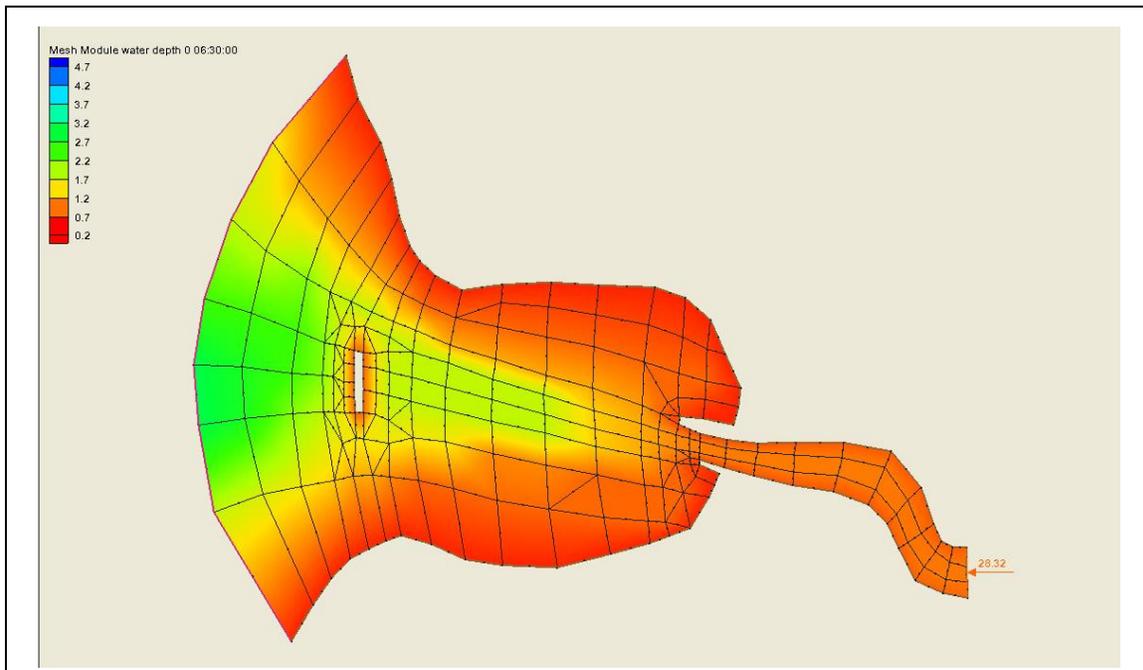


# SMS 12.1 Tutorial

## RMA4 Analysis



### Objectives

This lesson will instruct on how to run a solution using RMA4. If having not yet completed the RMA2 tutorial, review that tutorial before continuing. RMA4 is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D models. This lesson will use RMA4 to model three 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. It is recommended to consistently use metric units to avoid possible scaling mistakes.

### Prerequisites

- Overview Tutorial
- RMA2 Tutorial

### Requirements

- RMA4
- RMA2
- Mesh Module

### Time

- 30-45 minutes



<b>1</b>	<b>Case 1</b> .....	<b>2</b>
1.1	RMA4 Model Control .....	3
1.2	Boundary Conditions .....	3
1.3	Material Properties .....	4
1.4	Run RMA4 .....	4
1.5	Film Loop.....	5
<b>2</b>	<b>Case 2</b> .....	<b>6</b>
2.1	RMA4 Model Control .....	6
2.2	Boundary Conditions .....	7
2.3	Material Properties .....	7
2.4	Run RMA4 .....	7
2.5	Film Loop.....	7
<b>3</b>	<b>Case 3</b> .....	<b>8</b>
3.1	RMA4 Model Control .....	8
3.2	Boundary Conditions .....	8
3.3	Material Properties .....	9
3.4	Run RMA4 .....	9
3.5	Film Loop.....	9
<b>4</b>	<b>Other Changes</b> .....	<b>9</b>
<b>5</b>	<b>Conclusion</b> .....	<b>9</b>

## 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** to bring up the *Open* dialog.
2. Locate and select the file “madora.sms” from the data files folder for this tutorial and click **Open**. If SMS currently has geometry open, a prompt will appear asking if wanting to delete existing data. If this happens, click the **Yes** button.

The geometry will be displayed in the Graphics Window with the RMA2 boundary conditions (Figure 1).

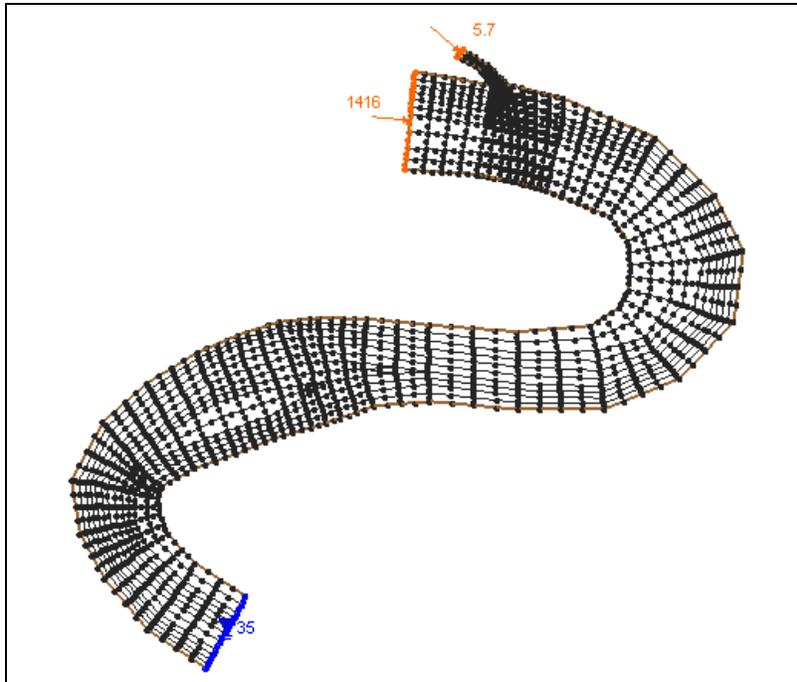


Figure 1 Madora mesh

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** to open the *RMA4 Model Control* dialog.
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. The *Open* dialog will appear. Locate and select “madora.sol” then click **Open**.
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 to select the nodestring at the smaller inflow boundary (labeled as 5.7).
  - a) If the arrows are not pointing into the larger channel select the *Nodestrings / Reverse Direction*.
2. Select *RMA4 / Assign BC*. The *RMA4 Assign Boundary Conditions* dialog will open.
3. Switch to *Transient* and *Concentration* and push the **Curve undefined** button. The *XY Series Editor* dialog appears.
4. A time series curve can be created in this dialog. To turn the pollutant on for only three 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) Click **OK** to exit the *XY Series Editor*.
5. Click **OK** to exit the *RMA4 Assign BC* dialog.

In this case, 1,000 ppm was applied as a boundary condition. RMA4 does not care about the units of the concentration because the output is relative to the initial number specified. For example, since a concentration of 1,000 was specified, the values in the solution will range from 0 to 1,000 as the plume spreads downstream. The concentration could be 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* to open the *RMA4 Material Properties* dialog.
2. Select “bank” material from the menu on the left of the dialog box.
3. Set the *Dx* and *Dy diffusion coefficients* to “10.0” ( $\text{m}^2/\text{s}$ ).
4. Repeat steps 2 and 3 for all remaining materials.
5. Click **OK** to exit the *RMA4 Material Properties* 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 users are saving to (where solution files will be saved). If the “madora.bin” file is not in this folder, either copy over this file (provided in the data files folder) or rerun RMA2.

Now 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 save the project as “madora.sms”.
2. Select *RMA4* / **Run RMA4**. A new *RMA4* window will open and show the program running.
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 turned on before exiting.

## 1.5 Film Loop

---

Once a solution has been created by SMS, a number of features can be used 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.

To create the film loop:

1. Click on “constituent 1 –” in the Project Explorer 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** . The *Save* dialog will appear.
4. Give this new loop a *File name* of “smsloop1.avi” then click **Save**.
5. Make sure the *Transient Data Animation* option is selected then press **Next**.
6. Set *Run Simulation For* as “23.5” hours and the *Starting at Time* as “0 00:30:00”. Make sure that *Specify Number of Frames* is selected and set to “48”. Click **Next**.
7. Click the **Display Options**  button to bring *Display Options* dialog.
8. Make sure *2D Mesh* is selected from the menu on the left then turn off everything except *Elements* and *Contours*.
9. Select the *Contours* tab and choose “Color fill” as the *Contour Method*. Under *Data Range*, 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 close the *Display Options* dialog and get back to the *Film Loop Setup* 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 finished watching the results by clicking red  button in the top right corner.

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** to bring up the *Open* dialog.
2. Select the file “noyo1.sms” from the data files folder and click **Open**. If there is still geometry open a prompt will ask if wanting to delete existing data and materials. If this happens, click the **Yes** button.

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.

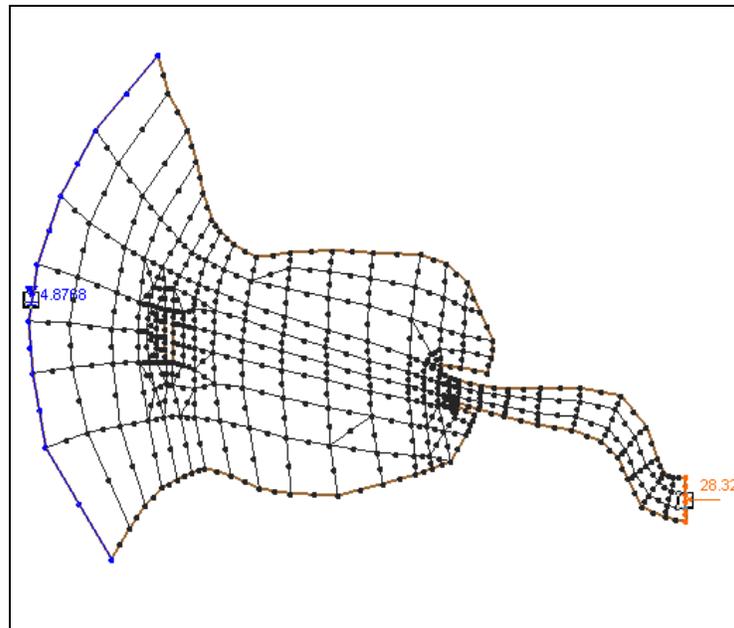


Figure 2 Noyo mesh

### 2.1 RMA4 Model Control

To set the model controls:

1. Select *RMA4* | **Model Control** to open the *RMA4 Model Control* dialog.
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  button under *RMA2 Solution File* to bring up *Open* dialog. Select the file “noyo1.sol” and click **Open**.
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 and select the nodestring at the right side of the model.
2. Select *RMA4 / Assign BC* to open the *RMA4 Assign Boundary Conditions* dialog.
3. Set a *Constant Concentration* of “100.0” (ppm) and click **OK** to exit the *RMA4 Assign BC* dialog.

---

## 2.3 Material Properties

---

To apply the diffusion:

1. Select *RMA4 | Material Properties* to bring up the *RMA4 Material Properties* dialog.
2. Select “material\_01” from the menu on the left.
3. Set the *Dx* and *Dy diffusion coefficients* to “1.0” (m<sup>2</sup>/s).
4. Repeat steps 2 and 3 for “material\_02”.
5. Click **OK** to exit the *RMA4 Material Properties* dialog.

---

## 2.4 Run RMA4

---

1. Save the “noyo1.sms” project by selecting *File / Save Project*.
2. Run RMA4 by selecting *RMA4 | Run RMA4*.
3. When the RMA4 model run is finished, click **Exit** after making certain the *Load solution* option is checked on.

---

## 2.5 Film Loop

---

1. Generate a film loop named “smsloop2.avi” using the same steps as for case 1 (Section 1.5) 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, view salinity intrusion into Noyo Bay.

1. Use the *File* | **Open** command to bring up the *Open* dialog.
2. Select the “noyo2.sms” file and click **Open**. Click **Yes** if asked if its alright to continue and delete data.

#### 3.1 RMA4 Model Control

---

To set the model times:

1. Select *RMA4* | **Model Control**. The *RMA4 Model Control* dialog will open.
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  button under *RMA2 Solution File* and **Open** select “noyo2.sol” in the *Open* dialog.
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 and select the nodestring at the left side of the model.
2. Select *RMA4* | **Assign BC** to open the *RMA4 Assign Boundary Conditions* dialog.
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** to bring up the *RMA4 Material Properties* dialog.
2. Select “material\_01” from the menu on the left.
3. Set the *Dx* and *Dy diffusion coefficients* to “1.0” (m<sup>2</sup>/s).
4. Repeat steps 2 and 3 for “material\_02”.
5. Push **OK** to exit the *RMA4 Material Properties* dialog.

---

### 3.4 Run RMA4

---

1. Save the “noyo2.sms” project by selecting *File* / **Save Project**.
2. Run RMA4 by selecting *RMA4* | **Run RMA4**.
3. When the RMA4 model run is finished, click **Exit** after making certain the *Load solution* option is checked on.

---

### 3.5 Film Loop

---

Generate a film loop named “smsloop3.avi” using the same steps as for case 1 (Section 1.5) 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

---

If desired, 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, there may be an RMA2 mesh and solution in English units needing to use RMA4. In this case, it's recommend to convert the coordinates of the RMA2 mesh using the *Edit* / **Reproject...** command then change the boundary conditions and material properties to metric and rerun RMA2 before setting up RMA4.

This concludes the *RMA4 Analysis* tutorial. Continue to experiment with the SMS interface or quit the program.