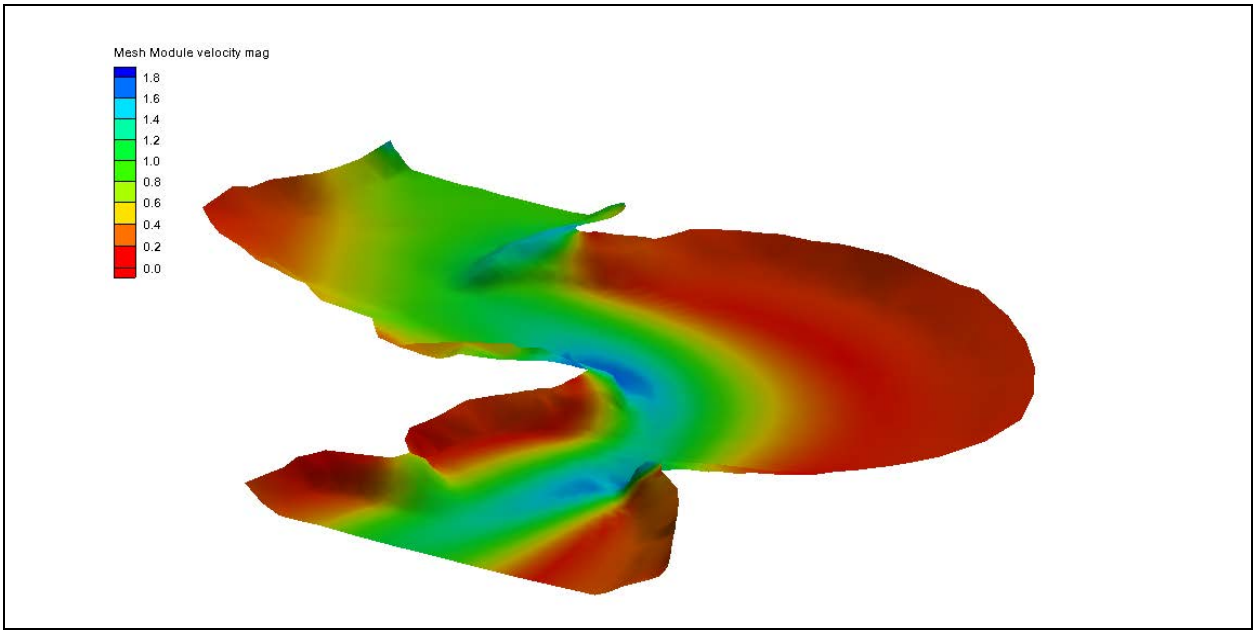
SMS

v. 11.1

SMS 11.1 Tutorial

Basic RMA2 Analysis



Objectives

This lesson will teach you how to prepare a mesh for an RMA2 simulation

Prerequisites

- Overview Tutorial

Requirements

- RMA2
- GFGEN
- Mesh Module

Time

- 15-35 minutes

AQUAVEO™

1 Introduction

You will be using the project file stmary.sms. This project contains a simulation (".sim" file) for RMA2. The file needed for this tutorial can be found in the Data Files Folder for this tutorial. 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 open the file:

1. Select *File* | *Open*.
2. Open the file *stmary.sms* from the Data Files Folder.
3. 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.

The mesh that is read in includes geometry (nodes and elements), material properties, and boundary conditions.

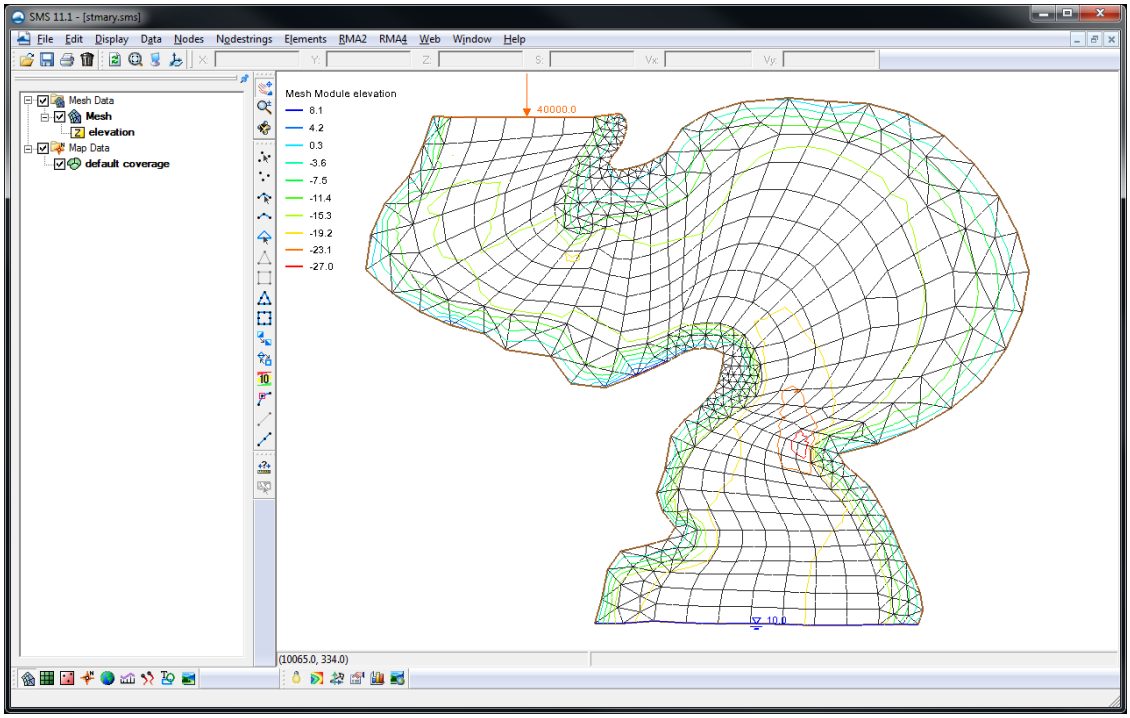


Figure 1. The mesh from stmary.sms.

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*.
2. In the *RMA2 Material Properties* dialog, highlight the material *main_channel* (ID 1).
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 50 for the eddy viscosity (E_{xx}).
4. Under the *Roughness* tab, specify a roughness value of 0.03. The roughness values are Manning's n values.
5. Highlight the material *left_bank* (ID 3). Set E_{xx} to 50 and set the roughness value to 0.045.
6. For the material labeled *right_bank* (ID 2), Set E_{xx} of 100 (higher turbulence requires a higher viscosity value) and set the roughness value to 0.04.
7. Click *OK*.

The material properties have now been properly defined.
Note: the material zones can be displayed by opening the *Display Options* dialog and turning on the *Materials* option under the *2D Mesh* tab.

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, weather conditions such as wind, general material properties, and numeric controls. These are set in the *RMA2* | *Model Control* command. For this simulation, we will use the default values. If you want to examine these:

1. Select *RMA2* | *Model Control*.
2. Peruse through the tabs looking at the options.
3. Click the *OK* button when you are done.

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.*
2. Make sure the *Save as type* is *Project Files* and enter the name *stmary_ready.sms*.
3. Click the *Save* button to save the simulation.

5 Running the Simulation

To run the simulation:

- Select *RMA2* | *Run RMA2.*

This command actually performs several tasks. These include:

1. 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 posted for the user.
2. 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 user runs the simulation. The location of the *GFGEN* executable is stored as a model preference. The progress of *GFGEN* will be displayed in a *Model Wrapper* dialog.
3. Running the *RMA2* simulation program. Once the binary geometry file is generated, the *Model Wrapper* dialog waits for the user 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 *Model Wrapper* dialog.

For this simulation, *RMA2* should finish quickly. The *Model Wrapper* dialog waits for the user to acknowledge the completion of the model run. By default, it will then load the solution file when you click the *Exit* button. (If you are running in *Demo Mode*, the solution *stmary_ready.sol* is found in the *tutorials/tut09_RMA2_StMary/output* directory and can be opened with the *File|Open* command.)
With the solution loaded, you are ready to evaluate the results. To do this:

1. Open the *Display Options* dialog.
2. Under the *2D Mesh* tab, make sure the *Contours* and *Vectors* options are checked.
3. Under the Contour Options tab, select Color Fill as the Contour Method.
4. Under the *Vectors* tab, make sure *Scale length to magnitude* as the option for *Shaft Length* is selected.
5. Close the *Display Options* dialog.

The *RMA2* solutions for velocity magnitude, water depth and water surface elevation can be viewed by selecting the desired data set in the *Project Explorer*.

6 Conclusion

This concludes the *Basic RMS2 Analysis* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.