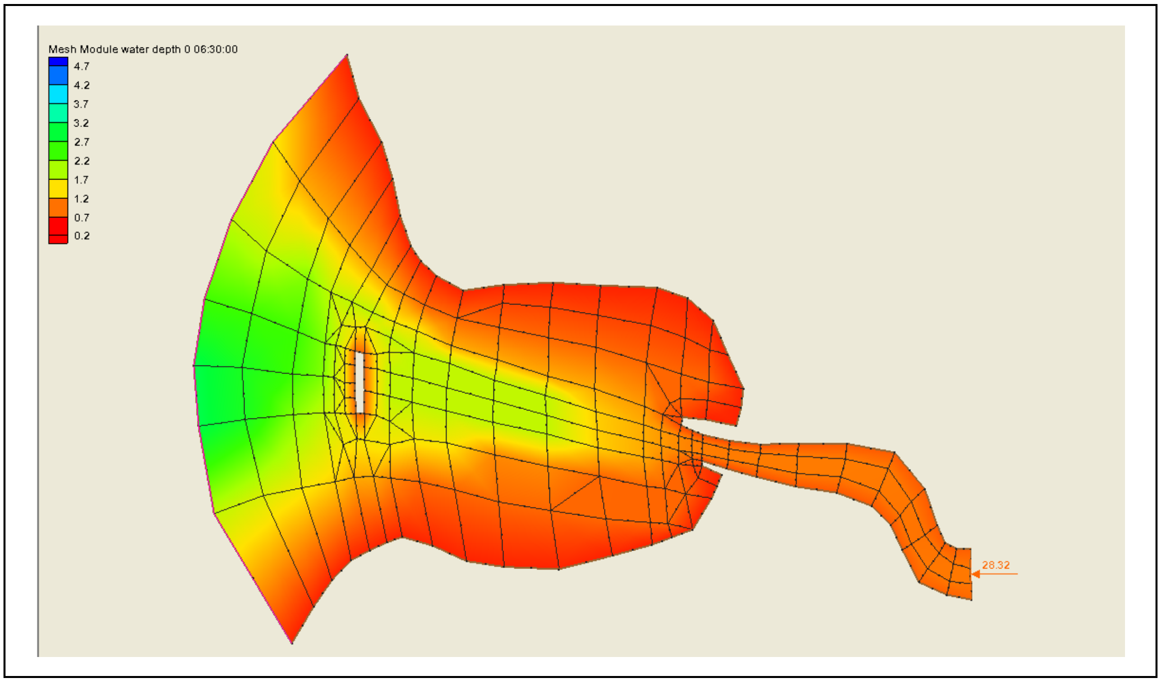
v. 11.1

SMS 11.1 Tutorial

RMA4 Analysis



Objectives

This lesson will teach you how to run a solution using *RMA4*. If you have not yet completed Lesson 4 on *RMA2*, you should do so now. *RMA4* is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D models. In this lesson, you will use *RMA4* to model 3 situations: an inflow of a constituent into a river, the inflow of a constituent into a bay, and salinity intrusion. Each case uses metric units for both the *RMA2* solution file and the *RMA4* input. We recommend that you consistently use metric units to avoid possible scaling mistakes

Prerequisites

- Overview Tutorial
- RMA2 Tutorial

Requirements

- RMA4
- RMA2
- Mesh Module

Time

- 30-45 minutes

1 Case 1

RMA4 can only be run after having initially run a solution in *RMA2*. This is because *RMA4* uses the flow solutions computed by *RMA2* to compute the constituent concentration as it flows through the mesh. An *RMA2* geometry and solution have been supplied. To open the *RMA2* files:

1. Select *File* | *Open*.
2. Select the file "madora.sms" from the Data Files Folder for this tutorial. If you still have geometry open, you will be asked if you want to delete existing data. If this happens, click the Yes button.

The geometry will be displayed on the screen with the *RMA2* boundary conditions.

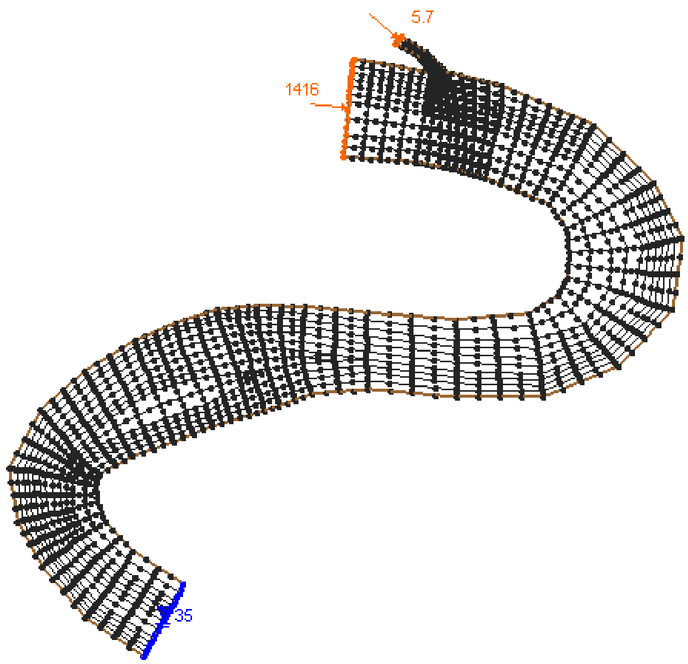


Figure 1 Madora mesh

This mesh was created in Metric units because *RMA4* requires Metric units. The main channel has a flow of 1,416 cms (50,000 cfs) with a channel entering with a flow of 5.7 cms (200 cfs).

1.1 RMA4 Model Control

RMA4 is a transient model. The *RMA2* solution is a steady state solution. *RMA4* will assume a steady flow throughout the mesh, but the boundary conditions will change. To set the time that *RMA4* will run:

1. Select *RMA4* | *Model Control*.
2. In the *General* tab, make sure the *Start Time* is set to 0.0, and set the *Time Step* to 0.5 (h), the *Total Steps* to 49, and the *Max Time* to 24 (h).
3. In the *Files* tab, make sure the *Last time step used from the RMA2 velocity file* is set to 0.0 (hrs) and the Time *subtracted from the RMA2 velocity file* is set to 0.0 (hrs).
4. Under *RMA2 Solution File* select the file button icon and choose "madora.sol".
5. Make sure that *Write RMA4 Solution File* is checked.
6. Turn on *Activate full report* in the *Informational Files* section.
7. Click *OK* to exit the RMA4 Model Control dialog.

1.2 Boundary Conditions

For this model, a pollutant has been dumped into the smaller channel for three hours. The concentration of the pollutant in the stream is 1,000 ppm. To apply this boundary condition:

1. Select the *Select Nodestring* tool from the *Toolbox*.
2. Select the nodestring at the smaller inflow boundary (labeled as 5.7).
3. If the arrows are not pointing into the larger channel select the *Nodestrings* | *Reverse Direction.*
4. Select *RMA4* | *Assign BC.*
5. Switch to *Transient* and push the *Curve undefined* button.
6. The *XY Series Editor* dialog appears. In this dialog, a time series curve can be created. To turn the pollutant on for only 3 hours:
    a. Enter the following *Time/Concentration* values:

Time	Concentration
0.0	1000.0
3.0	1000.0
3.1	0.0
24.0	0.0

    b. Push *OK* to exit the *XY Series Editor*.

7. Push *OK* to exit the *RMA4 Assign BC* dialog.

In this case, we applied 1,000 ppm as a boundary condition. *RMA4* does not care about the units of the concentration because the output is relative to the initial number you specify. For example, since we specified a concentration of 1,000, the values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration was ppm, ppt, or kg/kg; *RMA4* treats all concentrations as relative values.

1.3 Material Properties

The final step is to specify the Material diffusion. To do this:

1. Select *RMA4 | Material Properties*.
2. Select *bank* material (on the left side).
3. Set the *Dx* and *Dy* diffusion coefficients to 10.0 (m^2/s).
4. Repeat steps 2 and 3 for all remaining materials.
5. Click *OK* to exit the dialog.

Because *RMA4* does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the x and y directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.

1.4 Run RMA4

To run *RMA4* the *.bin file must be in the same directory you are saving to (where solution files will be saved). If the madora.bin file is not in this folder, you must either copy over this file (provided in the Data Files Folder1) or rerun RMA2.
You are now ready to save the data and run *RMA4*. To do this:

1. Select *File | Save Project (madora.sms)*. *RMA4* requires that the *RMA2* and *RMA4* filenames be the same, so you must save the project as *madora.sms*.
2. Select *RMA4 | Run RMA4.*
3. If the prompt shows a message that *RMA4* is *not found*, click the *File Browser* button and manually find the correct program executable.

When *RMA4* finishes, it will create the file *madora.qsl*, which is the solution file containing the constituent data at each node.

4. Click the *Exit* button to load the solution. Make sure that the *Load solution* button is on before exiting.

1.5 Film Loop

Once a solution has been created by *SMS*, you can use a number of features to view the results and adjust the model to better approximate the observed values. The easiest way to view the results from the *RMA4* solution is to use the film loop. Select File | Open.

To create the film loop:

1. Click on *constituent 1*- so it is active in the tree item.
2. Select *Data | Film Loop…* to bring up the *Film Loop Setup* dialog.
3. Make sure *Create AVI File* is selected and click on the *File Browser* to save this new loop as *smsloop1.avi*
4. Make sure the *Transient Data Animation* option is selected, then press *Next*.
5. Run the simulation for 23.5 hours starting at time “0 00:30:00”. Make sure that *Specify Number of Frames* is selected and set to 48.
6. Click *Next.*
7. Click the *Display Options* button .
8. Switch to the *2D Mesh* tab and turn off everything but *Elements* and *Contours*.
9. Select the *Contours* tab and choose *Color fill* as the *Contour Method*. Select to *Specify a range* from a *Min* of 0.0 to a *Max* of 10.0. (This is done because when the stream flow enters the main channel, the concentration quickly drops to between 0.0 and 10.0 ppm.)
10. Push *OK* to get back to the *Film Loop* dialog.
11. Push *Finish* to start generating the film loop. Each frame of the film loop will be generated. After the film loop is done generating, a new window will come up to play the results.
12. Close the AVI application when you are finished watching the results.

You will notice how the concentration drops quickly as the pollutant enters the main channel. This occurs because the inflow from the small channel is 0.4% of the inflow from the main channel.

2 Case 2

In this case, a constituent is coming into Noyo Bay from a river. The files for this case can be opened by:

1. Select *File* | *Open*.
2. Select the file noyo1.sms from the Data Files Folder. If you still have geometry open, you will be asked if you want to delete existing data and materials. If this happens, click the Yes button for both dialogs.

The geometry will be displayed on the screen with the *RMA2* boundary conditions as shown in Figure 2.

This mesh was initially created in English units and later converted to metric units to use in *RMA4*. The river flowing into Noyo Bay has a flow of 28.32 cms. The water surface elevation on the left side varies as the tide comes in and out over a 12-hour cycle repeated twice a day.

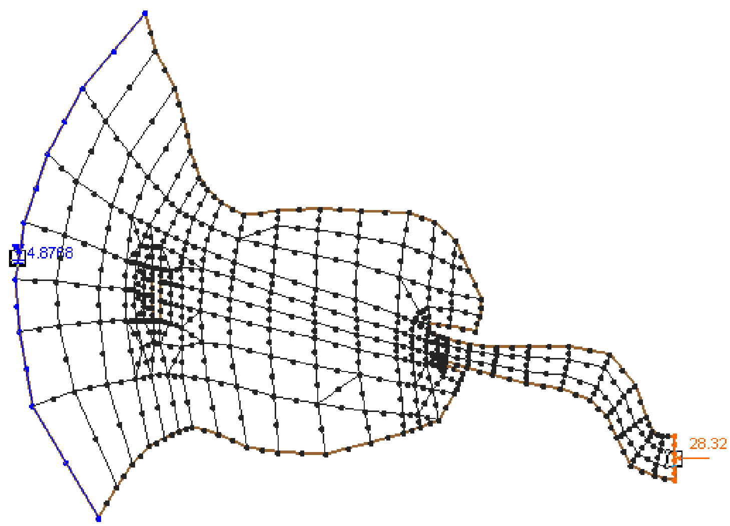


Figure 2 Noyo mesh

2.1 RMA4 Model Control

To set the model controls:

1. Select *RMA4* | *Model Control*.

2. In the *General* tab, make sure the *Start Time* is set to 0.0, and set the *Time Step* to 0.5 (h), the *Total Steps* to 49, and the *Max Time* to 24 (h).
3. In the *Files* tab, set the *Last time step used from the RMA2 velocity file* to 24.0 (hrs) and the *Time subtracted from the RMA2 velocity file* to 12.0 (hrs). This will cause RMA4 to use the last 12 hours of the RMA2 solution.
4. Click the folder icon button under *RMA2 Solution File* and select noyo1.sol.
5. Make sure that *Write RMA4 Solution File* is checked.
6. Turn on *Activate full report* in the Informational Files section.
7. Click *OK* to exit the RMA4 Model Control dialog.

2.2 Boundary Conditions

For this model, a constant inflow of 100 ppm of a pollutant enters the bay from the river. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the right side of the model and select *RMA4 | Assign BC*.
3. Set a *Constant Concentration* of 100.0 (ppm) and push *OK* to exit the *RMA4 Assign BC* dialog.

2.3 Material Properties

To apply the diffusion:

1. Select RMA4 | Material Properties.
2. Select *material_01* (on the left side).
3. Set the *Dx* and *Dy* diffusion coefficients to 1.0 (m^2/s).
4. Repeat steps 2 and 3 *for material_02*.
5. Click *OK* to exit the dialog.

2.4 Run RMA4

1. Save the noyo1.sms project.
2. Run *RMA4* as you did for Case 1.

3. After *RMA4* runs, open the solution file *noyo1.qsl*. As before, the *.bin file (noyo1.bin) must be in the save directory before running RMA4.

2.5 Film Loop

1. Generate a film loop named *smsloop2.avi* using the same steps as for case 1 (Section 2.5) with the following exceptions:
    - Run the simulation for 24 hours with 49 frames.
    - Do not set a range in the *Data Range* section of the *Contour Options* dialog.

3 Case 3

For this final case, we will view salinity intrusion into Noyo Bay.

1. Open *noyo2.sms*. If you still have geometry open, you will be asked if you want to delete existing data and materials. If this happens, click the *Yes* button for both dialogs.

3.1 RMA4 Model Control

To set the model times:

1. Select *RMA4 | Model Control*.
2. In the *General* tab, make sure the *Start Time* is set to 0.0, and set the *Time Step* to 0.5 (h), the *Total Steps* to 49, and the *Max Time* to 24 (h).
3. In the *Files* tab, set the *Last time step used from the RMA2 velocity file* to 24.0 (hrs) and the *Time subtracted from the RMA2 velocity file* to 12.0 (hrs). This will cause RMA4 to use the last 12 hours of the RMA2 solution.
4. Click the folder icon button under *RMA2 Solution File* and select noyo2.sol.
5. Make sure that *Write RMA4 Solution File* is checked.
6. Turn on *Activate full report* in the Informational Files section.
7. Click *OK* to exit the RMA4 Model Control dialog.

3.2 Boundary Conditions

For this model, a constant concentration of 8 ppm exists offshore and enters the bay from the left. To apply this boundary condition:

1. Select the *Select Nodestring* tool from the Toolbox.
2. Select the nodestring at the left side of the model and select *RMA4 | Assign BC*.
3. Set a *Constant* Concentration of 8.0 (ppm).
4. Select the *Factor applied when flow direction changes* and set the *Shock factor* to 0.5.
5. Push *OK* to exit the RMA4 Assign BC dialog.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

3.3 Material Properties

To apply the diffusion:

1. Select *RMA4 | Material Properties*.
2. Select *material_01* (on the left side).
3. Set the *Dx* and *Dy* diffusion coefficients to 1.0 (m^2/s).
4. Repeat steps 2 and 3 *for material_02*.
5. Push *OK* to exit the dialog.

3.4 Run RMA4

1. Save the noyo2.sms project and run RMA4 as you did for Cases 1 and 2.
2. Run *RMA4* as you did for Cases 1 and 2.
3. After *RMA4* runs, open the solution file *noyo2.qsl*.

3.5 Film Loop

Generate a film loop named *smsloop3.avi* using the same steps as for case 1 (Section 2.5) with the following differences:

- Set the simulation to run for 12 hours starting at 0 06:00:00. Also, set the Number of Frames to 25. Running these times will show a full tidal cycle that runs continuously.
- Do not set a range in the Data Range section of the Contour Options dialog.

4 Other Changes

You may want to play with the shock factor and diffusion coefficients to see how they affect the model. Other options include:

- Change the *diffusion coefficients* in all 3 cases to 0.5 and then try 10.0 to see the differences.
- Change the *shock factor* in the third case to 0.0 and 1.0. There is a large difference in how far the intrusion gets into the bay.

5 Conclusion

It is easiest to consistently use metric units when running *RMA4*. However, you may have an *RMA2* mesh and solution in English units that you want to use for *RMA4*. In this case, we recommend that you convert the coordinates of your *RMA2* mesh using the *Edit | Reproject…* command, change your boundary conditions and material properties to metric, and rerun *RMA2* before setting up *RMA4*.
This concludes the *RMA4 Analysis* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.