SMS 11.2 Tutorial
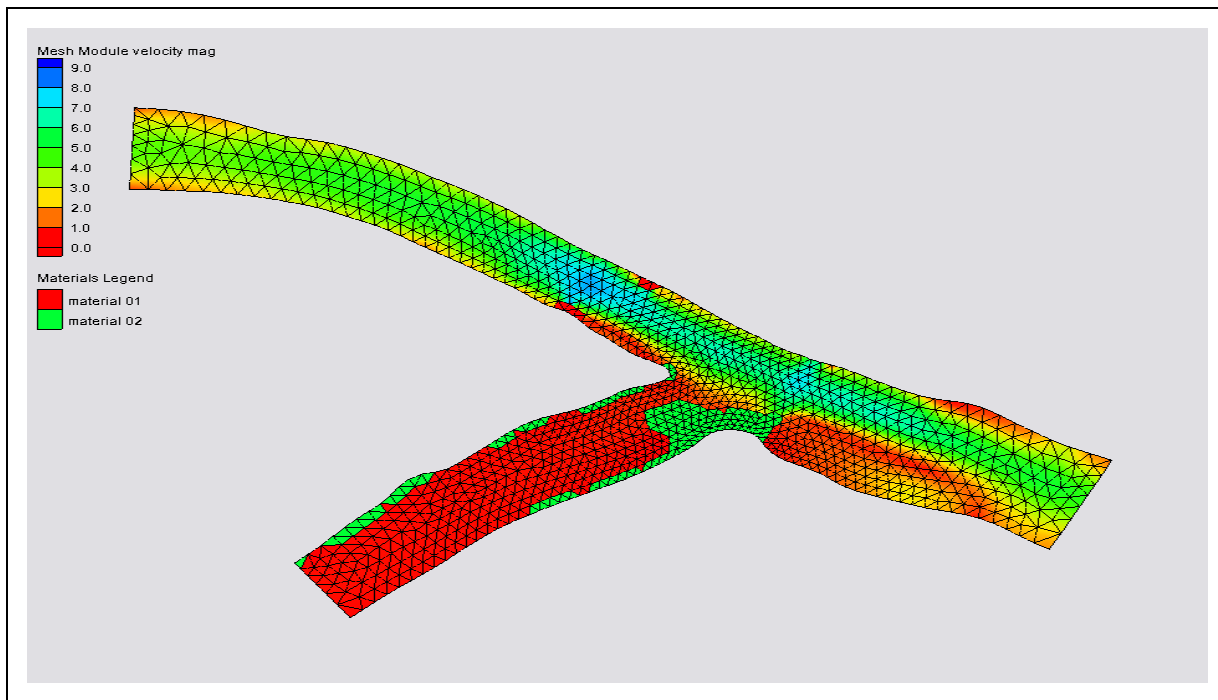# RMA2 Steering Module

Objectives

This lesson will teach you how to use revision cards for a spin down 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 and renumbered. To open the file:

1. Select *File | Open*.

2. Open the file *tribbase.geo* from 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. If SMS asks for units, click the *U.S. Survey* button. The geometry should appear similar to Figure 1 (display options may vary).
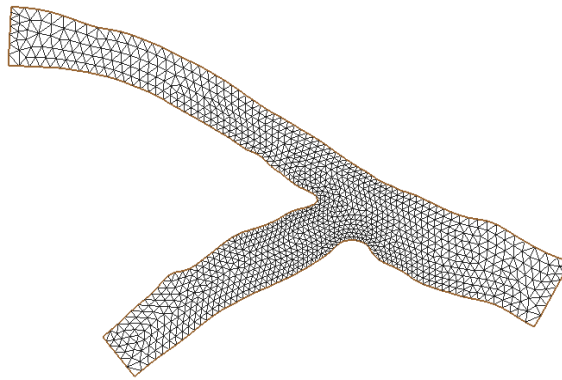

Figure 1.   The mesh contained in the file tribbase.geo.

2   Specifying Model Units

Before continuing, make sure that the units are English. To do this:

1. Right-click on the default coverage and select *Projection*.

2. In the Horizontal section, select *No Projection* and toggle on *Units*. Make sure the units for both horizontal and vertical are set to *U.S. Survey Feet*. Click OK.

3. Now, Right-click on the *tribbase* mesh item and select *Projection*. Repeat step 2.

4. Next, Select *Display | Projection*.

5. Make sure the Horizontal section is set to *No projection* and the Horizontal and Vertical Units are set to *U.S. Survey Feet*. 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 how to handle wetting and drying, the model units, simulation time, and the number of iterations to be performed by *RMA2*. Additional information on these parameters is found in the *SMS Help* and the *RMA2* documentation. To define the model parameters:

1. Select *RMA2 | Model Control*. This opens the *RMA2 Model Control* dialog, in which the model parameters are specified.

3.1 General

The *General* page contains various items such as simulation titles that describe what is being modeled. To set a title:

1. Click the *General* tab.

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

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

3.2 Timing

The *Timing* page 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. Click the *Timing* tab.

2. Be sure the *Simulation Type* is set to *Steady state*.

3. In the *Iterations For Flow Calculations* section, set the number of *Initial solution* iterations to 20. The model will run this number of iterations unless it converges first.

4. In the *Depth Convergence Parameters* section, set the *Steady state depth convergence* value to 0.0002. The model has converged when the maximum change in water depth between iterations at each node is less than this value.

3.3   Files

The *Files* page is used to specify various file options. To set these:

1. Click the *Files* tab.

2. Turn on the *Specify geometry file* option. Enter the file name "tribmesh.geo" and click the *Save* button. 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 the *Write hotstart file* option to have RMA2 save a hotstart output file from this simulation that can be used in other simulations.

3.4   Global Methods

The *Global Methods* page is used to set items relating to material properties such as default roughness, eddy viscosity method and wetting/drying. To set these parameters:

1. Click the *Global Methods* tab.

2. In the *Global Roughness Assignment* section, set the *Default roughness value* to 0.03.

3. Be sure the *Global Eddy Viscosity Assignment* type is set to *Traditional eddy viscosity approach*. With this method, exact eddy viscosity values need to be assigned for each material.

To accept all the above values:

Click the *OK* button to 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 and one outflow boundaries so three nodestrings must be created. The upper tributary is the main river and has much more water than the lower tributary. These boundaries are highlighted in Figure 2.
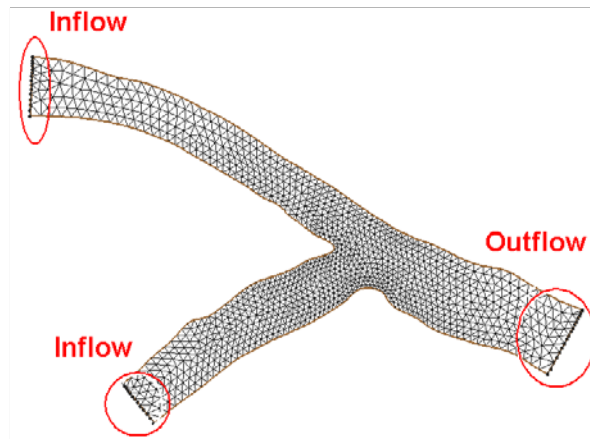
Figure 2.   Position of the boundary nodestrings in the mesh.

4.1   Creating Nodestrings

Nodestrings should be created from right to left when looking downstream and the first nodestring should be that which spans across the whole river section. Therefore the outflow nodestring at the right should be created first. To create this:

1. Choose the *Create Nodestrings* tool 〜 from the *Toolbox*.

2. Start the nodestring by clicking on the lower node at the outflow boundary.

3. Hold the *SHIFT* key and double-click on the upper node at the outflow boundary to create and end the nodestring.

The inflow nodestrings can be created now. To do this:

Create a nodestring across the top inflow boundary and then across the bottom one. Create each right to left when looking downstream.

4.2   Defining Flow Boundary Conditions

To assign the first flow condition:

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

2. Select the top left nodestring by clicking on the icon.

3. Select *RMA2 | Assign BC*.

4. Change the *Boundary Condition Type* to *Specified Flowrate* and assign a constant *Flowrate* of 55,000 (ft^3/s or cfs).

5. Make sure the *Flow Direction* is set to *Perpendicular to Boundary*.

6. Enter a *Flow Distribution* 0f 0.750. This allows more flow to come into the mesh in the deeper areas.

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

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

1. Select the bottom left nodestring.

2. Repeat the above steps to assign a perpendicular flow of 600 (cfs). Click OK when finished.

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. Select the outflow nodestring.

2. Select *RMA2 | Assign BC*.

3. Change the Boundary Condition Type to *Water surface elevation* and assign a constant value of 32 (ft).

4. In the *Advanced Options* section, make sure the *Make this nodestring the "Total Flow" nodestring* option is checked (all flow crosses this boundary).

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

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. 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 uses material one but portions of the smaller tributary are composed of a second material type.
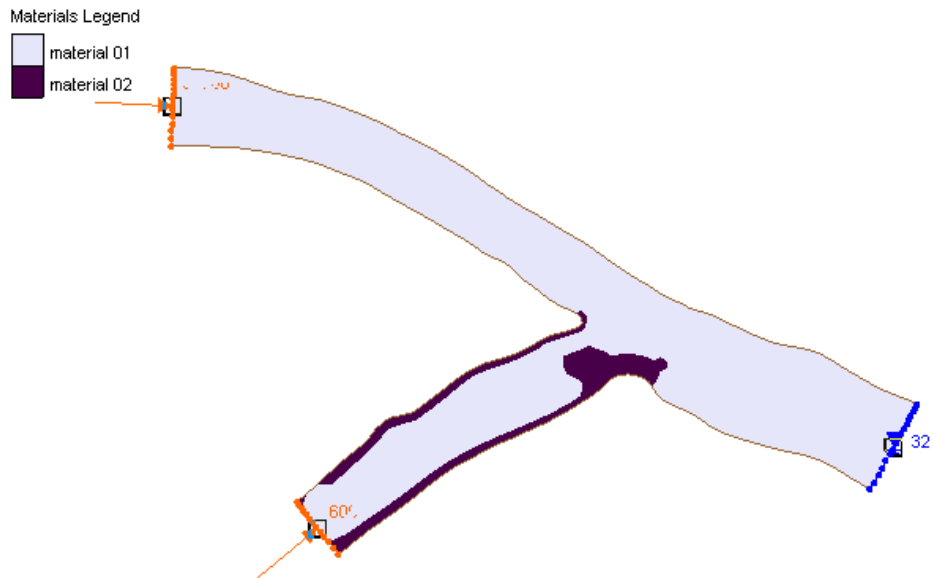
Figure 3.   The display of materials.

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 *RMA2 | Material Properties*.

2. In the *RMA2 Materials Properties* dialog, highlight *material 01*.

3. Under the *Turbulence* tab, make sure the *Standard eddy viscosity method* option is selected and the *Isotropic Values* box is checked. Enter a value of 25 for the eddy viscosity (E_{xx}) for this material.

4. Under the *Roughness* tab, make sure the *Override global specification* option is turned off for this material. This will use the global value of 0.03 that was already set in the *Materials* page of the *RMA2 Model Control* dialog.

5. Highlight *material 02* and set isotropic eddy viscosity to 50. Override the global roughness value for this material and set the roughness value to 0.04.

6. Click the *OK* button 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 you haven't done this, you will be prompted to do so. To save the simulation with the project:

1. Select *File | Save As*.

2. Make sure the *Save as type* is set to *Project Files(*.sms)* and enter "trib" as the *File name*.

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

6   Running the Model

At this point, you are ready to try running RMA2. To do this, choose *RMA2 | Run RMA2*. 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 if any anomalies are detected. For this model, two warnings should be detected.

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 we will ignore this warning.
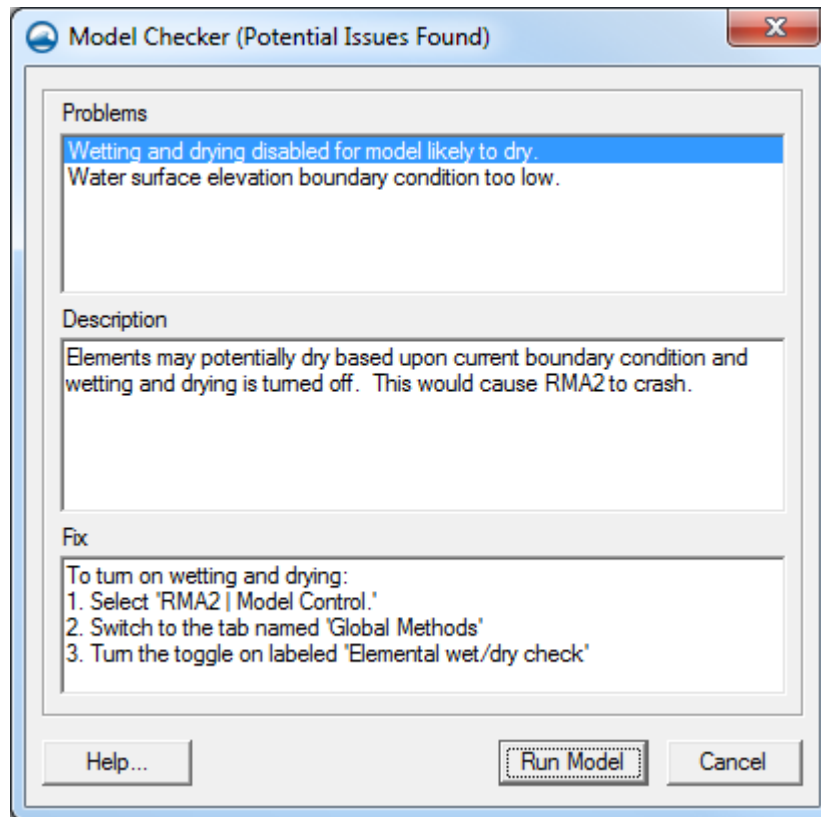
Figure 4.   Warning in RMA2 data.

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 *RMA2*.

2. Select *RMA2 | Model Control*.

3. On the *Global Methods* page turn on the *Elemental wet/dry check* option.

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

5. Select *File | Save Project* to resave the simulation with the project.

6. Issue the *RMA2 | Run RMA2* command again. The Model Checker should again issue the warning regarding water level. Ignore this and click the *Run Model* button to run the model.

7. Since this is a new *RMA2* simulation, *SMS* will first run the *GFGEN* program. *SMS* will launch the model 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.)

8. When *GFGEN* finishes, click the *Run RMA2* button. The *RMA2* executable then launches.

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.

Uncheck the *Load Solution* box and click the *Exit* button to close the *RMA2* output window.
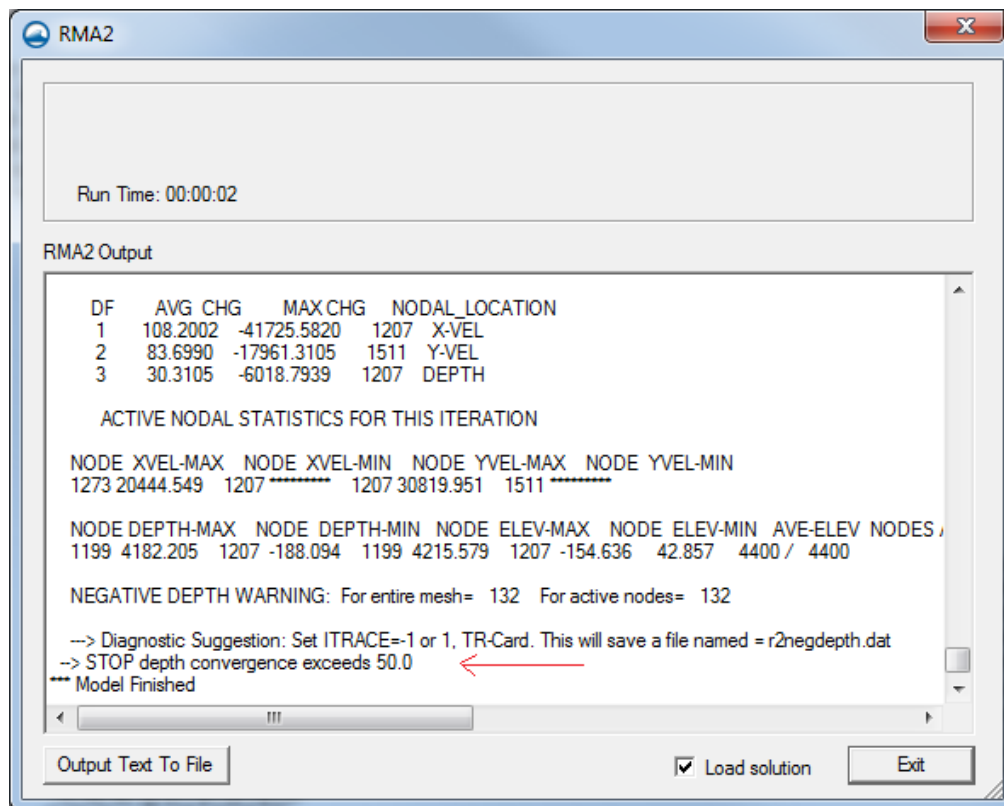

Figure 5.   Output from running trib.sim.

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 coldstart simulation. Revisions can be defined to allow the simulation to converge. A set of revisions is like a set of hotstart 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*.

2. In the *Mesh Module* tab, look at the *Maximum Z value* highlighted in Figure 6, and click *OK*.
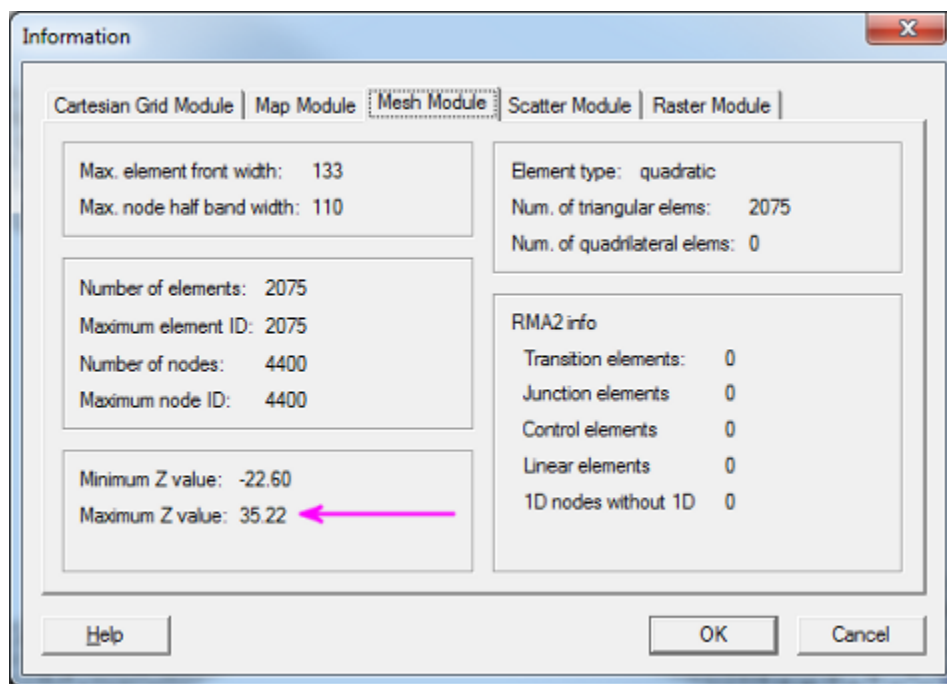

Figure 6.   Mesh Information dialog.

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. With the select nodestring tool, select the outflow nodestring.

2. Select *RMA2 | Assign BC*.

3. Change the *Water Surface Elevation* value to 35.5 (feet) and click *OK*.

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. Make sure the outflow nodestring is still selected.

2. Select *RMA2 | Revisions*.

3. In the *Existing Revisions* section, right-click the "0.0 hours" text and choose the *New Revision* item.

4. In the Selected Revision section, change the type to *Nodestring BC*.

5. Click the *Add* button and set the *Water Surface Elevation* value to 34 (feet). Then click the *OK* button in the *RMA2 Assign Boundary Conditions* dialog.

6. Repeat steps 3, 4 and 5 to add two more revisions, revising the boundary condition to 33, and finally 32 feet.

7. Click the *Close* button to assign these revisions.

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

You can now try running this simulation with the set of revisions. To do this:

1. Select *File | Save As*.

2. Set the *Save as type* to *Project Files (*.sms)* and save "trib_rev.sms".

3. Select *RMA2 | Run RMA2*. (Note: Execution of *RMA2* should begin immediately without running *GFGEN*.) *RMA2* should successfully run the initial simulation and all three revisions.

4. When *RMA2* finishes, click the *Exit* button. Leave the *Load Solutions* toggle selected to load the results of the model run.

5. To view the solutions, go to *Display | Display Options* and in the *2D Mesh* tab turn on *Materials, Contours,* and *Vectors*. Switch to the *Contours* tab. Change the *Color Method* to *Color fill*.

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

8   Conclusion

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*.

This concludes the *RMA2 Steering Module* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.