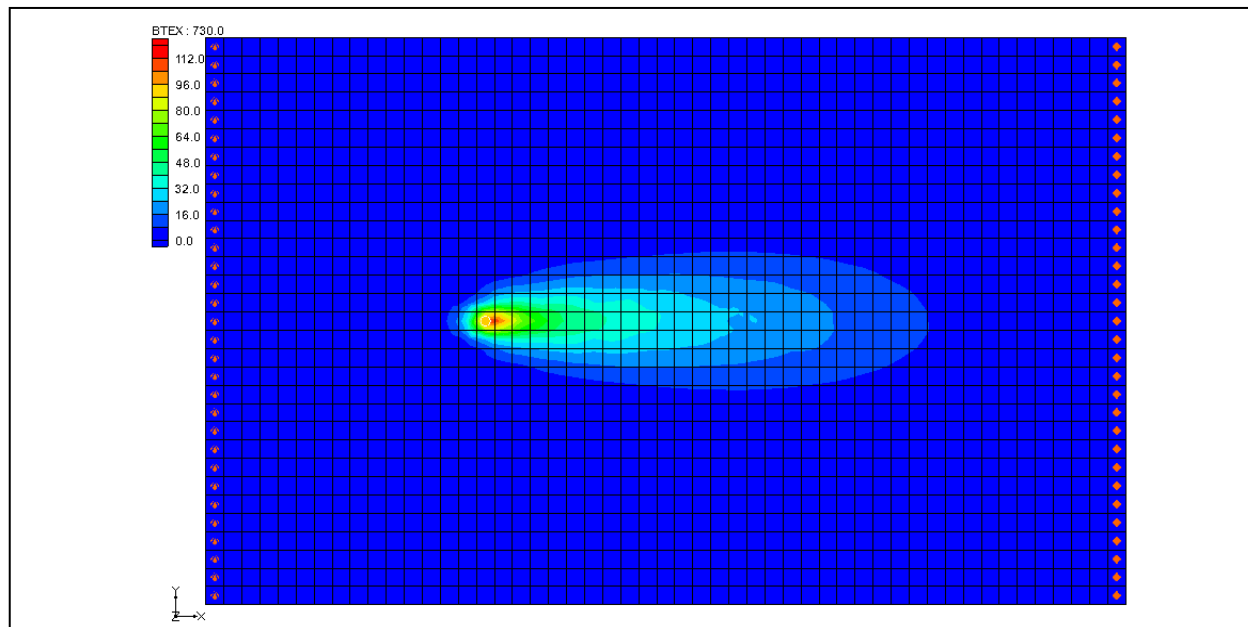


GMS 10.5 Tutorial

RT3D – Instantaneous Aerobic Degradation



Objectives

This tutorial demonstrates how to simulate the reaction for instantaneous aerobic degradation of hydrocarbons using a predefined user reaction package.

Prerequisite Tutorials

- MT2DMS – Grid Approach

Required Components

- Grid Module
- MODFLOW
- RT3D

Time

- 15–30 minutes



| | | |
|----------|--|-----------|
| 1 | Introduction..... | 2 |
| 1.1 | Description of Problem | 3 |
| 1.2 | Description of Reaction..... | 3 |
| 1.3 | Getting Started | 4 |
| 2 | Building the MODFLOW Model..... | 4 |
| 2.1 | Conceptual Model Approach vs. Grid Approach | 4 |
| 2.2 | Units | 4 |
| 2.3 | Creating the Grid..... | 5 |
| 2.4 | Initializing the Model | 5 |
| 2.5 | The Global Package | 5 |
| 2.6 | The LPF Package | 6 |
| 2.7 | Defining the Spill Well..... | 6 |
| 2.8 | Assigning the Head BC | 7 |
| 2.9 | Running MODFLOW..... | 8 |
| 3 | Building the Transport Model | 9 |
| 3.1 | Initializing the Model | 9 |
| 3.2 | The Basic Transport Package..... | 9 |
| 3.3 | Assigning Concentrations to the Left Boundary..... | 11 |
| 3.4 | The Advection Package..... | 12 |
| 3.5 | The Dispersion Package | 12 |
| 3.6 | The Source/Sink Mixing Package | 13 |
| 3.7 | The Chemical Reactions Package..... | 13 |
| 3.8 | Running RT3D | 13 |
| 3.9 | Viewing the Results..... | 14 |
| 3.10 | Viewing a Film Loop Animation..... | 15 |
| 4 | Conclusion..... | 15 |

1 Introduction

This tutorial illustrates the steps involved in performing a reactive transport simulation using the RT3D model. Several types of contaminant reactions can be simulated with RT3D by using a predefined or user-defined reaction package. The reaction simulated in this tutorial is instantaneous aerobic degradation of hydrocarbons using a predefined user reaction package.

The tutorial will review:

- Creating a 3D grid
- Defining MODFLOW inputs and boundary conditions
- Running MODFLOW
- Defining the transport model
- Running RT3D
- Viewing a film-loop animation

1.1 Description of Problem

The problem in this tutorial is shown in Figure 1. The site is a 510 m x 310 m section of a confined aquifer with a flow gradient from left to right. An underground storage tank is leaking fuel hydrocarbon contaminants at 2 m³/day at the location shown. Concentration of the hydrocarbons is 1000 mg/L and the spill may be assumed to be devoid of oxygen. Details of the aquifer hydrology and geometry are given below. Initial levels of hydrocarbon and oxygen in the aquifer are 0.0 and 9.0 mg/L, respectively. This tutorial will simulate a continuous spill event and compute the resulting hydrocarbon and oxygen contours after one year using instantaneous degradation kinetics.

The first part of the problem will be to compute a flow model of the site using MODFLOW. Using the flow field computed by MODFLOW, a reactive transport model will then be defined using RT3D.

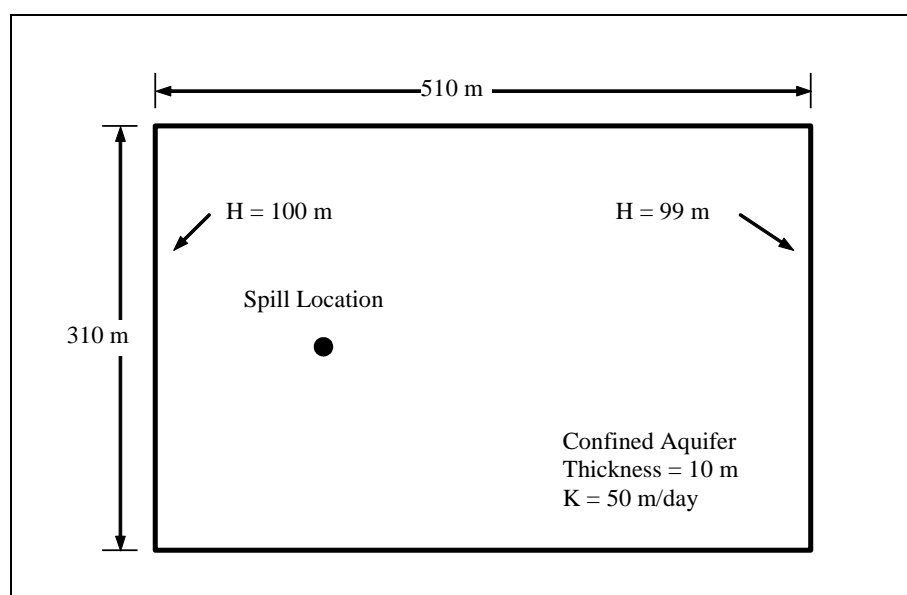
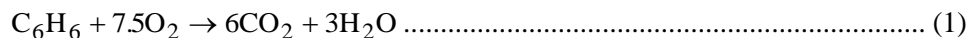


Figure 1 Sample problem to be modeled with RT3D

1.2 Description of Reaction

The reaction model used here simulates the instantaneous degradation of fuel hydrocarbons under aerobic conditions based on the following reaction (benzene basis):



Therefore, the mass ratio of benzene to oxygen, F, is 3.08 (i.e., 3.08 g of oxygen to 1 g of benzene). The instantaneous removal of either contaminant (H) or oxygen (O) is dictated by the following algorithm, where t refers to a particular time step:

$$\text{H}(t+1) = \text{H}(t) - \text{O}(t) / F \text{ and } \text{O}(t+1) = 0, \text{ when } \text{H}(t) > \text{O}(t) / F \dots\dots\dots (2)$$

$$\text{O}(t+1) = \text{O}(t) - \text{H}(t) \cdot F \text{ and } \text{H}(t+1) = 0, \text{ when } \text{O}(t) > \text{H}(t) \cdot F \dots\dots\dots (3)$$

Given this algorithm, either hydrocarbon or oxygen concentration in a given grid cell will be reduced to zero at each time step, depending on which component is stoichiometrically limiting in the prior time step.

1.3 Getting Started

Do the following to get started:

1. If GMS is not open, launch GMS.
2. If GMS is already open, select the *File / New* command to ensure the program settings are restored to the default state.
3. If asked to save changes, click **Don't Save** to close the dialog and restore GMS to a default state.

2 Building the MODFLOW Model

The first part of the simulation is to build a MODFLOW flow model. In this case, a steady state flow model will be used with the transport simulation.

2.1 Conceptual Model Approach vs. Grid Approach

In many cases, the most efficient way to create a MODFLOW simulation is to build a conceptual model of the site using the feature objects in the Map module. However, since this problem is a simple rectangular model, use the grid approach and edit values directly on the cells.

2.2 Units

First to define the units:

1. Select the *Edit | Units...* command to open the *Units* dialog.
2. For the *Length* unit, click the “...” button to open the *Display Projection* dialog.
3. Change the vertical and horizontal *Units* to “Meters”.
4. Click **OK** to close the *Display Projection* dialog.
5. For the *Time* unit, ensure that “d” is selected.
6. For the *Mass* unit, ensure that “mg” is selected.
7. For the *Force* unit, ensure that “N” is selected.
8. For the *Concentration* unit, ensure that “mg/l” is selected.

9. Select **OK** to close the *Units* dialog.

2.3 Creating the Grid

The first step in defining the MODFLOW simulation is to create a single layer grid.

1. In the Project Explorer, right-click on the empty space and select **New / 3D Grid...** to bring up the *Create Finite Difference Grid* dialog.
2. In the *X-Dimension* section, enter “510” for the *length* and “51” for *Number cells*.
3. In the *Y-Dimension* section, enter “310” for the *length* and “31” for *Number cells*.
4. In the *Z-Dimension* section, enter “1” for *Number cells* (the length in Z does not matter).
5. Select **OK** to close the *Create Finite Difference Grid* dialog and create the 3D grid.

At this point, a simple grid should appear in the Graphics Window.

2.4 Initializing the Model

Now to initialize the MODFLOW data:

1. In the Project Explorer, right-click on the “ grid” item and select **New MODFLOW...** to open the *MODFLOW Global/Basic Package* dialog.

2.5 The Global Package

The next step is to define the data in the MODFLOW Global Package.

Starting Head

First, define an initial head of 100 meters everywhere in the grid:

1. Turn off the *Starting heads equal grid top elevations* option.
2. Select the **Starting Heads...** button to open the *Starting Heads* dialog.
3. Select the **Constant → Grid...** button to open the *Grid Value* dialog.
4. Enter a value of “100”.
5. Select **OK** to close the *Grid Value* dialog.

6. Select **OK** to exit the *Starting Heads* dialog.



For the layer elevations, there is a value of 10.0 m for the top elevation. The bottom elevation is at zero. The existing grid already has these values, so these values do not need to be entered.

It is necessary to change some values in the IBOUND array to designate the constant head boundaries. Do this later by selecting the cells and using the **Cell Properties** command.

7. Select **OK** to exit the *MODFLOW Global/Basic Package* dialog.

2.6 The LPF Package

Next to define the data in the LPF package:

1. In the Project Explorer, expand the “ MODFLOW” item.
2. Right-click on the “ LPF” package and select **Properties...** to open the *LPF Package* dialog.
3. Under *Layer type*, select the *Confined* option.

Next to enter a horizontal hydraulic conductivity value of 50 m/day.


4. Select the **Horizontal Hydraulic Conductivity...** button to open the *Horizontal Hydraulic Conductivity* dialog.
5. Select the **Constant → Grid...** button to open the *Grid Value* dialog.
6. Enter a value of “50”.
7. Select **OK** to close the *Grid Value* dialog.
8. Select **OK** to exit the *Horizontal Hydraulic Conductivity* dialog.

The rest of the default options in the package should be sufficient.

9. Select **OK** to exit the *LPF Package* dialog.

2.7 Defining the Spill Well


The spill will be simulated by defining an injection well at the location of the spill with a small injection rate. During the transport simulation, a concentration will be assigned to the well. Do the following to define the well:

1. Select the **Select Cells**  tool then, while watching the I, J, K fields in the status bar, locate the cell at I=16, J=16, K=1.

2. Right-click on the selected cell and select the **Sources/Sinks...** command from the pop-up menu to open the *MODFLOW Sources/Sinks* dialog.
3. Select the *Wells (WEL)* item in the list box.
4. Select the **Add BC** button.
5. Enter a value of “2.0” (m^3/d) for the *Q (flow)*.
6. Select **OK** to exit the *MODFLOW Sources/Sinks* dialog.

2.8 Assigning the Head BC

The final step before running MODFLOW is to assign the constant head boundary conditions (BC) on the left and right sides of the model. First, assign the head BC on the left side.

1. Select the **Select j**  tool and select the leftmost column in the grid.
2. Right-click on any of the cells in the leftmost column in the grid and select the **Properties...** command to open the *3D Grid Cell Properties* dialog.
3. In the *IBOUND* row, switch the option to “Specified head” in the drop-down list.
4. Leave the *Starting head* value at “100.0”.
5. Select **OK** to close the *3D Grid Cell Properties* dialog.

Next to assign the head boundary condition on the right side.

6. Right-click on any of the cells in the rightmost column in the grid and select the **Properties...** command to open the *3D Grid Cell Properties* dialog.
7. In the *IBOUND* row, switch the option to “Specified head” in the drop-down list.
8. Enter “99” for the *Starting head* value.
9. Select **OK** to close the *3D Grid Cell Properties* dialog.

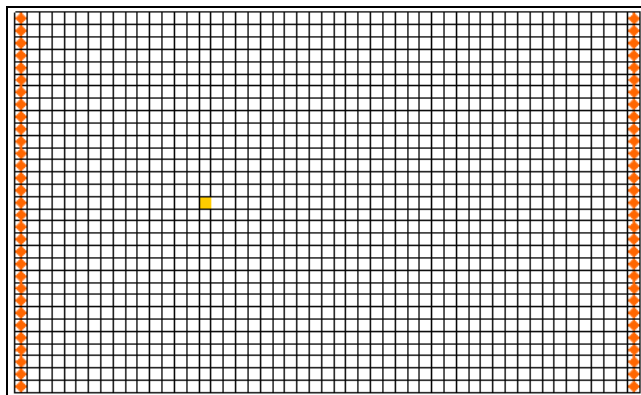


Figure 2 The MODFLOW grid

2.9 Running MODFLOW

At this point, it is possible to save the model and run MODFLOW.

1. Select the *File* | **Save As...** command to open the *Save As* dialog.
2. Create or locate a folder where the model can be saved.
3. Enter “flowmod” for the name of the simulation.
4. Select the **Save** button.

To run MODFLOW:

5. Select the *MODFLOW* | **Run MODFLOW** command to start the *MODFLOW* model wrapper.
6. When the simulation is finished, select the **Close** button to exit the *MODFLOW* model wrapper.

GMS imports the solution automatically, showing a series of contours indicating a flow from left to right. Note: The color ramp may need to be reversed to match Figure 3.

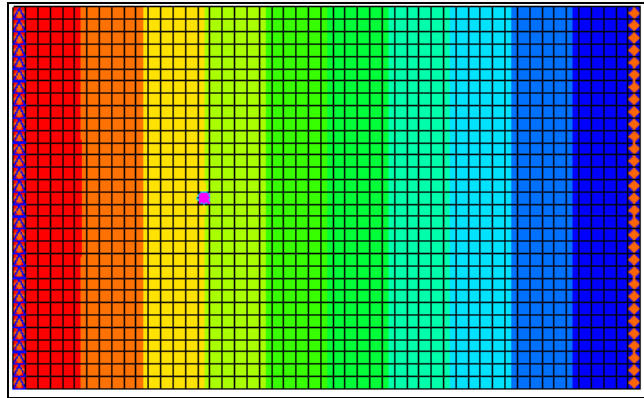



Figure 3 The MODFLOW solution

3 Building the Transport Model

Now that the flow model is established, the next step is to perform the RT3D simulation. For this part of the simulation, select the reaction, define the reaction data, define the supplemental layer data needed by RT3D, and assign concentrations to the well.

3.1 Initializing the Model

The first step is to initialize the RT3D data.

1. Right-click on the “ grid” item in the Project Explorer and select the **New MT3DMS...** command to open the *Basic Transport Package* dialog.

3.2 The Basic Transport Package

Now, enter the data for the Basic Transport Package. Start with selecting RT3D as the transport model and selecting the appropriate packages.

1. In the *Model* section, select the *RT3D* option.
2. Select **Packages...** to open the *MT3DMS/RT3D Packages* dialog.
3. Select the following packages:
 - *Advection package*
 - *Dispersion package*
 - *Source/sink mixing package*
 - *Chemical reaction package*
4. Under *RT3D reactions*, select the “Inst. Aerobic Deg. of BTEX” option.

5. Select **OK** to exit the *MT3DMS/RT3D Packages* dialog.

Starting Concentrations

Note that in the layer data section of the dialog, the species associated with the reaction being modeled are listed by name. The next step is to define the starting concentration for each of these species. By default, all of the starting concentrations are zero. This is the correct concentration for BTEX. For oxygen, define an ambient concentration of 9.0 mg/L.

1. Select the *Oxygen* species in the *Species* list.
2. Enter a value of “9.0” (mg/l) in the *Starting Conc.* column.

Porosity

Next, porosity must be considered. The aquifer is a uniform horizontal aquifer with a top elevation of 10 m, a thickness of 10 m, and a porosity of 0.3. Since 0.3 is the default porosity assigned by GMS, no changes need to be made.

Stress Periods

Next to define the stress periods. Since the injection rate and the boundary conditions do not change, set a single stress period with a length of 730 days (two years).

1. Select **Stress Periods...** to open the *Stress Periods* dialog.

Since there is already one stress period by default, simply change the length of the stress period

2. Enter a value of “730” for the *Length*.
3. Enter a value of “10” for the *Num Time Steps*.
4. Select **OK** to exit the *Stress Periods* dialog.

Output Options

Finally, define the output options. One binary solution file is created by RT3D for each of the species. By default, RT3D saves a solution at each transport step for each species. Since this results in large files containing more solutions than are needed for the simple post-processing intended here, simply specify that a solution be saved every 73 days (every time step).

1. Select **Output Control...** to open the *Output Control* dialog.
2. Select the *Print or save at specified times* option.
3. Select **Times...** to open the *Variable Time Steps* dialog.

4. Select **Initialize Values...** to open the *Initialize Time Steps* dialog.
5. Enter “73.0” (d) for the *Initial time step size*.
6. Enter “73.0” (d) for the *Maximum time step size*.
7. Enter “730.0” (d) for the *Maximum simulation time*.
8. Select **OK** to exit the *Initialize Time Steps* dialog
9. Select **OK** to exit the *Variable Time Steps* dialog.
10. Select **OK** to exit the *Output Control* dialog.

This completes the input for the Basic Transport package.

11. Select **OK** to exit the *Basic Transport Package* dialog.


3.3 Assigning Concentrations to the Left Boundary

The left boundary of the model is a constant head boundary. Since the head at the left boundary is greater than the head at the right boundary, the left boundary acts as a source and water enters the model from the left. Thus, the concentrations of the species at the left boundary must be defined. Otherwise, the water entering the model will have the default concentration (zero) for each of the species. This is often true for most species, but there are exceptions. For this model, the incoming fluid should have an ambient oxygen concentration of 9.0 mg/L. The oxygen concentration can be assigned one of two ways:

1. The cells at the boundary can be marked as specified concentration cells by changing the ICBUND value at the cells to -1.
2. The concentration of the incoming fluid at the boundary can be defined using the Source/Sink Mixing package.

The first method requires the fewest number of steps to set up. As long as the boundary acts as a source throughout the simulation, either of these approaches will achieve the same result. However, if the boundary acts as a sink during any part of the simulation, the second option should be used. With the second option, if the boundary switches to a sink due to changing heads in the interior of the model, the concentrations at the cells would be computed by RT3D. With the first option, the concentrations at the cells are fixed, regardless of the flow direction.

For this problem, the flow field is steady state and the boundary acts as a source. Thus, the first option is the most appropriate here. To mark the cells as specified concentration cells, do the following:

3. Use the **Select Cells**  tool to select the cells on the left boundary by dragging a box that just surrounds the cells.

4. Select the *RT3D | Cell Properties...* command to open the *3D Grid Cell Properties* dialog.
5. Change the *ICBUND* value to “-1”.
6. Select **OK** to close the *3D Grid Cell Properties* dialog.

Note that the starting concentration values do not need to be changed since the values are already assigned via the starting concentration arrays.

3.4 The Advection Package

The next step is to initialize the data for the Advection package. This tutorial will be using the MOC method.

1. Select the *RT3D | Advection Package...* command to open the *Advection Package* dialog.
2. In the *Solution scheme* drop-down menu, select the “Method of characteristics (MOC)” method.
3. Select **OK** to close the *Advection Package* dialog.


3.5 The Dispersion Package

Next, enter the data for the Dispersion package. The aquifer has a longitudinal dispersivity of 10.0 m and a transverse dispersivity of 3.0 m. The vertical dispersivity is assumed equal to the longitudinal dispersivity.

1. Select the *RT3D | Dispersion Package...* command to open the *Dispersion Package* dialog.
2. Select **Longitudinal Dispersivity...** to open the *Longitudinal Dispersivity* dialog.
3. Select **Constant** → **Grid...** to open the *Grid Value* dialog.
4. Enter a value of “10.0”.
5. Select **OK** to close the *Grid Value* dialog.
6. Select **OK** to exit the *Longitudinal Dispersivity* dialog.
7. Enter a value of “0.3” for the *TRPT* value.
8. Select **OK** to exit the *Dispersion Package* dialog.

3.6 The Source/Sink Mixing Package

Next to define the concentration at the spill location. To define the source at the spill location, select the well and assign a concentration.

1. Use the **Select Cells**  tool to select the cell containing the injection well (spill location) by clicking anywhere in the interior of the cell.
2. Select the *RT3D | Point Sources/Sinks...* command to open the *MODFLOW/RT3D Sources/Sinks* dialog.
3. Click the **Add BC** button near the bottom of the dialog.
4. In the *All* column, change the *Type (ITYPE)* to “well (WEL)”.
5. In the *BTEX* column, enter a concentration of “1000” (mg/L).
6. Leave the oxygen at the default value of “0.0”.
7. Select **OK** to exit the *MODFLOW/RT3D Sources/Sinks* dialog.

3.7 The Chemical Reactions Package

The final step is to initialize the data for the Chemical Reactions package. In this case, the default values are adequate, so no changes are necessary.

3.8 Running RT3D

It is now possible to save the model and run RT3D. Before running RT3D, save the current project under a different name than the MODFLOW model. Because this is done, it is necessary to explicitly tell RT3D the name of the MODFLOW model.

1. Select the *RT3D | Run Options...* command to open the *Run Options* dialog.
2. Select the *Single run with selected MODFLOW solution* option. Confirm that the “flowmod (MODFLOW)” solution is selected.
3. Click **OK** to close the *Run Options* dialog.
4. Select the *File | Save As...* command to open the *Save As* dialog.
5. Check to make sure the directory is the same one the model was saved in initially.
6. Enter “btexmod” for the *File name*.
7. Select **Save** to save the new project file and close the *Save As* dialog.



To run RT3D:

8. Select the *RT3D* / **Run RT3D...** command to launch the *RT3D* model wrapper.
9. When the simulation is finished, select **Close** to exit the *RT3D* model wrapper.

GMS will read in the solution automatically.

3.9 Viewing the Results


The visible contours are for BTEX at the end of the first time step (73.0 days). To view the BTEX solution at 730 days, do the following:

1. Expand the “ btexmod (RT3D)” solution in the Project Explorer.
2. Select the “ BTEX” dataset.


The *Time Step Window* should now appear below the Project Explorer. The *Time Step Window* is set to appear when a transient dataset is selected in the Project Explorer.

3. Select the 730.0 time step in the *Time Step Window*.

To get a better plot of the concentration contours:

4. Select **Contour Options**  from the main toolbar to open the *Dataset Contour Options – 3D Grid – BTEX* dialog.
5. Select the “Color Fill” option from the first *Contour method* drop-down.
6. Select **OK** to exit the *Dataset Contour Options – 3D Grid – BTEX* dialog.

Next, view the Oxygen solution at 730 days.

7. Select the “ Oxygen” dataset in the Project Explorer.

Feel free to use the Project Explorer and *Time Step Window* to view the solution at other time steps.

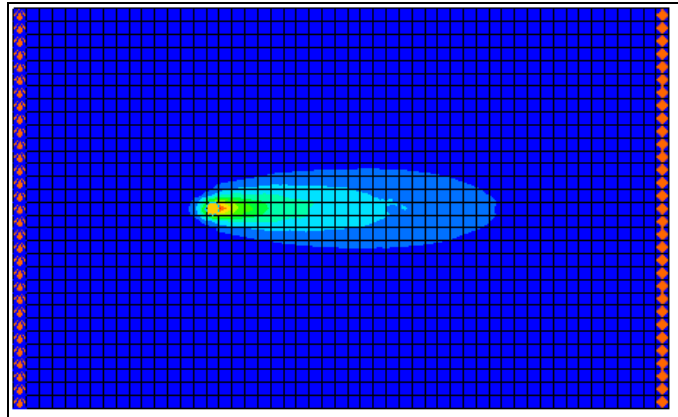






Figure 4 The BTEX solution set at 730 days

3.10 Viewing a Film Loop Animation

The most effective post-processing tool for transient data is to set up a film loop animation showing the fate of the BTEX through the entire duration of the simulation.

1. Select the “ BTEX” dataset in the Project Explorer.

To set up the animation:

2. Select the *Display / Animate...* command to open the *Animation Wizard – Options* dialog.
3. Make sure the *Dataset* option is turned on then click **Next >** to go to the *Animation Wizard – Datasets* page of the dialog.
4. Turn on the *Display clock* option.
5. Select the **Finish** button to generate the animation and close the *Animation Wizard* dialog. The AVI application will launch.
6. After viewing the animation, select the **Stop**  button to stop the animation.
7. Select the **Step**  button to move the animation one frame at a time.
8. Feel free to experiment with some of the other playback controls. When finished, close  the window and return to GMS.

4 Conclusion

This concludes the tutorial. Continue to explore using RT3D or close the program.