Select Nodestring** tool, select the lower inflow boundary nodestring

9. Repeat steps 2–7 to assign a perpendicular flow, entering "600.0" in the ft^3/s field in step 4.

4.3   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. Using the **Select Nodestrings** tool, select the outflow nodestring.

2. Select *RMA2* / **Assign BC…** to bring up the *RMA2 Assign Boundary Conditions* dialog.

3. In the *Boundary Condition Type* section, select *Water surface elevation.*

4. In the *Water Surface Elevation* section, enter "32.0" as the *Constant.*

5. In the *Advanced Options* section, turn on *Make this nodestring the "Total Flow" nodestring*.

This means that flow crosses this boundary.

6. Click **OK** to assign the boundary condition and close the *RMA2 Assign Boundary Conditions* dialog.

4.4   Defining Material Properties

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

1. Click **Display Options** to bring up the *Display Options* dialog.

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

3. On the *2D Mesh* tab, turn on *Materials* and turn off *Nodes* and *Elements*.

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

The display should appear similar to Figure 3 (material colors may vary). Most of the model uses material one but portions of the smaller tributary are composed of a second material type.
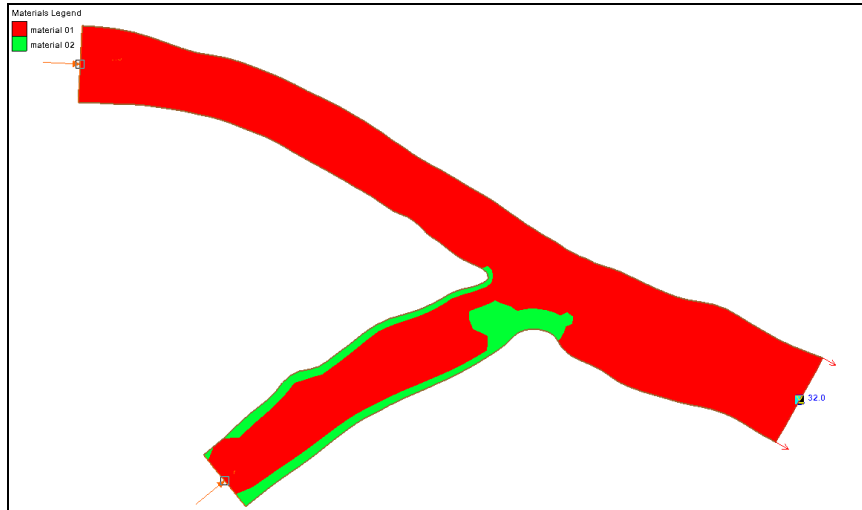
Figure 3   Display of materials

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

5. Click **Display Options** 🖥 to bring up the *Display Options* dialog.

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

7. On the *2D Mesh* tab, 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 particular simulation. The material properties define how water flows through the element.

To edit the material parameters:

1. Select *RMA2* | **Material Properties…** to open the *RMA2 Materials Properties* dialog.

2. Select "material 01" in the list on the left.

This is the main channel material.

3. On the *Turbulence* tab, select *Standard eddy viscosity method* and turn on *Isotropic Values*.

4. Enter "25.0" as the *Exx*.

This is the isotropic eddy viscosity.

5. On the *Roughness* tab, turn off *Override global specification*.

This makes the model use the global value of "0.03" that was previously set in the *Global Methods* tab of the *RMA2 Model Control* dialog.

6. Select "material 02" (the right bank material) from the list on the left.

7. On the *Turbulence* tab, enter "50.0" as the *Exx*.

8. On the *Roughness* tab, turn on *Override global specification*, select *Specify roughness value*, and enter "0.04" in the field to the right.

9. Click **OK** to close the *RMA2 Material Properties* dialog.

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

5   Saving the Simulation

SMS can launch the RMA2 numerical model. However, the model will read the data from files. Therefore, the simulation must be saved prior to running. If the project hasn't been saved prior to running the model, a prompt will appear asking to save the project.

To save the simulation with 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 "trib.sms" as the *File name*.

4. Click **Save** to save the project and close the *Save As* dialog.

6   Running the Model

Now to try running RMA2. To do this:

1. Select *RMA2* | **Run RMA2** to bring up the *Model Checker* dialog.

Before SMS launches the model, a quick model check is done on the data to make sure everything is valid. The model check brings up the dialog if any anomalies are detected. For this model, two warnings should be detected (Figure 4).

The second warning says that the default starting elevation for the simulation will leave portions of the domain dry. This can lead to instabilities. One option is to raise the initial water level, but if it is much higher than the outflow boundary elevation, instabilities can develop at those locations. If the simulation is not stable with the default elevation, it may be best to use incremental loading as will be demonstrated later. For now, ignore this warning.
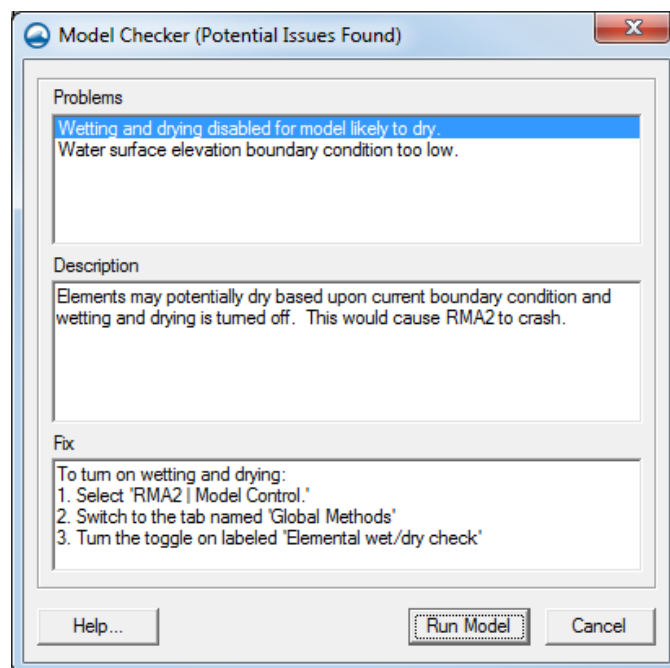

Figure 4   Warning in the Model Checker dialog

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

2.   Click **Cancel** to close the *Model Checker* dialog without running RMA2.

3.   Select *RMA2* / **Model Control…** to open the *RMA2 Model Control* dialog.

4.   On the *Global Methods* tab, in the *Global Wet/Dry Technique* section, turn on *Elemental wet/dry check*.

5.   Click **OK** to close the *RMA2 Model Control* dialog.

6.   **Save** ⊟ the project.

7.   Select *RMA2* / **Run RMA2** to bring up the *Model Checker* dialog.

The warning regarding water level should appear in the dialog.

8.   Ignore the warning and click **Run Model** to bring up the *GFGen* model wrapper dialog.

Since this is a new RMA2 simulation, SMS first launches the GFGen program in a model wrapper dialog. This dialog displays the textual output from the model and the status of the run. (The location of the GFGen executable is specified in the SMS preferences. If no location is specified, SMS will ask for the location of the executable.)

9.   When *GFGen* finishes, click **Run RMA2** to close the *GFGen* model wrapper dialog and open the *RMA2* model wrapper dialog.

RMA2 will run through four iterations and diverge. At the bottom of the *RMA2 output* window is the text "STOP depth convergence exceeds 50.0", as highlighted in Figure 5. This is how RMA2 declares that it has not successfully converged.

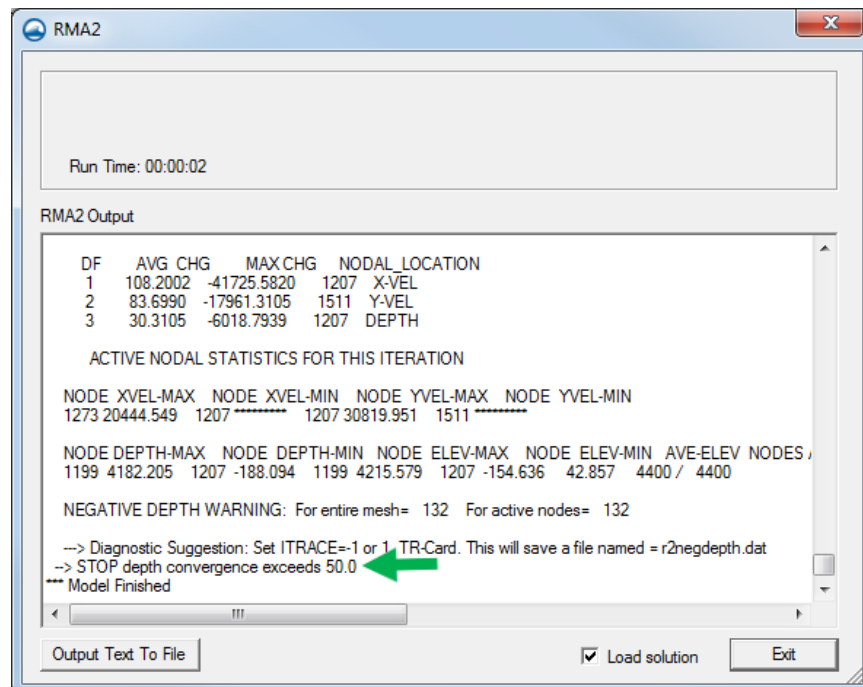10. Turn off *Load Solution* and click **Exit** to close the *RMA2* model wrapper dialog.

Figure 5   Output showing divergence after four iterations

7   Using RMA2 Revisions

Various things can contribute to a model not converging. In this case, SMS had given an error message that the initial water surface was low. The low water surface elevation for this simulation does not allow RMA2 to converge from the cold start simulation. Revisions can be defined to allow the simulation to converge. A set of revisions is like a set of hot start simulations except that they are all run at once with no user interaction.

7.1   Checking for High Bathymetry

The first thing to do is find the highest bathymetric value from all nodes. To do this:

1. Select *File |* **Get Info…** to open the *Information* dialog (Figure 6).

2. On the *Mesh Module* tab, take note of the *Maximum Z value*.
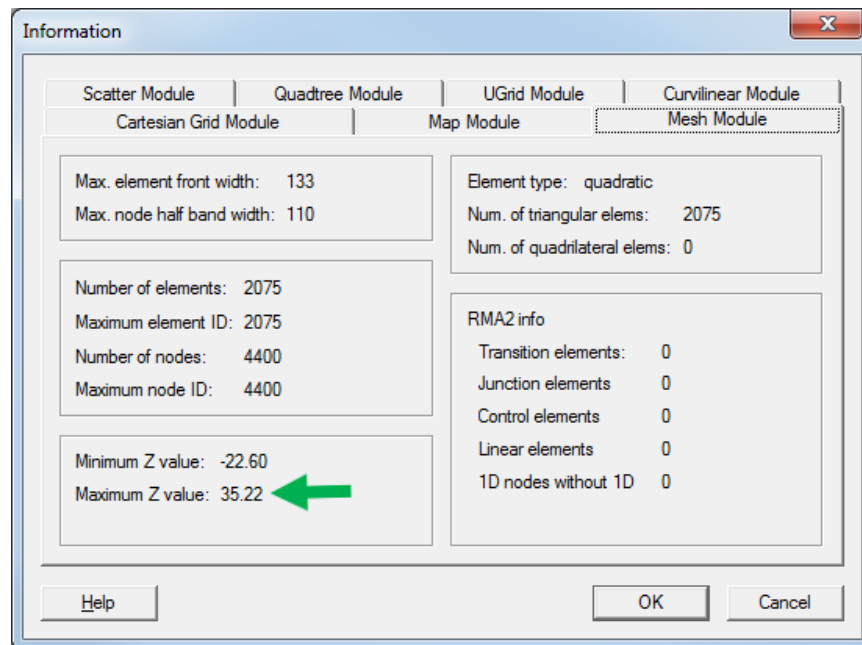
3. Click **OK** to close the *Information* dialog.

Figure 6   Information dialog for the mesh

7.2   Changing Initial Water Surface Elevation

The initial water surface elevation value should be increased to be above the highest bathymetry value. To change this value:

1. Using the **Select Nodestring** tool, select the outflow nodestring.

2. Select *RMA2* / **Assign BC…** to open the *RMA2 Assign Boundary Conditions* dialog.

3. In the *Water Surface Elevation* section, enter "35.5" as the *Constant*.

4. Click **OK** to set the new elevation and close the *RMA2 Assign Boundary Conditions* dialog.

7.3   Creating Revisions

The water surface elevation can be revised to the final value of 32 feet through a series of revisions. To create the revisions:

1. Using the **Select Nodestring** ⭧ tool, select the outflow nodestring if it is not still selected.

2. Select *RMA2* / **Revisions…** to open the *RMA2 Revisions* dialog.

3. In the *Existing Revisions* section, right-click the "0.0 hours" entry and select **New Revision**.

A new item labeled "Revision 1" should appear under the "0.0 hours" label.

4. Select "Nodestring BC" from the drop-down at the bottom of the *Selected Revision* section.

5. Click **Add** to bring up the *RMA2 Assign Boundary Conditions* dialog.

6. In the *Water Surface Elevation* section, enter "34.0" as the *Constant*.

7. Click **OK** to close the *RMA2 Assign Boundary Conditions* dialog.

8. Repeat steps 3–7 to add a second revision, entering "33.0" as the *Constant* in step 6.

9. Repeat steps 3–7 to add a third revision, entering "32.0" as the *Constant* in step 6.

10. Click **Close** to assign these new revisions and exit the *RMA2 Revisions* dialog.

RMA2 will first run an initial simulation with the downstream water surface elevation at 35.5 feet, and then will run five additional simulations, revising the boundary condition down by one foot each time. Each successive step will use the previous solution as a starting point for the next simulation. Only one solution file will be generated, and it will have the final water surface elevation value of 32 feet.

7.4   Running a New Simulation

Now to try running this simulation with the set of revisions. To do this:

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 "trib_rev.sms" as the *File name*.

4. Click **Save** to save the project under the new name and close the *Save As* dialog.

5. Select *RMA2 | **Run RMA2*** to bring up the *GFGen* model wrapper dialog.

6. Once GFGen finishes, click **Run RMA2** to close the *GFGen* model wrapper dialog and bring up the *RMA2* model wrapper dialog.

RMA2 should successfully run the initial simulation and all three revisions.

7. When RMA2 finishes, turn on *Load solution* and click **Exit** to close the *RMA2* model wrapper dialog.

8. Click **Display Options** 🖥 to open the *Display Option* dialog.

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

10. On the *2D Mesh* tab, turn on *Materials*, *Contours*, and *Vectors*.

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

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

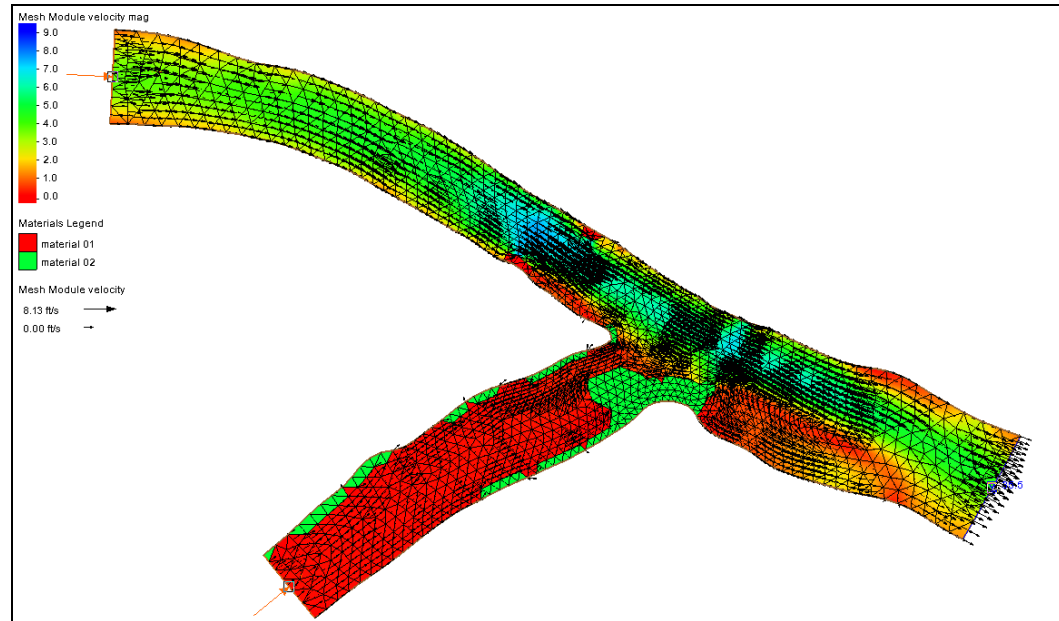The project should appear similar to Figure 7.

Figure 7   Mesh with materials, contours, and vectors visible

The process of incremental loading allows models such as RMA2 to compute solutions for a wider array of conditions by transitioning to those conditions from more stable scenarios. In this case, the stable scenario was higher water levels to begin with.

The difficulty with using revisions is that all the steps in the revision patterns must be defined before running the model. If one of the steps produces too large of a change, the model could become unstable, resulting in a failed run. In this situation, the modeler would have to edit the desired revisions and start over. An alternative to revisions is the steering module in SMS.

8   Conclusion

This concludes the "RMA2 Steering Module" tutorial. Feel free to continue to experiment with the SMS interface, or exit the program.