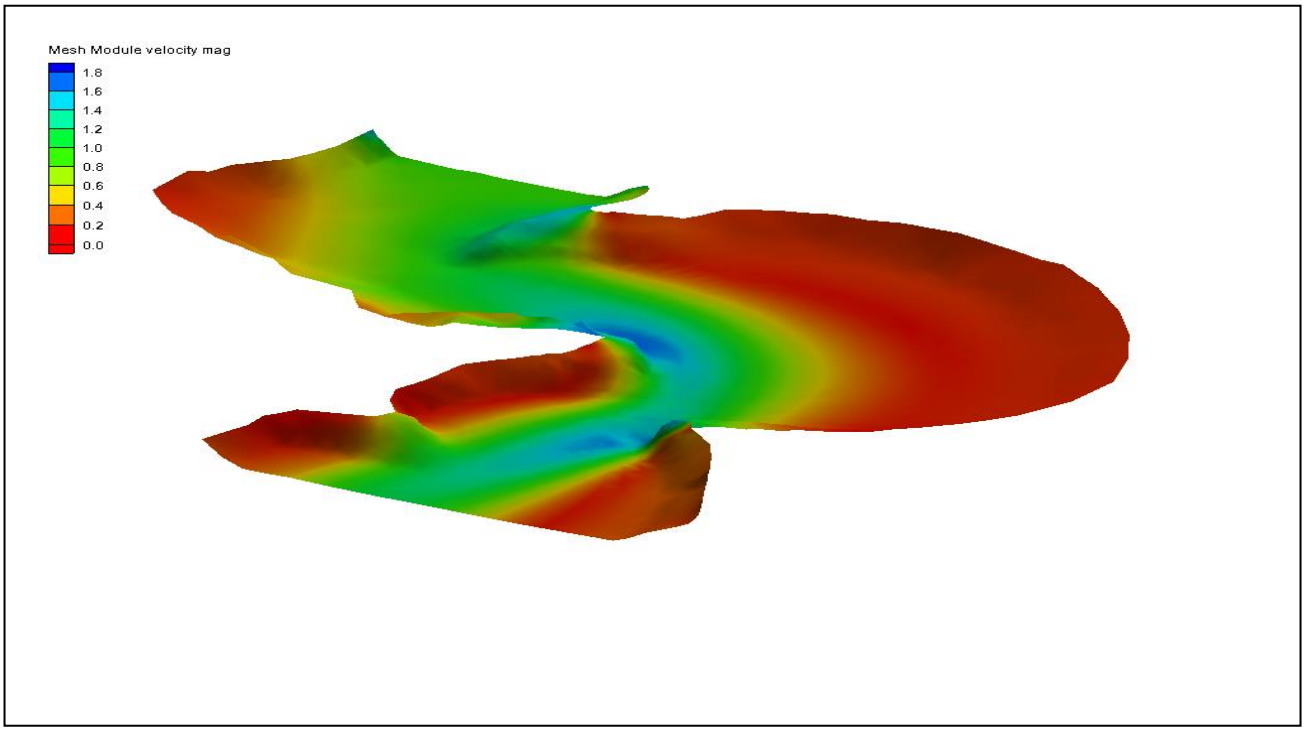
SMS

v. 12.1

SMS 12.1 Tutorial

Basic RMA2 Analysis



Objectives

This tutorial instructs on how to prepare a mesh for an RMA2 simulation.

Prerequisites

- Overview Tutorial

Requirements

- RMA2
- GFGEN
- Mesh Module

Time

- 15-25 minutes

AQUAVEO™

1 Introduction

Start by opening 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** to bring up the *Open* dialog.
2. Locate the file "stmary.sms" from the data files folder for this tutorial and click **Open**. If geometry is still open from a previous tutorial, a prompt will ask if wanting to delete existing data. If this happens, click the **Yes** button.

The mesh that is read in will appear in the Graphics Window includes geometry (nodes and elements), material properties, and boundary conditions (Figure 1).

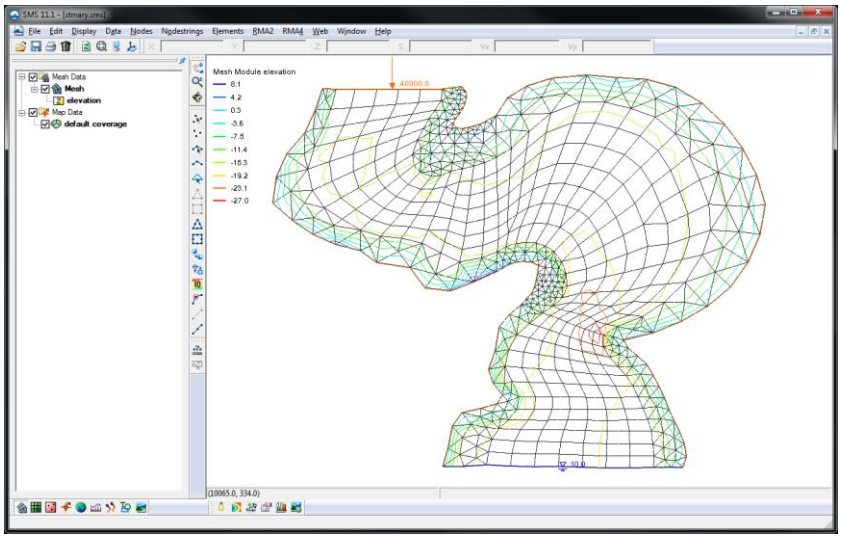


Figure 1 Mesh from stmary.sms.

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. Click on *main_channel* in the list on the left to highlight it.
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 (*Exx*).
4. Under the *Roughness* tab, enter "0.03" for the *Specify roughness value* option. The roughness values are Manning's *n* values.
5. Highlight the material *left_bank* from the list on the left.
6. 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 (*Exx*).
7. Under the *Roughness* tab, enter "0.045" for the *Specify roughness value* option. The roughness values are Manning's *n* values.
8. Highlight the material *right_bank* from the list on the left.
9. 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 "100" for the eddy viscosity (*Exx*).
10. Under the *Roughness* tab, enter "0.04" for the *Specify roughness value* option. The roughness values are Manning's *n* values.
11. Click **OK** to close the *RMA2 Material Properties* dialog.

The material properties have now been properly defined.

Note: the material zones can be displayed by opening the *Display Options* (*Display* | **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, use the default values. If wanting to examine these:

1. Select *RMA2* | **Model Control** to bring up the *RMA2 Model Control* dialog.
2. Peruse through the tabs looking at the options.
3. Accept the default options the click the **OK** 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. Make sure the *Save as type* is "Project Files (*.sms)" and enter "stmary_ready.sms" for the *File name*.
3. Click the **Save** button to save the simulation.

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**.
    - If a dialog asks to find an executable file for the model run, click on the folder icon and locate the requested executable.
2. In the *GFGen* model wrapper, click **Run RMA2** when done to start the *RMA2* model run.
3. Make certain the Load solution option is checked on, then click **Exit** when the *RMA2* model wrapper finishes.

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. Select *Display* | **Display Options** to bring up the *Display Options* dialog.

2. Highlight *2D Mesh* in the list on the left then make sure the *Contours* and *Vectors* options are turned on and *Elements* is turned off.
3. Switch to the *Contours* tab, and select "Color Fill" as the *Contour Method*.
4. Under the *Vectors* tab, make sure "Scale length to magnitude" is selected as the option for *Shaft Length*.
5. Close the *Display Options* dialog by clicking **OK**.

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

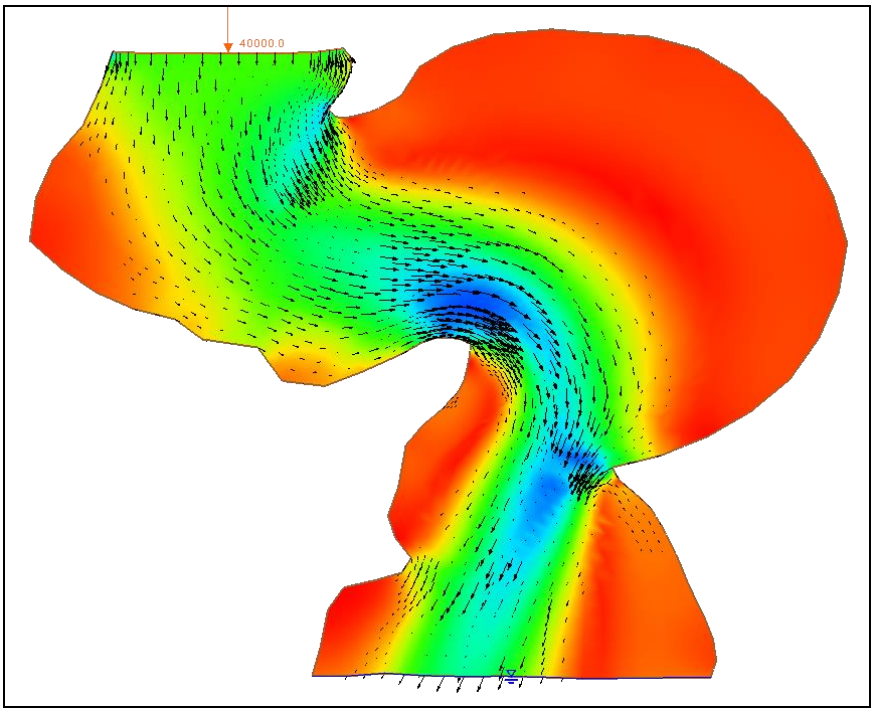


Figure 2 The velocity magnitude solution set

6 Conclusion

This concludes the *Basic RMS2 Analysis* tutorial. Continue to experiment with the SMS interface or quit the program.