

# GMS 7.0 TUTORIALS

---

## ***RT3D – Rate-Limited Sorption Reaction***

This tutorial illustrates the steps involved in using GMS and RT3D to model sorption reactions under mass-transfer limited conditions. The flow model used in this simulation is different from the one used in other example problems. Hence, the steps involved in creating input data for both the flow and the reactive transport models will be described in this tutorial exercise.

### **1.1 Contents**

---

1.1	Contents .....	2-1
1.2	Outline.....	2-2
1.3	Required Modules/Interfaces .....	2-2
<b>2</b>	<b>Description of the Reaction Model .....</b>	<b>2-2</b>
<b>3</b>	<b>Description of Problem.....</b>	<b>3-4</b>
<b>4</b>	<b>Getting Started .....</b>	<b>4-5</b>
<b>5</b>	<b>Building the Flow Model .....</b>	<b>5-5</b>
5.1	Reading in the Map File .....	5-5
5.2	Units .....	5-6
5.3	Creating the Grid.....	5-6
5.4	Initializing the MODFLOW Data .....	5-7
5.5	The Global Package .....	5-7
5.6	Specified Head Boundaries .....	5-8
5.7	The LPF Package .....	5-8
5.8	Creating the Wells.....	5-9
5.9	Saving and Running the Flow Model.....	5-10
<b>6</b>	<b>Building the Transport Model .....</b>	<b>6-10</b>
6.1	Initializing the Model.....	6-10
6.2	The Basic Transport Package.....	6-11
6.3	The Advection Package.....	6-13
6.4	The Dispersion Package.....	6-14
6.5	The Source/Sink Mixing Package .....	6-14
6.6	The Reaction Package .....	6-15
6.7	Saving and Running the Simulation.....	6-15

6.8	Viewing the Solution .....	6-15
<b>7</b>	<b>Comparison to Other Solutions .....</b>	<b>7-16</b>
7.1	Importing the Solutions.....	7-16
7.2	MT3DMS Solution .....	7-16
7.3	Comparing the Solid Phase Concentrations .....	7-17

## 1.2 Outline

---

This is what you will do:

1. Open a map file.
2. Create a 3D grid.
3. Define conditions.
4. Run MODFLOW.
5. Define conditions for the transport model.
6. Run RT3D.
7. Import other solutions and compare them.

## 1.3 Required Modules/Interfaces

---

You will need the following components enabled to complete this tutorial:

- Grid
- Map
- MODFLOW
- RT3D

You can see if these components are enabled by selecting the *Help | Register* command.

## 2 Description of the Reaction Model

---

The fate and transport of an organic pollutant in subsurface environments is often highly dependent on its sorption characteristics. Under most natural groundwater flow conditions, the partitioning of contaminants between the solid and aqueous phases can be assumed to be at a local equilibrium. Thus, the more widely used retardation approach for modeling sorption may provide an adequate description for the overall transport. However, when external pumping and injection stresses are imposed on an aquifer (e.g. using a pump-treat-system), the equilibrium assumption may fail. This would lead to some well-known conditions such as the plume tailing effect (i.e. low contaminant levels always observed at the extraction well) and/or the rebounding effect (i.e. aquifer seems to

be clean but the aqueous concentrations start to increase immediately after stopping the treatment system). These conditions cannot be simulated using the standard linear retardation approach since they require a mass-transfer description for the sorption reactions.

In the mass-transfer limited sorption model, the exchange of contaminants between the soil and groundwater is assumed to be rate limited. The rate of exchange is dictated by the value of the mass-transfer coefficient. When the mass-transfer rate is high (relative to the overall transport), the rate-limited model relaxes to the retardation model. On the other end of the spectrum, a very low mass-transfer coefficient would mimic fully sequestered conditions where the contaminants in the soil phase are assumed to be irreversibly adsorbed and trapped into the soil pores. Under this extreme condition, it might be possible to simply clean the groundwater plume and leave the sequestered soil contaminant in the aquifer because the sorbed contaminants may not pose any potential risk to the environment. In either of the extreme conditions, pump-and-treat is the best option to remediate the groundwater plume. Unfortunately, in most instances, the mass-transfer coefficient is expected to lie in an intermediate range, causing the well-known limitations to the pump-and-treat system.

When sorption is assumed to be rate-limited, it is necessary to track contaminant concentrations in both mobile (groundwater) and immobile (soil) phases. Following Haggerty and Gorelick's approach (Water Resource Research, 30(2), 435-446, 1994), the fate and transport of a sorbing solute at aqueous and soil phases can be predicted using the following transport equations:

$$\frac{\partial C}{\partial t} + \frac{\rho}{\phi} \frac{\partial \tilde{C}}{\partial t} = \frac{\partial}{\partial x_i} \left( D_{ij} \frac{\partial C}{\partial x_j} \right) - \frac{\partial}{\partial x_i} (v_i C) + \frac{q_s}{\phi} C_s \dots\dots\dots (1)$$

$$\frac{\rho}{\phi} \frac{\partial \tilde{C}}{\partial t} = \xi \left( C - \frac{\tilde{C}}{\lambda} \right) \dots\dots\dots (2)$$

where  $C$  is the concentration of the contaminant in the mobile-phase [ $ML^{-3}$ ],  $\tilde{C}$  is the concentration of the contaminant in the immobile phase (mass of the contaminants per unit mass of porous media, [ $MM^{-1}$ ]),  $\rho$  is the bulk density of the soil matrix,  $\phi$  is the soil porosity,  $\xi$  is the first-order, mass-transfer rate parameter [ $T^{-1}$ ], and  $\lambda$  is the linear partitioning coefficient (which is equal to the linear, first-order sorption constant  $K_d$ ) [ $L^3M^{-1}$ ]. It can be mathematically shown that the above model formulation relaxes to the retardation model when the value of  $\xi$  becomes high (see Clement et al. 1998 paper in the spring issue of Groundwater Monitoring and Remediation journal).

The mass-transfer model discussed above has been implemented as an RT3D reaction package (one mobile species and one immobile species). After employing reaction-operator splitting, the reaction package for the problem reduces to:

$$\frac{dC}{dt} = -\xi \left( C - \frac{\tilde{C}}{\lambda} \right) \dots\dots\dots (3)$$

$$\frac{d\tilde{C}}{dt} = \frac{\phi\xi}{\rho} \left( C - \frac{\tilde{C}}{\lambda} \right) \dots\dots\dots (4)$$

These two differential equations are coded into the model #4 designated as the rate-limited sorption reaction module.

### 3 Description of Problem

The example problem we will be solving in this tutorial is shown in Figure 1. The site is a 304 m x 152 m section of an unconfined aquifer with flow gradient from left to right. A spill at the center of the site has created a contaminant plume as shown in the figure. A pump-and-treat system, using three injection and extraction wells at the constant rate of 115 m<sup>3</sup>/day, will be used to clean the contaminant plume. The aqueous concentration of contaminant level throughout the plume is assumed to be at 300 mg/L. The linear partitioning coefficient ( $K_d$  or  $\lambda$ ) for the contaminant is assumed to be  $1.0 \times 10^{-7}$  (L/mg), soil dry bulk density,  $\rho$ , is assumed to be  $1.5 \times 10^6$  (mg/L), and porosity is assumed to be 0.3. Note these parameters yield an effective retardation coefficient value of 1.5 ( $R = 1 + \rho\lambda / \phi$ ). Assuming equilibrium conditions exist before starting the pump-and-treat system, the initial soil-phase contaminant concentration levels,  $\tilde{C} = \lambda C$ , can be estimated to be at  $3 \times 10^{-5}$  (mg of contaminant / mg of soil). The objective of the treatment system is to clean both dissolved and soil-phase contamination. The model will simulate the effectiveness of the system under different mass transfer conditions. A 3000 day simulation will be performed. The mass-transfer coefficient values will be varied to simulate retardation conditions (using  $\xi = 0.1 \text{ day}^{-1}$ ), intermediate conditions (using  $\xi = 0.002 \text{ day}^{-1}$ ) and sequestered conditions (using  $0.0001 \text{ day}^{-1}$ ). Time series plots and contour plots will be used to visualize the treatment scenarios under different field conditions.

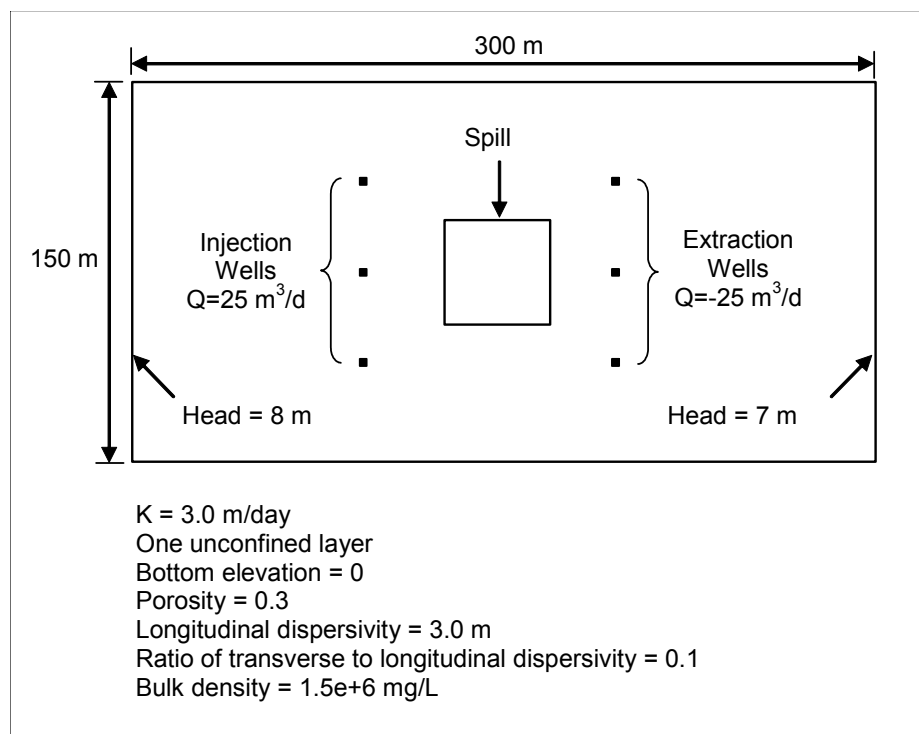


Figure 1. Sample Problem.

## 4 Getting Started

If you have not yet done so, launch GMS. If you have already been using GMS, you may wish to select the *New* command from the *File* menu to ensure the program settings are restored to the default state.

## 5 Building the Flow Model

The first step in setting up the problem is to build the MODFLOW flow model. The model will be a steady state one layer unconfined model with 6.1 m x 6.1 m cells. The flow solution will then be used to drive the transport model.

### 5.1 Reading in the Map File

Before creating the flow model, we will first read in a map file that contains some drawing objects for background display that will aid in building the model.

1. Select the *File | Open* command.
2. In the *Open* dialog, locate and open the file entitled `tutfiles\RT3D\rlimsorp\site.gpr`.

The blue rectangle is the model boundary and the red rectangle is the spill location.

## 5.2 Units

To define the units:

1. Select the *Edit | Units* command.
2. For the *Length* units, select **m**.
3. For the *Time* units, select **d**.
4. For the *Mass* units, select **mg**.
5. For the *Concentration* units, select **mg/l**.
6. Select *OK*.

## 5.3 Creating the Grid

To create the grid:

1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New | 3D Grid* command.
2. In the *Create Finite Difference Grid* dialog, change the values to match the following figure.

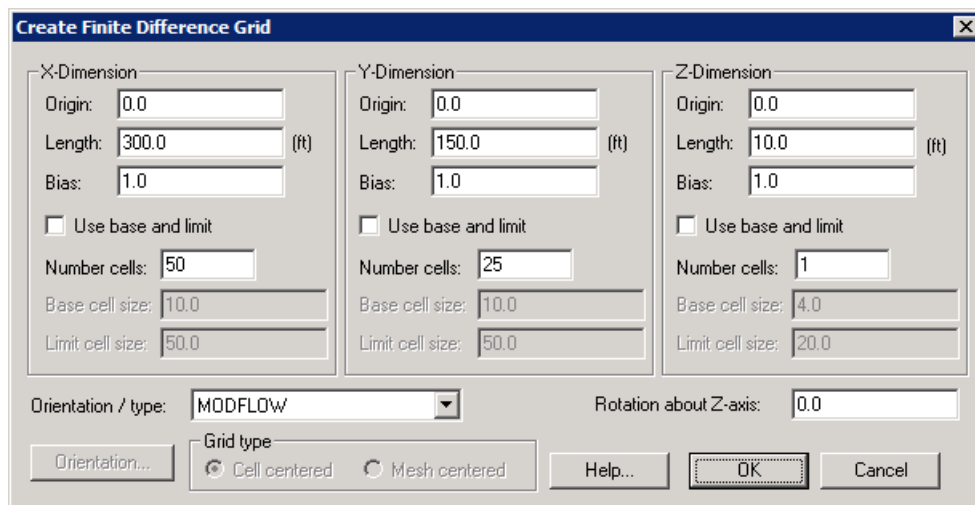


Figure 2. Creating the Grid.

3. Select the *OK* button.


A grid will appear.

---

## 5.4 Initializing the MODFLOW Data

---

To initialize the MODFLOW data:

1. In the *Project Explorer* right-click on the 3D grid  and, from the pop-up menu, select the *New MODFLOW* command.

This brings up the *MODFLOW Global/Basic Package* dialog.

---

## 5.5 The Global Package

---

Next, we will define the data in the Global package.

### IBOUND

The IBOUND array is used to designate the constant head boundaries. However, we will mark the boundaries later in the tutorial by directly selecting the cells.

### Starting Head

We will assign a starting head of 10 m everywhere in the grid.

1. Select the *Starting Heads* button.
2. Select the *Constant* → *Layer* button.
3. Enter a value of **10** and click *OK*.
4. Click *OK* to return to the *Global Package* dialog.

### Top Elevation

The top elevation is a constant value of 10 m throughout the grid.

1. Select the *Top Elevation* button.
2. Select the *Constant* → *Layer* button.
3. Enter a value of **10** and click *OK*.
4. Click *OK* to return to the *Global Package* dialog.


### Bottom Elevation

The bottom elevation is a constant value of zero throughout the grid. Since zero is the default value, no changes need to be made.

1. Select the *OK* button to exit the *Global Package* dialog.

## 5.6 Specified Head Boundaries

Next, we will define the specified head boundaries.

1. Choose the *Select J* tool .
2. Right-click on any of the cells in the leftmost column of the grid and then, from the pop-up menu, select the *Properties...* command.

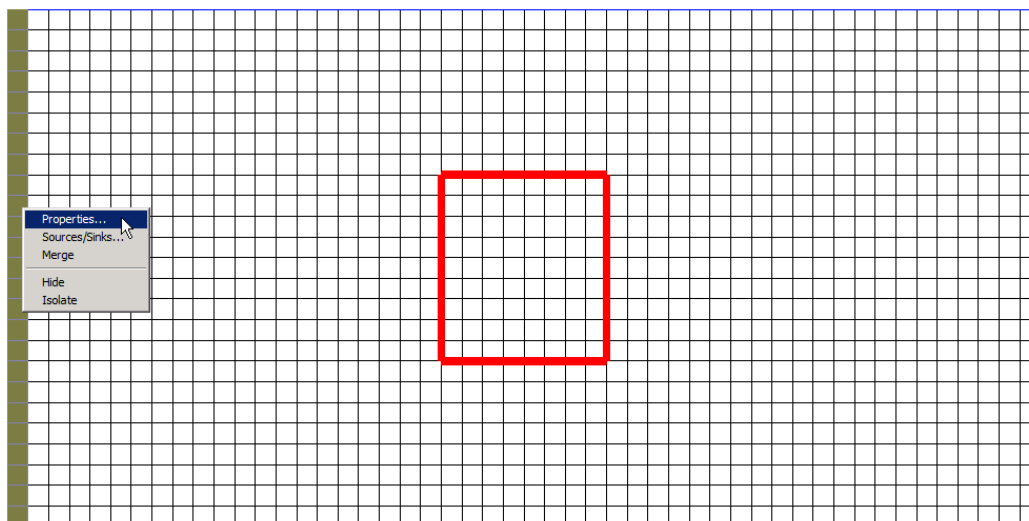




Figure 3. Changing the Properties of the Cells on the Left of the Grid.

3. In the *IBOUND* row, switch the option to **Specified Head** in the pull-down list and change the *Starting Head* value to **8**.
4. Select the *OK* button.
5. Right-click on any of the cells in the rightmost column of the grid and then, from the pop-up menu, select the *Properties...* command.
6. In the *IBOUND* row, switch the option to **Specified Head** in the pull-down list and change the *Starting Head* value to **7**.
7. Select the *OK* button.
8. Click anywhere outside the grid to unselect the cells.

## 5.7 The LPF Package

Next, we will define the input for the LPF package. We will enter a hydraulic conductivity that is constant throughout the grid.


1. In the *Project Explorer* expand the *MODFLOW* item  and right-click on the *LPF Package*  and select *Properties*.



2. Select the *Horizontal Hydraulic Conductivity* button.
3. Select the *Constant* → *Layer* button.
4. Enter a value of **3.0** and click *OK*.
5. Select the *OK* button to return to the *LPF Package* dialog.
6. Select the *OK* button to exit the *LPF Package* dialog.

## 5.8 Creating the Wells

Next, we will create the wells. To create the injection wells:

1. Choose the *Select Cell* tool .
2. While monitoring the *ijk* indices in the *Status Bar*, locate and select the cell at (I=7, J=16).
3. While holding down the Shift key, locate and select the cells at (I=13, J=16) and (I=19, J=16).

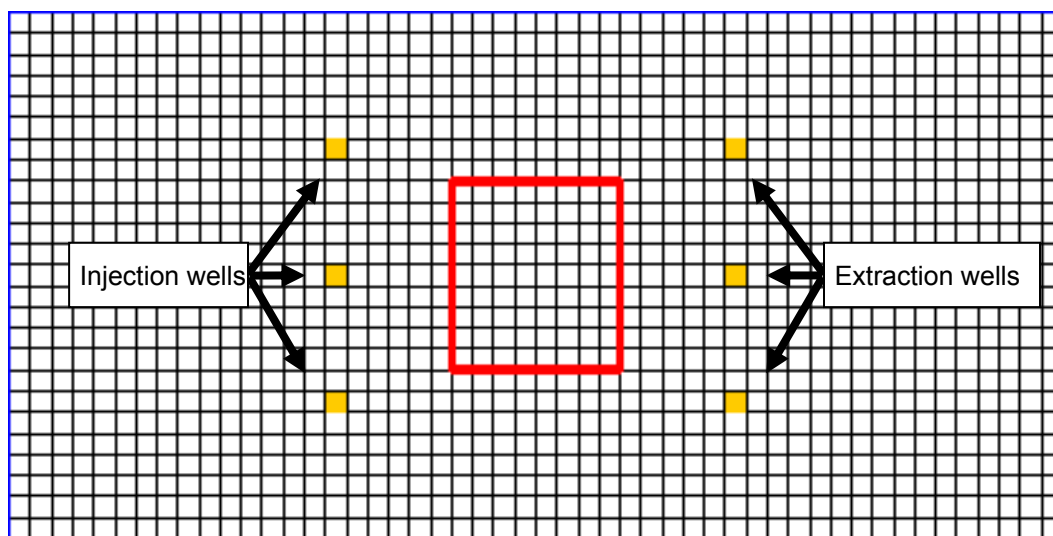


Figure 4. Location of Injection and Extraction Well Cells.

4. Select the *MODFLOW* | *Sources/Sinks...* command.
5. In the *Well* tab, select the *New* button.
6. Enter a value of **25** for the *Q (Flow)* of each well.
7. Select the *OK* button.
8. Click anywhere outside the grid to unselect the cells.

To create the extraction wells:

9. While holding down the Shift key, locate and select the cells at (I=7, J=35), (I=13, J=35) and (I=19, J=35).
10. Select the *MODFLOW* | *Sources/Sinks...* command.
11. In the *Well* tab, select the *New* button.
12. Enter a value of **-25.5** for the *Q (Flow)* for each well.
13. Select the *OK* button.
14. Click anywhere outside the grid to unselect the cells.

---

## 5.9 Saving and Running the Flow Model

At this point, we are ready to save the model and run MODFLOW.

1. Select the *File* | *Save As* command.
2. Make sure the path is still set to **tutfiles\RT3D\rlimsorp**.
3. Enter "**rlimsorp**" for the file name.
4. Select the *Save* button to save the files.

To run MODFLOW:

5. Select the *MODFLOW* | *Run MODFLOW* command.
6. When the simulation is finished, select the *Close* button.

GMS will automatically read in the solution. You should see a series of contours indicating a flow from left to right with mounds around the injection wells and cones of depression around the extraction wells.

---


## 6 Building the Transport Model

We are now ready to build the RT3D transport model. We will first build a model that simulates retardation conditions. We will then compare that solution to a solution from MT3DMS. Finally, we will compare the retardation solution to a solution representing intermediate and sequestered conditions.

---

### 6.1 Initializing the Model

To initialize the RT3D model:

1. In the *Project Explorer* right-click on the 3D grid  and, from the pop-up menu, select the *New MT3D* command.

## 6.2 The Basic Transport Package

---

First, we will define the data for the Basic Transport package.

1. In the *Model* section, select *RT3D*.

### Packages

Next, we will select which packages we will use:

1. Select the *Packages* button.
2. Turn on the following packages:
  - *Advection package*
  - *Dispersion package*
  - *Source/sink mixing package*
  - *Chemical reaction package*
3. In the *RT3D Reactions* section, select the reaction titled *Rate-Limited Sorption Reactions*.
4. Select the *OK* button.

### Stress Periods

Next, we will define a single stress period with a length of 3000 days.

1. Select the *Stress Periods* button.
2. Change the *Length* value to **3000.0**.
3. Select the *OK* button.

### Output Control

By default, RT3D outputs a solution at every transport step. We will change this so that a solution is output every 200 days.

1. Select the *Output Control* button.
2. Select the *Print or save at specified times* option.
3. Select the *Times* button.


4. Select the *Initialize Values* button.
5. Enter **200.0** for the *Initial time step size*.
6. Enter **200.0** for the *Maximum time step size*.
7. Enter **3000.0** for the *Maximum simulation time*.
8. Select the *OK* button to exit the *Initialize Time Steps* dialog.
9. Select the *OK* button to exit the *Variable Time Steps* dialog.
10. Select the *OK* button to exit the *Output Control* dialog.

## Porosity

Next, we will define the porosity as 0.3. Since this is the default supplied by GMS, no changes need to be made.

## Starting Concentrations

A starting concentration must be defined for both the aqueous phase concentration and the solid phase concentration. The default starting concentrations are zero. We need to change the starting concentrations at the plume location. While this can be accomplished with the Starting Concentration dialog, it is more convenient to select the cells and directly assign the values.

1. Select the *OK* button to exit the *Basic Transport Package* dialog.
2. Select the *Select Cell* tool 
3. Drag a box that just encloses the red rectangle defining the spill location.

Before assigning the values, we will unselect the cells in the four corners of the grid. This will give the plume a slightly more rounded shape.

4. While holding down the *Shift* key, select each of the cells in the four corners of the spill location to unselect them.

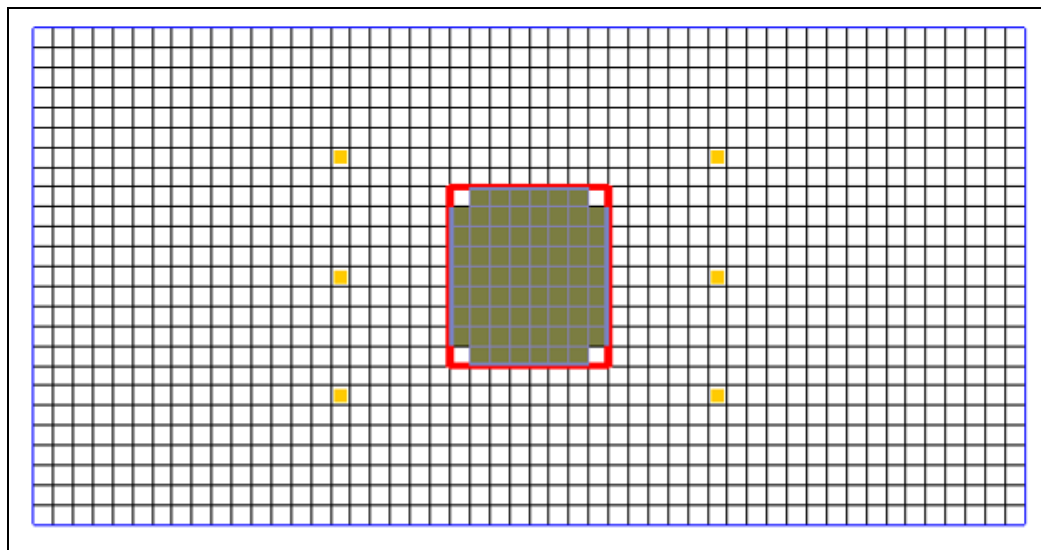


Figure 5. Cells where Starting Concentration will be Assigned.

5. Select the *MT3D | Cell Properties* command.
6. Enter a value of **300** for the starting concentration in the *Aqueous conc.* column.
7. Enter a value of **3e-5** for the starting concentration in the *Solid conc.* column.
8. Select the *OK* button.
9. Click anywhere outside the grid to unselect the cells.

This completes the input for the Basic Transport package.

### 6.3 The Advection Package

Next, we will define the input data for the Advection package.

1. Select the *MT3D | Advection Package* command.
2. For the *Solution scheme* select the **Modified method of characteristics (MMOC)**.
3. Select the *Particles* button.
4. At the top of the dialog, change the *Max. number of cells any particle will be allowed to move per transport step* value to **2**.
5. Select the *OK* button to exit the *Particles* dialog.
6. Select the *OK* button to exit the *Advection Package* dialog.

## 6.4 The Dispersion Package

Next, we will enter the data for the Dispersion package.

1. Select the *MT3D | Dispersion Package* command.
2. Enter a value of **0.1** for the *TRPT* value.
3. Select the *Longitudinal Dispersivity* button.
4. Select the *Constant* → *Layer* button.
5. Enter a value of **3.0**.
6. Select the *OK* button twice to return to the *Dispersion Package* dialog.
7. Select the *OK* button to exit the *Dispersion Package* dialog.

## 6.5 The Source/Sink Mixing Package

For the Source/Sink Mixing Package, we will assign a zero concentration to the incoming fluid from the injection wells.

1. While holding the *Shift* key, select each of the three injection wells (the wells on the left).
2. Select the *MT3D | Point Sources/Sinks* command.
3. Turn **on** the toggle in the *All* row and the column to the left of the *Well* column. This turns on all three wells.

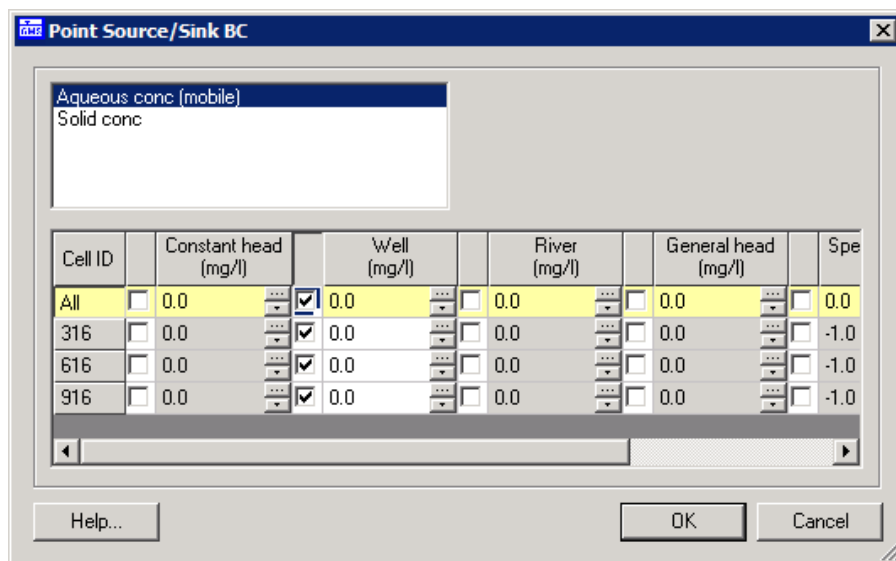


Figure 6. Point Source/Sink BC dialog.

4. Leave the concentration at **0.0** and select the *OK* button to exit.
5. Click anywhere outside the grid to unselect the cells.

## 6.6 The Reaction Package

---

The last step in setting up the transport model is to enter the data for the Reaction package.

1. Select the *MT3D | Chemical Reaction Package* command.
2. For the *Bulk Density* enter a value of **1.5e6**.
3. In the *Reaction Parameters* section, enter a value of **0.1** for *mass transfer coeff*.
4. Enter a value of **1e-7** for the *partitioning coeff*.
5. Select the *OK* button to exit.

## 6.7 Saving and Running the Simulation

---

At this point, we are ready to save the model and run RT3D.

1. Select the *File | Save* command.


To run RT3D:

2. Select the *MT3D | Run RT3D* command.
3. Select *OK* at the prompt.
4. When the simulation is finished, select the *Close* button.


## 6.8 Viewing the Solution

---

First, we will view the solid phase concentration solution at 600 days.


1. Expand the *rlmsorp (RT3D)* solution  in the *Project Explorer*.
2. Select the *Solid conc* data set to make it active.
3. Select the **600** time step from the *Time Step* window.

To better illustrate the variations, we will turn on the color ramp.

4. Select the *Contour Options*  button.
5. Change the *Contour method* to **Color Fill**.

6. In the *Data Range* section turn **on** the *Use each timestep's max and min*.
7. Select the *OK* button to exit the *Contour Options* dialog.

Next, we will view the aqueous phase concentration solution at 600 days.

8. Select the  *Aqueous conc* data set from the *Project Explorer*.

Notice that although the magnitudes of the concentration values are different, the spatial distribution of the plume is identical for the solid and aqueous phase.

## 7 Comparison to Other Solutions

Next, we will compare our initial solution to other solutions with different mass transfer coefficients. To save time, these solutions have already been computed. We simply need to read them into GMS.

### 7.1 Importing the Solutions

We will read in the previously computed solutions. Although the solutions were originally created as separate solutions, the solutions have been combined into a single solution set for convenience.

1. Select the *MT3D | Read Solution* command.
2. Select the file entitled **othersol.rts**.
3. Select the *OK* button.

The solution just imported contains the following data sets:

Name	Description
Aqueous (mt3d)	Solution from an MT3DMS simulation
Solid (interm)	Solution from an RT3D simulation with the mass transfer coeff. = 0.002. This represents an intermediate condition between the retardation condition and the sequestered condition.
Solid (sequest)	Solution from an RT3D simulation with the mass transfer coeff. = 0.0001. This represents the sequestered condition.

### 7.2 MT3DMS Solution

First, we will examine the MT3DMS simulation. The solution we have computed has a large mass transfer rate and simulates retardation conditions. Therefore, it should be very similar to a solution computed using MT3DMS. To confirm this:

1. Using the *Time Step* window, select the time step at  $t = 600$  days.


Note that the spatial distribution of the plume appears to be identical to the plume computed earlier by RT3D.



### 7.3 Comparing the Solid Phase Concentrations


---

Next, we will look at the solid phase concentrations and analyze the effect of the mass transfer coefficients.

1. If necessary, expand the  *rlimsorp (RT3D)* solution in the *Project Explorer*.
2. Select the *Solid conc* data set.

Notice that after 600 days the bulk of the sorbed plume has moved over to the vicinity of the extraction wells.

Next, we will look at the intermediate solution. This solution was computed using a mass transfer coefficient of 0.002. This is part way between the retarded condition and the sequestered condition.

3. Using the *Project Explorer*, expand the *othersol (RT3D)* solution .
4. Select the *Solid (interm)* data set.

Notice that some of the sorbed plume has moved toward the extraction wells but much of the plume is still in the original location.

Next, we will examine the sequestered solution. This solution was computed using a mass transfer coefficient of 0.0001.

5. Using the *Project Explorer*, select the *Solid (Sequest)* data set .

Notice that the sorbed contaminants are still in the original location.

This concludes the tutorial.