SMS 13.0 Tutorial
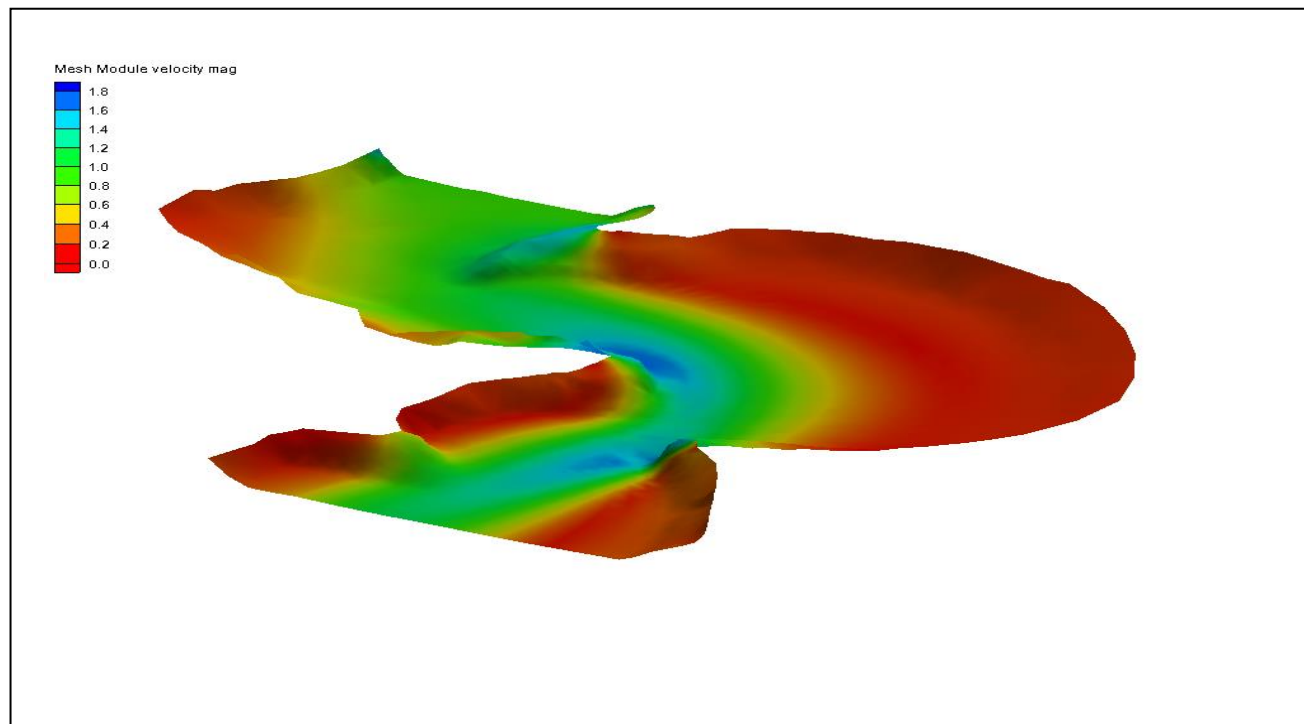
Basic RMA2 Analysis

Objectives

Learn how to prepare a mesh for an RMA2 simulation.

Prerequisites	Requirements	Time
• Overview Tutorial	• RMA2 • GFGEN • Mesh Module	• 15–25 minutes

AQUAVEO™

1   Introduction

The project file for this tutorial contains a simulation ("*.sim" file) for RMA2. The simulation includes links to all the files needed by RMA2 (or TABS-MD) to run an analysis. The actual input data is stored in the files named in the simulation file.

To import the project file:

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

2. Select "Project Files (*.sms)" from the *Files of type* drop-down.

3. Browse to the *data files* folder for this tutorial and select "stmary.sms".

4. Click **Open** to import the project and exit the *Open* dialog.

5. If geometry is still open from a previous tutorial, a prompt will ask if existing data should be deleted. If this happens, click **Yes**.

The imported mesh includes geometry (nodes and elements), material properties, and boundary conditions (Figure 1).
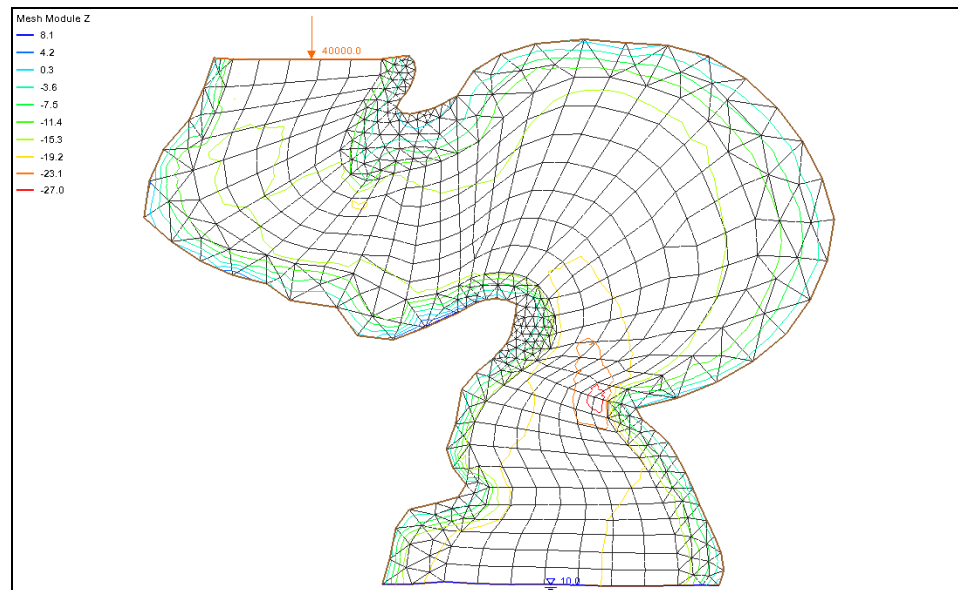

Figure 1   Mesh from stmary.sms.

2   Defining Material Properties

Each element of the mesh is assigned a material type. Each material type includes a value for Manning's roughness coefficient, parameters for turbulence, and parameters for wetting and drying. These material properties must be changed for this analysis. The material properties define how water flows through the element (see the *SMS Help* for details of what each parameter represents).

To edit the material parameters:

1. Select *RMA2 |* **Material Properties…** to bring up the *RMA2 Material Properties* dialog.

2. Select "main_channel" from the list on the left.

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

4. Enter "50.0" as the *Exx* (eddy viscosity).

5. On the *Roughness* tab, enter "0.03" as the *Specify roughness value*.

The roughness values are Manning's *n* values.

6. Select "left_bank" from the list on the left.

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

8. Enter "50.0" as the *Exx* (eddy viscosity).

9. On the *Roughness* tab, enter "0.045" as the *Specify roughness value* option.

10. Select "right_bank" from the list on the left.

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

12. Enter "100.0" as the *Exx* (eddy viscosity).

13. On the *Roughness* tab, enter "0.04" as the *Specify roughness value*.

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

The material properties have now been properly defined. The material zones can be displayed by doing the following:

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

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

17. On the *2D Mesh* tab, turn on *Materials* and click **OK** to close the *Display Options* dialog.

With *Materials* turned on, the project will appear similar to Figure 2.

18. When done reviewing the materials, follow steps 15–17 but turn off *Materials* in step 17.
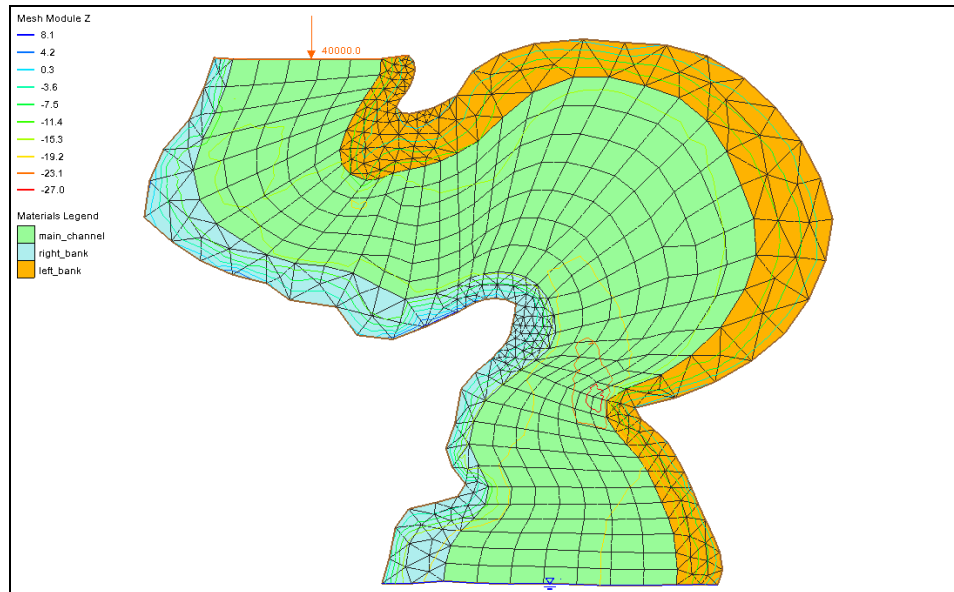
Figure 2   Mesh with Materials turned on in the Display Options dialog

3   Model Parameters

RMA2 includes many model parameters that may be set to represent various conditions. These include physical attributes such as water temperature and density, and weather conditions such as wind, general material properties, and numeric controls. These are set in the *RMA2 Model Control* dialog. For this simulation, use the default values.

To review these settings:

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

2. Review the available options on each of the tabs.

3. When done reviewing the options, click **Cancel** to close the *RMA2 Model Control* dialog.

4   Saving the Simulation

The boundary conditions (inflow rate and head at the outflow) were previously defined inside the map module. These were read in with the simulation. The entire simulation can now be saved. To save the simulation:

1. Select *File* / **Save As** to bring up the *Save As* dialog.

2. Select "Project Files (*.sms)" from the *Save as type* drop-down.

3. Enter "stmary_ready.sms" as the *File name*.

4. Click **Save** to save the simulation and exit the *Save As* dialog.

5   Running the Simulation

The **Run RMA2** command actually performs several tasks. These include:

- Performing a model check to detect missed components. If no problems are detected, this step produces no visible effects. If the model is missing a required component (for example, if no boundary conditions existed), or there is an error in the simulation (such as an invalid mesh domain), a list of problems is displayed.

- Running the Geometry File Generation (GFGen) program. Before running the finite element analysis, the ASCII geometry file created by SMS must be converted to a binary format that RMA2 can understand. The program is launched automatically when the simulation runs. The location of the GFGen executable is stored as a model preference. The progress of GFGen will be displayed in a *GFGen* model wrapper dialog.

- Running the RMA2 simulation program. Once the binary geometry file is generated, the model *GFGen* wrapper dialog waits to move on to the actual simulation. The location of the RMA2 executable is also stored as a model preference. The progress of the model is displayed in the *RMA2* model wrapper dialog.

To run the simulation:

1. Select *RMA2* / **Run RMA2** to bring up the *GFGen* model wrapper dialog.

2. If a dialog asks to find an executable file for the model run, click on the folder icon and navigate to *Program Files\SMS 13.0 64-bit\models\TABS\gfgenv456.exe*.

3. Once GFGen is finished, click **Run RMA2** to close the *GFGen* model wrapper dialog and bring up the *RMA2* model wrapper dialog.

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

For this simulation, RMA2 should finish quickly. If running in Demo Mode, the solution "stmary_ready.sol" is found in the *tutorials/ RMA2/data files/output* directory and can be opened with the *File* / **Open…** command.

With the solution loaded, evaluate the results. To do this:

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 *Contours* and *Vectors* and turn off *Elements*.

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

5. On the *Vectors* tab, in the *Arrow Options* section, select "Scale length to magnitude" from the *Shaft Length* drop-down.

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

7. Select " water surface elevation" to make it active.

The project should appear similar to Figure 3. View the other RMA2 solutions by selecting " velocity mag" or " water depth" in the Project Explorer.
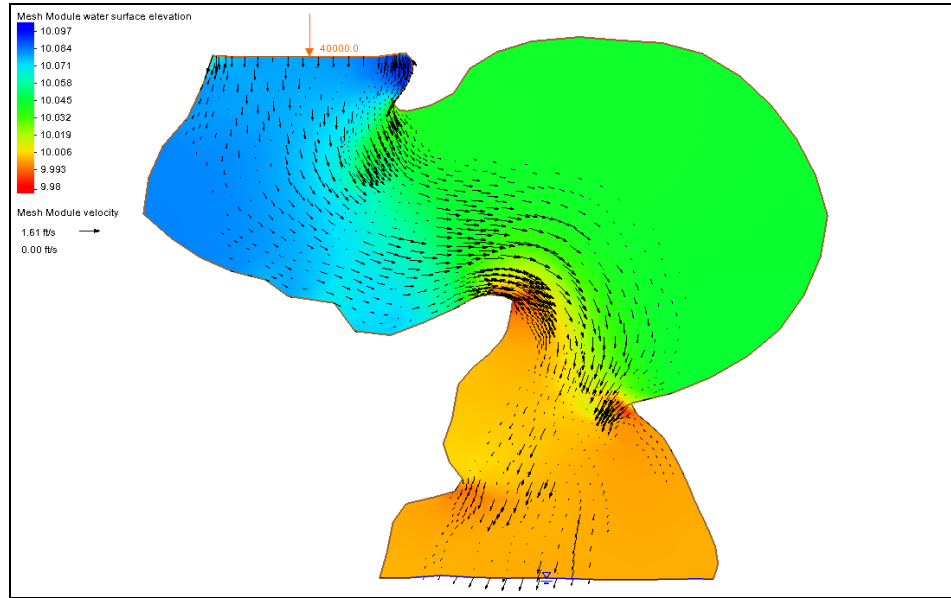
Figure 3   The velocity magnitude solution set

6   Conclusion

This concludes the "Basic RMS2 Analysis" tutorial. Feel free to continue to experiment with the SMS interface, or exit the program.