SMS 12.3 Tutorial
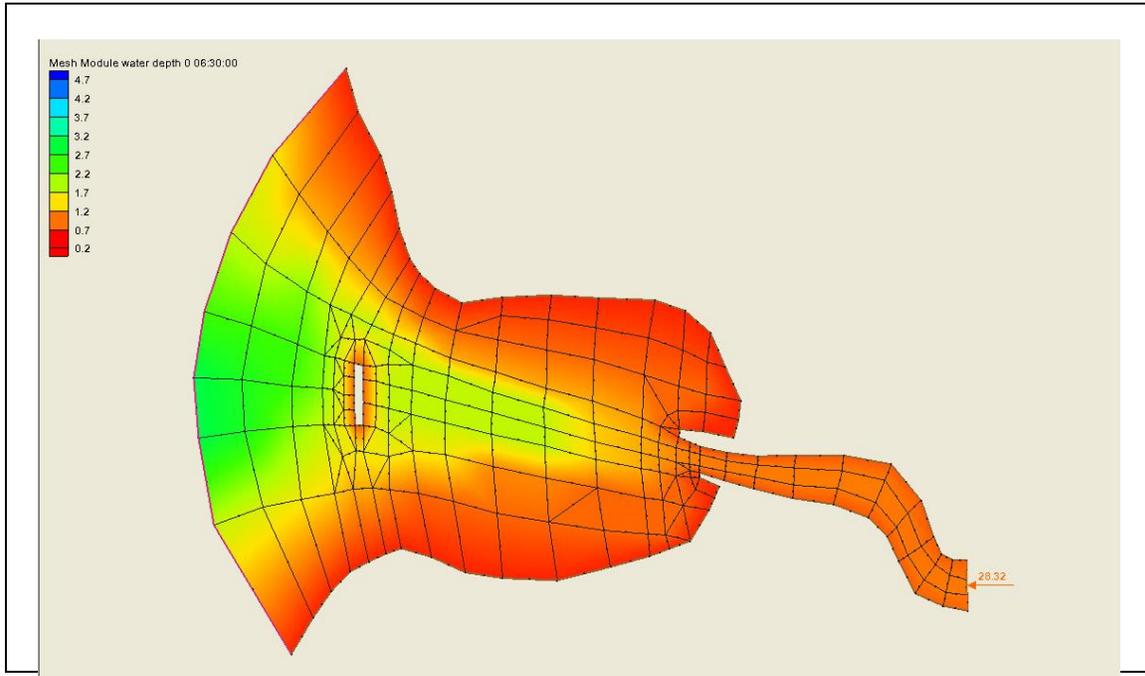
RMA4 Analysis

Objectives

Learn how to run a solution using RMA4.

Prerequisites

- Overview Tutorial
- Basic RMA2 Analysis

Requirements

- RMA4
- RMA2
- Mesh Module

Time

- 30–45 minutes

AQUAVEO™

1   Introduction

RMA4 is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D models. 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.

This tutorial uses 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.

2   Case 1

An RMA2 geometry and solution have been supplied as part of the files for this tutorial.

To open the RMA2 files:

1. Select *File | **Open…*** to open 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 "madora.sms".

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

5. If SMS currently has geometry open, a prompt will appear asking to delete existing data. If this happens, click **Yes**.

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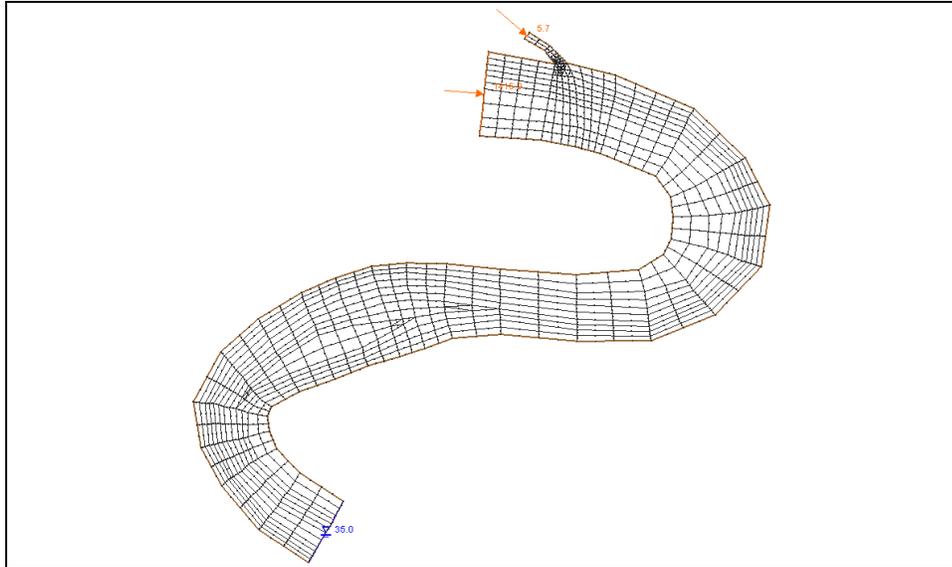
Figure 1   Madora mesh

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).

2.1   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. On the *General* tab, in the *Time control* section, enter "0.0" as the *Start Time.*

3. Enter "0.5" as the *Time Step.*

4. Enter "49" as the *Total Steps.*

5. Enter "24.0" as the *Max Time*.

6. On the *Files* tab, in the *RMA4 Input Files* section, enter "0.0" as both the *Last time step used from the RMA2 velocity file* and the *Time subtracted from the RMA2 velocity file*.

7. Click the 📁 button under *RMA2 Solution File* to bring up the *Open* dialog.

8. Browse to the *data files\madora\TABS/madora* folder and select "madora.sol".

9. Click **Open** to exit the *Open* dialog.

10. In the *RMA4 Output Files* section, turn on *Write RMA4 Solution File*.

11. In the *Informational Files* section, turn on *Activate full report*.

12. Click **OK** to exit the *RMA4 Model Control* dialog.

2.2   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. Using the **Select Nodestring** ⸙ tool, select the nodestring at the smaller inflow boundary (labeled as "5.7").

2. If the inflow boundary arrows are not pointing toward the larger channel, select *Nodestrings* / **Reverse Direction**.

3. Select *RMA4* / **Assign BC…** to bring up the *RMA4 Assign Boundary Conditions* dialog.

4. In the *Concentration* section, select *Transient* and click **Curve undefined** to bring up the *XY Series Editor* dialog.

A time series curve can be created in this dialog. To turn the pollutant on for only three hours:

5. Enter the following values into the appropriate columns in the dialog:

Time	Concentration
0.0	1000.0
3.0	1000.0
3.1	0.0
24.0	0.0

6. Click **OK** to exit the *XY Series Editor* dialog.

7. Click **OK** to exit the *RMA4 Assign Boundary Conditions* 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. Because RMA4 treats all concentrations as relative values, the concentration could be ppm, ppt, or kg/kg.

2.3   Material Properties

The final step before running RMA4 is to specify the material diffusion. To do this:

1. Select *RMA4* / **Material Properties…** to open the *RMA4 Material Properties* dialog.

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

3. Enter "10.0" as both *Dx* and *Dy*.

These are the diffusion coefficients.

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.

2.4   Run RMA4

To run RMA4, the BIN file must be in the same directory as where solution files are saved (in this case, *data files\madora\TABS\madora*). If "madora.bin" is not in this folder, either copy over this file (provided in the data files folder) or rerun RMA2.

Now 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.

2.   Select *RMA4* / **Run RMA4…** to bring up the *RMA4* model wrapper dialog.

3.   If a prompt advises that RMA4 is not found, click the **File Browser** 📁 button and manually find the correct program executable.

RMA4 creates the solution file "madora.qsl" upon finishing. This solution file contains the constituent data at each node.

4.   When RMA4 finishes, turn on *Load solution* and click **Exit** to import the solution and close the *RMA4* model wrapper dialog.

2.5   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.   Select "🔲 constituent 1-" in the Project Explorer to make it active.

2.   Select *Data* / **Film Loop…** to bring up the *General Options* page of the *Film Loop Setup* dialog.

3.   In the *Film Loop Files* section, turn on *Create AVI File* and click 📁 to bring up the *Save* dialog.

4.   Enter "smsloop1.avi" as the *File name* and click **Save** to close the *Save* dialog.

5.   In the *Select Film Loop Type* section, select *Transient Data Animation*.

6.   Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.

7.   In the *Filmloop Time* section, enter "23.5" in the *Run Simulation for* field and select "hours" from the drop-down.

8.   Enter "0 00:30:00" as the *Starting at Time*.

9.   Select *Specify Number of Frames* and enter "48" in the field to the right.

10. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.

11. In the *Options* section, click **Display Options** 🖥 to bring up the *Display Options* dialog.

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

13. On the *2D Mesh* tab, click **All Off**, then turn on *Elements* and *Contours*.

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

15. In the *Data Range* section, turn on *Specify a range* and enter "0.0" as the *Min* and "10.0" as the *Max*.

This is done because the concentration quickly drops to between 0.0 and 10.0 ppm when the stream flow enters the main channel.

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

17. Click **Finish** to close the *Film Loop Setup* dialog and start generating the film loop.

After the all frames of the film loop finish generating, the external Play AVI Application will open. Notice how the concentration drops quickly as the pollutant enters the main channel (Figure 2). This occurs because the inflow from the small channel is 0.4% of the inflow from the main channel.
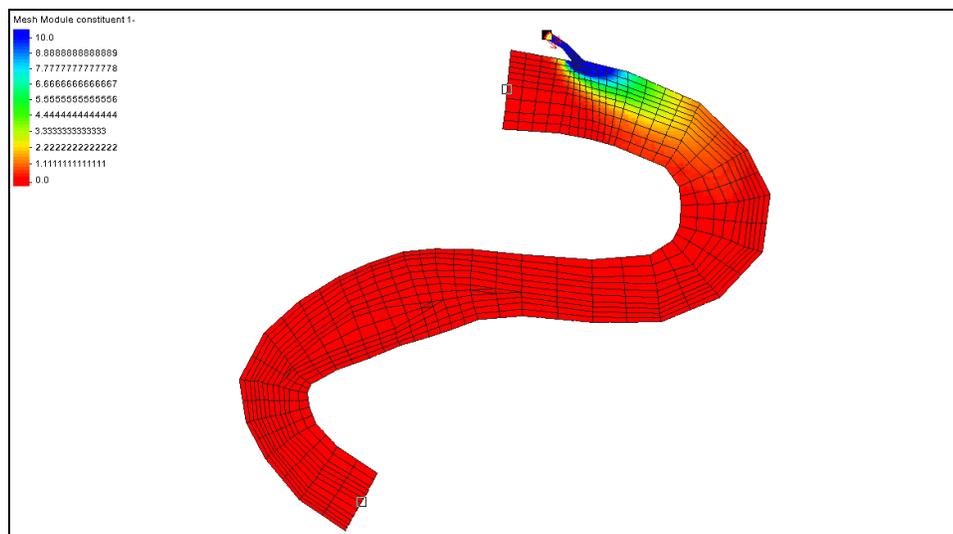

Figure 2   Pollutant concentration drops quickly upon entering main channel

18. When done watching the film loop, close the *Play AVI application* by clicking the ❌ button in the top right corner of the application window.

3   Case 2

In this case, a constituent is coming into Noyo Bay from a river. 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.

The files for this case can be opened by doing the following:

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 "noyo1.sms".

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

5. If asked to delete existing data and materials, click **Yes**.

The geometry will be displayed on the screen with the RMA2 boundary conditions as shown in Figure 3.
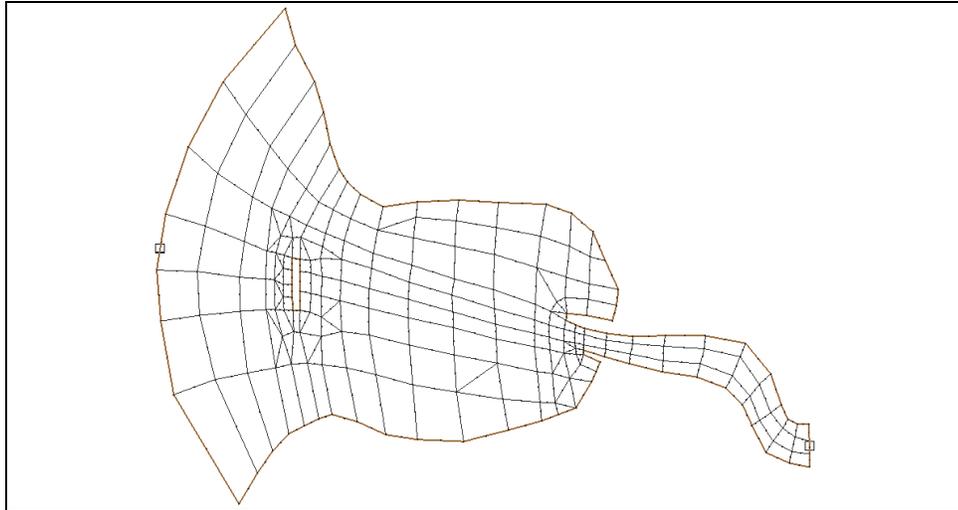

Figure 3   Noyo mesh

3.1   RMA4 Model Control

To set the model controls:

1. Select *RMA4 | **Model Control…*** to open the *RMA4 Model Control* dialog.

2. On the *General* tab, in the *Time control* section, enter "0.0" as the *Start Time*.

3. Enter "0.5" as the *Time Step*.

4. Enter "49" as the *Total Steps*.

5. Enter "24.0" as the *Max Time*.

6. On the *Files* tab, in the *RMA4 Input Files* section, enter "24.0" as the *Last time step used from the RMA2 velocity file*.

7. Enter "12.0" as the *Time subtracted from the RMA2 velocity file*.

This causes RMA4 to use the last 12 hours of the RMA2 solution.

8. Click the 📁 button under *RMA2 Solution File* to bring up the *Open* dialog.

9. Browse to the *data files\noyo1\TABS\noyo1* folder and select "noyo1.sol".

10. Click **Open** to exit the *Open* dialog.

11. In the *RMA4 Output Files* section, turn on *Write RMA4 Solution File*.

12. In the *Informational Files* section, turn on *Activate full report*.

13. Click **OK** to exit the *RMA4 Model Control* dialog.

3.2   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, do the following:

1. Using the **Select Nodestring** ⬈ tool, select the small nodestring at the right side of the model.

2. Select *RMA4* | **Assign BC…** to open the *RMA4 Assign Boundary Conditions* dialog.

3. In the *Concentration* section, enter "100.0" as the *Constant.*

4. Click **OK** to exit the *RMA4 Assign Boundary Conditions* dialog.

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 list on the left.

3. Enter "1.0" for both *Dx* and *Dy*.

These are the diffusion coefficients.

4. Repeat steps 2 and 3 for "material_02".

5. Click **OK** to exit the *RMA4 Material Properties* dialog.

3.4   Run RMA4 and Create a Film Loop

1. **Save** 🖫 the project.

2. Select *RMA4* | **Run RMA4…** to bring up the *RMA4* model wrapper dialog.

3. When the RMA4 model run finishes, turn on *Load solution* and click **Exit** to close the *RMA4* model wrapper dialog.

4. Repeat steps 1–17 in Section 2.5 to generate a film loop, but enter "smsloop2.avi" in step 4, "24.0" in step 7, "49" in step 9, and turn off *Specify a range* in step 15 (a data range does not need to be entered this time).

Notice how the constituent diffuses throughout the bay over the 24 hour period (Figure 4).
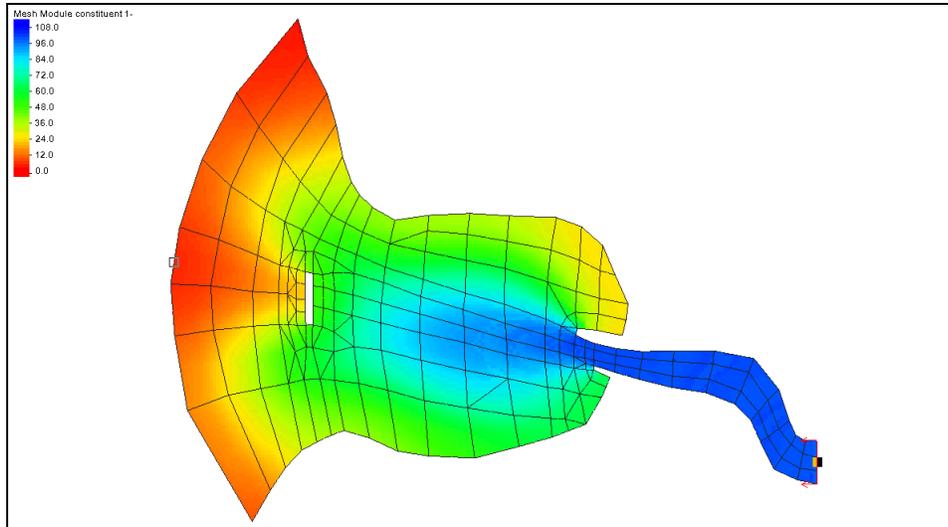
Figure 4   Frame 8 of the AVI playback showing the diffusion throughout the bay

4   Case 3

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

1. Click **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 "noyo2.sms".
4. Click **Open** to import the project and exit the *Open* dialog.
5. Click **Yes** if asked to delete existing data.

4.1   RMA4 Model Control

To set the model times:

1. Select *RMA4* | **Model Control…** to bring up the *RMA4 Model Control* dialog.
2. On the *General* tab, in the *Time control* section, enter "0.0" as the *Start Time*.
3. Enter "0.5" as the *Time Step*.
4. Enter "49" as the *Total Steps*.
5. Enter "24.0" as the *Max Time*.
6. On the *Files* tab, in the *RMA4 Input Files* section, enter "24.0" as the *Last time step used from the RMA2 velocity file*.
7. Enter "12.0" as the *Time subtracted from the RMA2 velocity file*,

This causes RMA4 to use the last 12 hours of the RMA2 solution.

8. Click the 📁 button to bring up the *Open* dialog.
9. Browse to the *data files\noyo2\TABS\noyo2* folder and select "noyo2.sol".
10. Click **Open** to exit the *Open* dialog.

11. In the *RMA4 Output Files* section, turn on *Write RMA4 Solution File*.

12. In the *Informational Files* section, turn on *Activate full report*.

13. Click **OK** to exit the *RMA4 Model Control* dialog.

4.2   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. Using the **Select Nodestring** ⬑ tool, select the large nodestring at the left side of the model.

2. Select *RMA4* | **Assign BC…** to open the *RMA4 Assign Boundary Conditions* dialog.

3. In the *Concentration* section, enter "8.0" as the *Constant*.

4. In the lower section, select *Factor applied when flow direction changes*.

5. Enter "0.5" as the *Shock factor*.

6. Click **OK** to exit the *RMA4 Assign Boundary Conditions* 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.

4.3   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 list on the left.

3. Enter "1.0" for both *Dx* and *Dy*.

This is the diffusion coefficient.

4. Repeat steps 2 and 3 for "material_02".

5. Click **OK** to exit the *RMA4 Material Properties* dialog.

4.4   Run RMA4 and Create a Film Loop

1. **Save** 🖫 the project.

2. Select *RMA4* | **Run RMA4…** to bring up the *RMA4* model wrapper dialog.

3. When the RMA4 model run finishes, turn on *Load solution* and click **Exit** to close the *RMA4* model wrapper dialog.

4. Repeat steps 1–17 in Section 2.5 to generate a film loop, but enter "smsloop3.avi" in step 4, "12.0" in step 7, "0 06:00:00" in step 8, "25" in step 9, and turn off *Specify a range* in step 15 (a data range does not need to be entered this time).

Running these times will show a full tidal cycle that runs continuously. Figure 5 shows frame 12 in the animation.
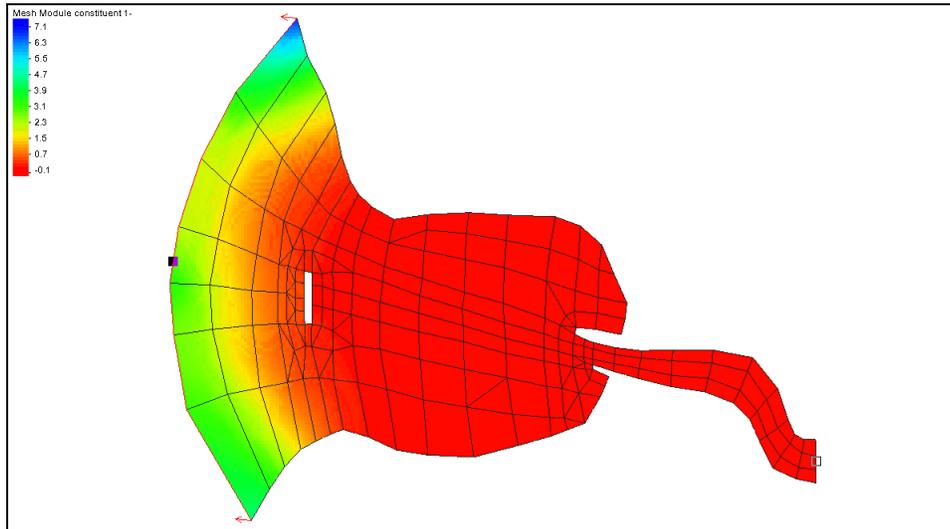

Figure 5   Salinity intrusion into the bay

5   Other Changes

If desired, alter 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.

6   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 is recommend to convert the coordinates of the RMA2 mesh to metric 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. Feel free to continue experimenting with the SMS interface or quit the program.