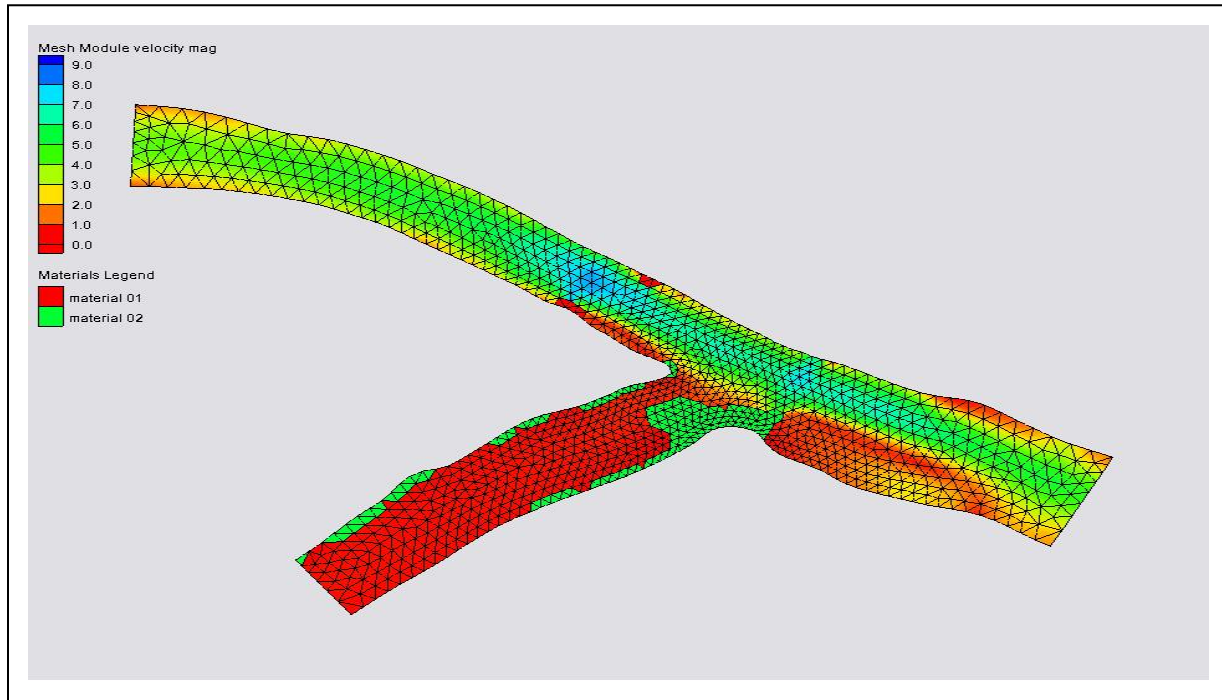


SMS 12.2 Tutorial

RMA2 Steering Module



Objectives

This tutorial instructs on how to use revision cards for a spin down simulation in RMA2.

Prerequisites

- 5 Overview Tutorial
- 6 RMA2 Tutorial

Requirements

- 2 RMA2
- 3 GFGEN
- 4 Mesh Module

Time

- 1 30-60 minutes



1	Introduction	2
2	Specifying Model Units	3
3	Defining Model Parameters.....	3
3.1	General	3
3.2	Timing.....	3
3.3	Files	4
3.4	Global Methods.....	4
3.5	Setting the Model Parameters.....	4
4	Defining Boundary Conditions	4
4.1	Creating Nodestrings.....	5
4.2	Defining Flow Boundary Conditions	5
4.3	Defining Head Boundary Conditions	6
4.4	Defining Material Properties	6
5	Saving the Simulation	8
6	Running the Model.....	8
7	Using RMA2 Revisions	10
7.1	Checking for High Bathymetry	10
7.2	Changing Initial Water Surface Elevation.....	11
7.3	Creating Revisions	11
7.4	Running a New Simulation	12
8	Conclusion.....	12

1 Introduction

The geometry has already been created and renumbered. To open the file:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Locate and select the file “tribbase.geo” from the data files folder for this tutorial then click **Open**. If there is geometry open from a previous tutorial, a prompt will ask if wanting to delete existing data. If this happens, click the **Yes** button.
3. If SMS asks for units, click the **U.S. Survey** button.

The geometry should appear similar to Figure 1 (display options may vary).

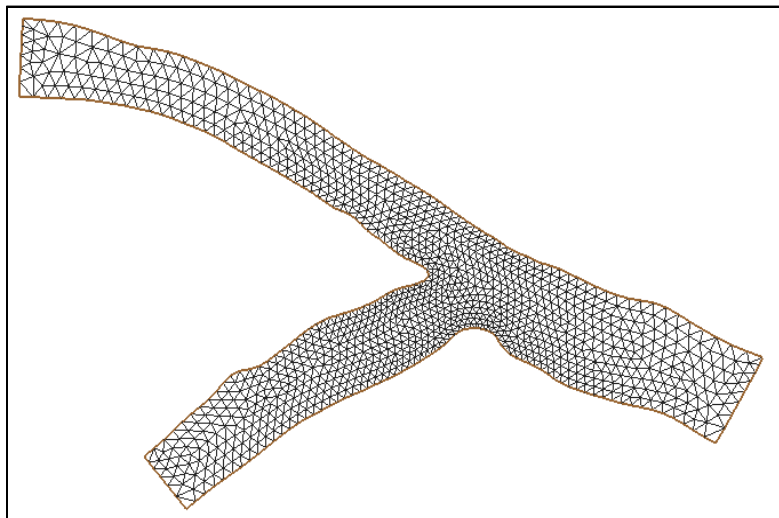


Figure 1 Mesh contained in “tribbase.geo”

2 Specifying Model Units

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

1. Right-click on the default coverage in the Project Explorer – it should be called “Area Property” – and select **Projection** to bring up the *Object Projection* dialog.
2. In the *Horizontal* section, select *No Projection* and check on *Units*. Make sure the *Units* for both *Horizontal* and *Vertical* are set to “U.S. Survey Feet”. Click **OK** to close the *Object Projection* dialog.
3. Right-click on the “tribbase” mesh item from the Project Explorer and select **Projection**. Repeat step 2.
4. Select *Display* | **Projection** to bring up the *Display Projection* dialog.
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 *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. 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 by clicking the box. A *Save* dialog will appear.
3. Enter the *File name* of “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.
4. 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.

3.5 Setting the Model Parameters

To accept all the above values:

1. 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.

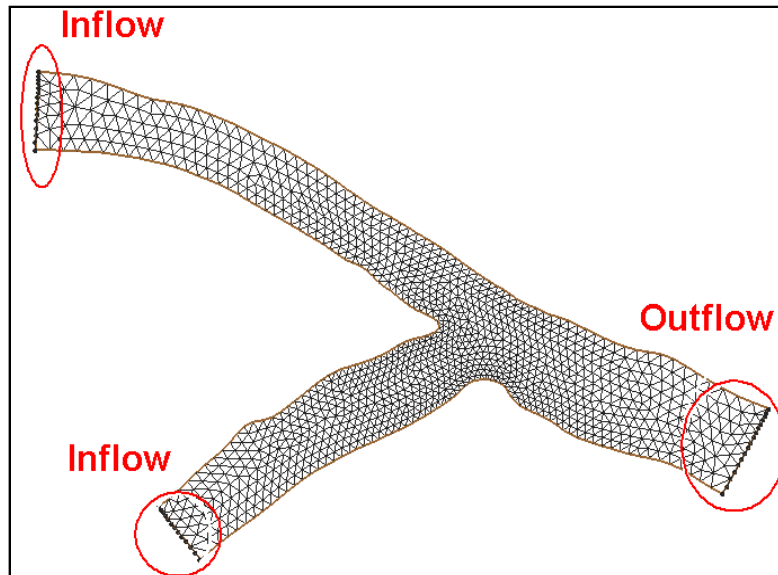



Figure 2 Position of the boundary nodestrings in the mesh

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.
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:

4. 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. An icon or box will appear at the center of each nodestring.
2. Select the top left nodestring by clicking on the icon.


3. Select *RMA2 / Assign BC* to bring up the *RMA2 Assign Boundary Conditions* dialog.
4. Change the *Boundary Condition Type* to *Specified Flowrate* and select *Constant* under *Flowrate* then enter “55000” (ft³/s or cfs).
5. Make sure the *Flow Direction* has *Perpendicular to Boundary* checked on.
6. Enter a *Flow Distribution* of “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 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:

1. Select the bottom left nodestring.
2. Repeat steps 3–7 above to assign a perpendicular flow. Use all values above except enter “600” (cfs) as the *Constant Flowrate*.


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, still using the **Select Nodestrings**  tool.
2. Select *RMA2 / Assign BC* to bring up the *RMA2 Assign Boundary Conditions* dialog.
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. The *Display Options* dialog will open.
2. Highlight *2D Mesh* from the list on the left and 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 though actual material colors may vary. Most of the model uses material one but portions of the smaller tributary are composed of a second material type.

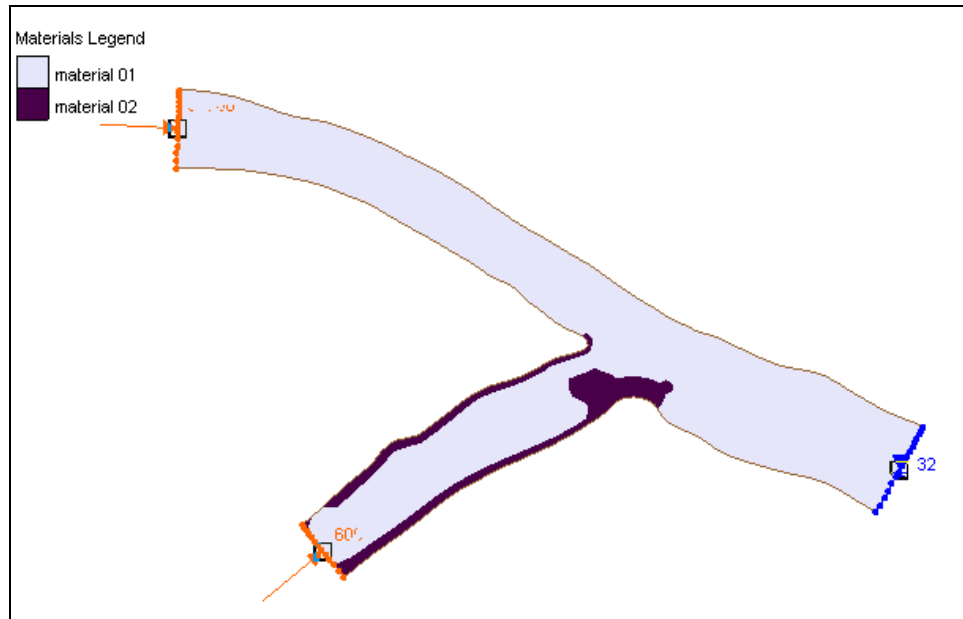



Figure 3 Display of materials

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

1. Select *Display* | **Display Options**, or click the **Display Options**  macro, to bring up the *Display Options* dialog again.
2. Highlight *2D Mesh* from the list on the left and turn off the *Materials* option.
3. Turn on the *Elements* options.
4. Click the **OK** button 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. Highlight “material 01” in the list on the left. This is the main channel material.
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”, which is the right bank material, and in the *Turbulence* tab set isotropic eddy viscosity (E_{xx}) to “50”.
6. Click on the *Roughness* tab. Override the global roughness value by checking the *Override global specification* box and set the *Specify roughness value* to “0.04”.
7. 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 the project hasn't been save, a prompt will appear to do so. To save the simulation with the project:

1. Select *File* | **Save As** to open the *Save As* dialog.
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, try running RMA2. To do this:

1. Select *RMA2* / **Run RMA2**. The *Model Checker* will appear.

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, ignore this warning.

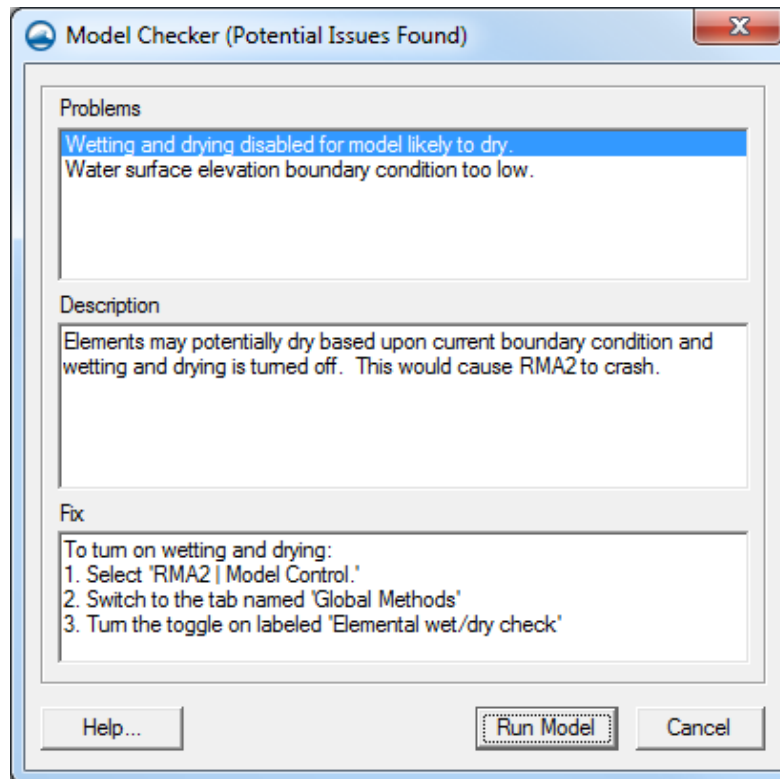


Figure 4 Warning in RMA2 data

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* to open the *RMA2 Model Control* dialog.
3. On the *Global Methods* tab 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. Select the *RMA2 / Run RMA2* command again.
7. The Model Checker should again issue the warning regarding water level. Ignore this and click the **Run Model** button to run the model.

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.

9. Uncheck the *Load Solution* box and click the **Exit** button to close the RMA2 model wrapper window.

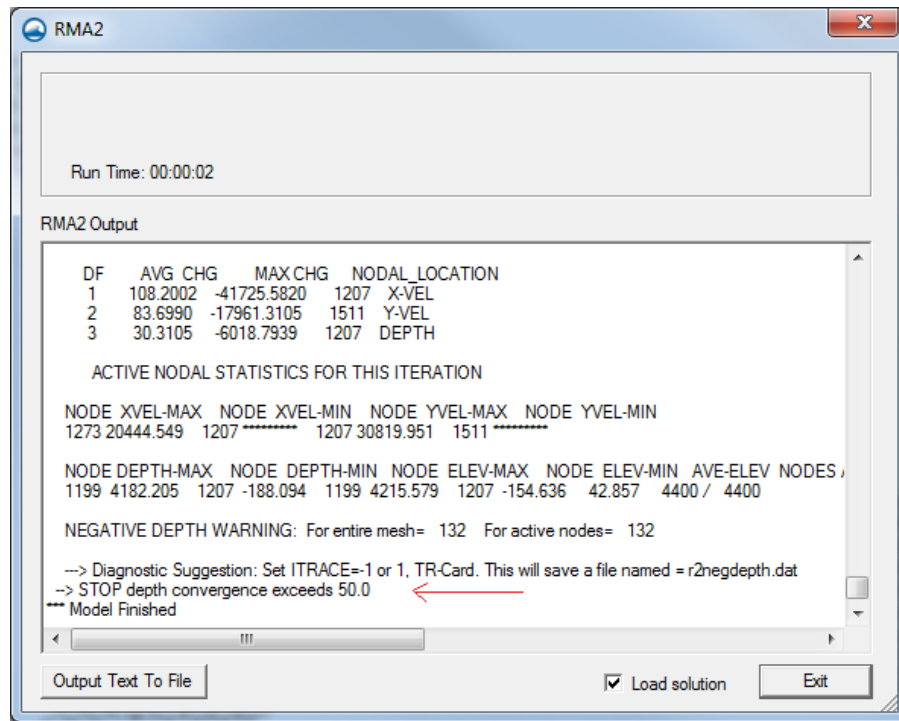


Figure 5 Output from running trib.sim

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** to open an *Information* dialog.
2. In the *Mesh Module* tab, look at the *Maximum Z value* highlighted in Figure 6 and click **OK** to close the *Information* dialog.

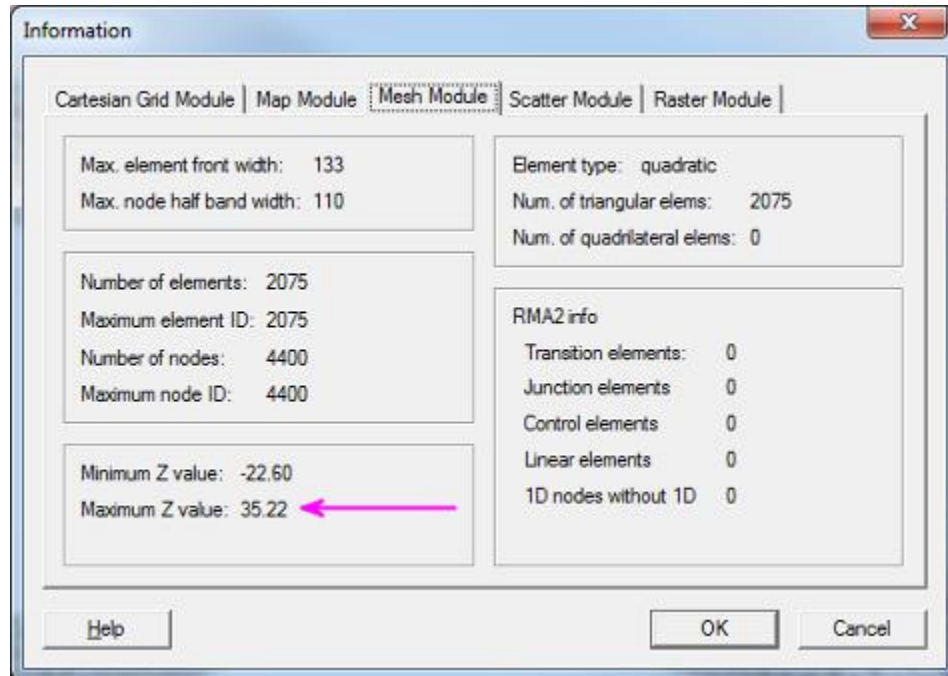



Figure 6 Mesh Information dialog

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** to open the *RMA2 Assign Boundary Conditions* dialog.
3. Change *Constant* under the *Water Surface Elevation* to “35.5” (feet) and click **OK** to set a 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. Make sure the outflow nodestring is still selected.
2. Select **RMA2 / Revisions** to open the *RMA2 Revisions* dialog.
3. In the *Existing Revisions* section, right-click the “0.0 hours” text and choose the **New Revision** item. A new item labeled “Revision 1” should appear under the “0.0 hours” label.
4. In the *Selected Revision* section, change the type to “Nodestring BC”.
5. Click the **Add** button to bring up the *RMA2 Assign Boundary Conditions* dialog.
6. Set *Constant* value under the *Water Surface Elevation* to “34” (feet). Then click the **OK** to close the *RMA2 Assign Boundary Conditions* dialog.

7. Repeat steps 3, 4 and 5 to add two more revisions, revising the boundary condition to “33”, and finally “32” feet.
8. 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

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

1. Select *File* | **Save As** to access the *Save As* dialog.
2. Set the *Save as type* to “Project Files (*.sms)” and enter a *File Name* of “trib_rev.sms”. Click **Save**.
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 Solution* box checked on to load the results of the model run.
5. To view the solutions, go to *Display* | **Display Options** to open the Display Option dialog.
6. Highlight *2D Mesh* in the list on the left and turn on *Materials*, *Contours*, and *Vectors*.
7. Switch to the *Contours* tab. Change the *Contour Method* to “Color fill”.
8. 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. Continue to experiment with the SMS interface or quit the program.