Objectives
Learn how to use a conceptual model when using MT3DMS. Perform two transport simulations, analyzing the long-term potential for migration of leachate from a landfill.

Prerequisite Tutorials
- MODFLOW – Conceptual Model Approach I
- MT2DMS – Grid Approach

Required Components
- Map Module
- Grid Module
- MODFLOW
- MT3DMS

Time
- 30–50 minutes
1 Introduction

MT3DMS simulations can be constructed using the grid approach, where data are entered on a cell-by-cell basis. They can also be constructed using the conceptual model approach, where the data are entered via points, arcs, and polygons. This tutorial describes how to use the conceptual model approach.

The problem for this tutorial is an extension of the problem described in the tutorial entitled “MODFLOW – Conceptual Model Approach I”. It is recommended that tutorial be completed before continuing.

In that tutorial, a site in East Texas was modeled. This tutorial uses the solution from that model as the flow field for the transport simulation. The model included a proposed landfill. For this tutorial, two transport simulations will be run to analyze the long term potential for migration of leachate from the landfill. The first simulation will be
modeling transport due to advection and dispersion only. The second simulation will include sorption and decay in addition to advection.

This tutorial will discuss and demonstrate opening a MODFLOW model and solution, defining conditions for an MT3DMS simulation, converting the conceptual model to MT3DMS, running MODFLOW, and running MT3DMS. It will then demonstrate creating an animation, defining additional parameters and rerunning MT3DMS, and creating a time series plot.

2 Getting Started

Do the following to get started:

1. If necessary, launch GMS.
2. If GMS is already running, select *File | New* to ensure that the program settings are restored to their default state.

2.1 Importing the Project

The first step is to import the East Texas project, which includes the MODFLOW model and solution and all other files associated with the model.

1. Click *Open* to bring up the *Open* dialog.
2. Select “Project Files (*.gpr)” from the *Files of type* drop-down.
3. Browse to the \Tutorials\MODFLOW\modfmap\sample2 directory and select “modfmap2.gpr”.
4. Click *Open* to import the project and exit the *Open* dialog.

The imported model should appear similar to Figure 1.

![Initial model after importing](image)
2.2 Defining the Units

Next, define the units for mass and concentration. Do not change the length and time units as these must be consistent with the flow model.

1. Select Edit | Units… to open the Units dialog.
2. Select “kg” from the Mass drop-down.
3. Select “ppm” from the Concentration drop-down.
4. Click OK to close the Units dialog.

3 Initializing the MT3DMS Simulation

Now that the MODFLOW model imported, initialize the MT3DMS simulation.

1. Fully expand the “3D Grid Data” folder.

2. Right-click on “grid” and select New MT3DMS… to open the Basic Transport Package dialog.

3.1 Defining the Species

MT3DMS is a multi-species model, so define the number of species and name each one. In this case, there is only one species.

1. Click Define Species… to open the Define Species dialog.
2. Click New to add a new species to the spreadsheet.
3. Enter “leachate” in the Name column.
4. Click OK to close the Define Species dialog.

3.2 Defining the Stress Periods

Next, define the stress periods.

1. Click Stress Periods… to open the Stress Periods dialog.

In this case, the flow solution computed by MODFLOW is steady state. Therefore, any desired sequence of stress periods and time steps could be defined. For this tutorial, only one stress period is needed because the leachate from the landfill will be released at a constant rate. Enter the length of the stress period (i.e., the length of the simulation) and let MT3DMS compute the appropriate transport time step length by leaving the transport step size at zero.

2. On row 1, enter “3000.0” in the Length column.
3. Scroll to the right and enter “4000” in the Max trans. steps column.

4. Click OK to exit the Stress Periods dialog.

### 3.3 Setting the Output Control

By default, MT3DMS outputs a solution at every transport step. This results in a large output file, so change the output to write a solution every time step (every 300 days).

1. Click Output Control… to open the Output Control dialog.
2. Below the Retardation factor section, select Print or save at specified times.
3. Click Times… to open the Variable Time Steps dialog.
4. Click Initialize Values… to open the Initialize Time Steps dialog.
5. Enter “300.0” as the Initial time step size.
6. Enter “1.0” as the Bias.
7. Enter “300.0” as the Maximum time step size.
8. Enter “3000.0” as the Maximum simulation time.
9. Click OK to close the Initialize Time Steps dialog.
10. Click OK to close the Variable Time Steps dialog.
11. Click OK to close the Output Control dialog.

### 3.4 Selecting the Packages

Next, specify which of the MT3DMS packages to use.

1. Click Packages… to open the MT3DMS/RT3D Packages dialog.
2. Turn on Advection package, Dispersion package, Source/sink mixing package, and Transport observation package.
3. Click OK to close the MT3DMS/RT3D Packages dialog.

Note that the Basic Transport Package dialog also includes some layer data. This tutorial will address the data for these arrays at a latter point.

4. Click OK to exit the Basic Transport Package dialog.

### 4 Assigning the Aquifer Properties

MT3DMS requires that a porosity and dispersion coefficient be defined for each of the cells in the grid. While these values can be assigned directly to the cells, it is sometimes
convenient to assign the parameters using polygonal zones defined in the conceptual model. The parameters are converted to the grid cells using the \texttt{Map \rightarrow MT3DMS} command.

### 4.1 Turning on Transport

Do the following to assign the porosities and dispersion coefficients to the polygons:

1. Fully expand the \texttt{Map Data} folder.
2. Right-click on the \texttt{East Texas} conceptual model and select \texttt{Properties…} to open the \texttt{Conceptual Model Properties} dialog.
3. In the spreadsheet, check the box to the right of \texttt{Transport}.
4. Select \texttt{“MT3DMS”} from the \texttt{Transport model} drop-down.
5. Click \texttt{Define Species…} on the \texttt{Species} row to open the \texttt{Define Species} dialog.
6. Click \texttt{New} to create a new species.
7. Enter “leachate” in the \texttt{Name} column.
8. Click \texttt{OK} to close the \texttt{Define Species} dialog.
9. Click \texttt{OK} to exit the \texttt{Conceptual Model Properties} dialog.
10. Right-click on \texttt{“Layer 1”} and select \texttt{Coverage Setup…} to open the \texttt{Coverage Setup} dialog.
11. In the \texttt{Areal Properties} section, turn on \texttt{Porosity} and \texttt{Long. dispersivity}.
12. Click \texttt{OK} to close the \texttt{Coverage Setup} dialog.
13. Repeat steps 10–12 for the \texttt{“Layer 2”} coverage.

### 4.2 Assigning the Parameters to the Polygons

To assign the porosities and dispersion coefficients to the polygons:

1. Select \texttt{“Layer 1”} to make it active.
2. Using the \texttt{Select Polygons} tool, double-click on the layer polygon to open the \texttt{Attribute Table} dialog.
3. On row 1, enter “0.3” in the \texttt{Porosity} column.
4. Enter “20.0” in the \texttt{Long. disp.} column.
5. Click \texttt{OK} to close the \texttt{Attribute Table} dialog.
6. Select \texttt{“Layer 2”} to make it active.
7. Repeat steps 2-5, but enter “0.2” in the Porosity column.

8. Click anywhere outside the model to deselect the highlighted polygon.

### 4.3 Assigning the Recharge Concentration

The purpose of this model is to simulate the transport of contaminants emitted from the landfill. When the flow model was constructed, a separate, reduced value of recharge was assigned to the landfill site. This recharge represents leachate from the landfill, and a concentration needs to be assigned to this recharge. The concentration can be assigned directly to the recharge polygon in the conceptual model.

1. Right-click on the “Recharge” coverage and select **Coverage Setup…** to open the **Coverage Setup** dialog.

2. In the Areal Properties column, turn on **Recharge conc.**

3. Click **OK** to close the **Coverage Setup** dialog.

4. Select “Recharge” to make it active.

5. Using the **Select Polygons** tool, double-click on the landfill polygon to open the **Attribute Table** dialog.

6. On row 1, enter “20000.0” in the **leachate Recharge conc.** column.

7. Click **OK** to close the **Attribute Table** dialog.

8. Click anywhere outside the model to deselect the polygon.

### 5 Converting the Conceptual Model

At this point, it is possible to assign the aquifer parameters and the recharge concentration to the cells using the conceptual model.

1. Select **Feature Objects | Map → MT3DMS** to bring up the **Map → Model** dialog.

2. Select **All applicable coverages** and click **OK** to close the **Map → Model** dialog.
6 Options for the Conceptual Model

6.1 Layer Thicknesses

MT3DMS requires an HTOP array defining the top elevations of the uppermost aquifer to define the aquifer geometry. A thickness array must then be entered for each layer. With the layer geometry defined in the MODFLOW model, no further input is necessary.

6.2 The Advection Package

This tutorial uses the *Third Order TVD scheme (ULTIMATE)* solver for the Advection package. This is the default, so nothing needs to be done.

6.3 The Dispersion Package

The longitudinal dispersivity values were automatically assigned from the conceptual model. Only the remaining three parameters need to be entered.

1. Select MT3DMS | Dispersion Package… to open the Dispersion Package dialog.
2. On both rows 1 and 2, enter “0.2” in the TRPT column.
3. On both rows 1 and 2, enter “0.1” in the TRVT column.
4. On both rows 1 and 2, enter “0.0” in the DMCOEF column.
5. Click OK to exit the Dispersion Package dialog.

6.4 The Transport Observation Package

This package determines the mass flux into the river source/sink.

1. Select MT3DMS | Transport Observation Package… to open the Transport Observation Package dialog.
2. Turn off Compute concentrations at observation points.
3. Turn on Compute mass flux at source/sinks.
4. Click OK to exit the Transport Observation Package dialog.

6.5 The Source/Sink Mixing Package Dialog

The only data for the source/sink mixing package required in this package for this simulation are the concentrations assigned to the recharge from the landfill. These values
were automatically assigned to the appropriate cells from the conceptual model, so the input data for this package are complete.

7 Saving and Running MODFLOW and MT3DMS

Now that the MT3DMS data has been completely entered, save the model before running the simulations.

1. Select File | Save As… to bring up the Save As dialog.
2. Select “Project Files (*.gpr)” from the Save as type drop-down.
3. Browse to the \Tutorials\MT3D\mt3dmap directory.
4. Enter “run1.gpr” as the File name.
5. Click Save to save it under the new name and close the Save As dialog.

7.1 Running MODFLOW

MT3D requires the HFF file generated by MODFLOW. Because the project was saved in a different folder than the one where the MODFLOW simulation was opened, the HFF file does not exist in the new location. MODFLOW must be rerun to create the HFF file in the current folder.

To run MODFLOW, do the following:

1. Click Run MODFLOW to bring up the MODFLOW model wrapper dialog.
2. When the simulation finishes, turn on Read solution on exit and Turn on contours (if not on already).
3. Click Close to import the solution and close the MODFLOW dialog.
4. Save the project.

7.2 Running MT3DMS

Now run MT3DMS:

1. Click Run MT3DMS to bring up the MT3DMS model wrapper dialog.
2. When the simulation is finished, turn on Read solution on exit.
3. Click Close to import the solution and close the MT3DMS dialog.
4. Save the project.
### 8 Viewing the Solution

Now view the results of the MT3DMS model run.

1. Expand “run1 (MT3DMS)” and select the “leachate” dataset.
2. Select time step 10 (3000.0) in the Time Step Window.

It is often helpful to use the color-filled contours option. To do this, follow these steps:

3. Click **Contour Options** to open the Dataset Contour Options – 3D Grid – leachate dialog.
4. In the Data range section, turn on Specify a range.
5. Enter “1.0” for Min.
6. Enter “130.0” for Max.
7. In the Contour method section, select “Color fill” from the first drop-down.
8. Click **OK** to exit the Dataset Contour Options – 3D Grid – leachate dialog.

A display of color-shaded contours confined to the area adjacent to the landfill should be visible (Figure 2).

![Figure 2](image.png) **Figure 2** Leachate dispersion from the landfill at time step 10 (3000.0)

Do the following to view the solution for layer two in cross section view:

10. Select a cell that has some leachate dispersion in the vicinity of the landfill.
11. Switch to **Side View**.
12. Use the up and down arrows to view the solution along different columns.
13. When finished, switch to Plan View.

Note that the leachate eventually reaches both the river and the well. Now view the mass flux of leachate into the river that was computed with the transport observation package.

14. Select the “SourcesSink” coverage to make it active.

15. Using the Select Arcs tool, select the specified head arc along the bottom of the model. The computed mass flux is reported along the bottom of the window.

### 8.1 Generating a Mass vs. Time Plot

A plot of Mass vs. Time for a selected feature object can be created and viewed by doing the following:

1. Click Plot Wizard to bring up the Step 1 of 2 page of the Plot Wizard dialog.
2. Select “Mass vs. Time” from the list on the left.
3. Click Finish to create the plot and close the Plot Wizard dialog.

The plot shows the mass flux that leaves the model through the selected river arc.

4. Close the plot window and maximize the GMS Graphics Window.
5. Frame the project to resize the view of the model grid.

### 8.2 Viewing an Animation

Next, observe how the solution changes over the course of the simulation by generating an animation. Do the following to set up the animation:

1. Select the leachate” dataset in the Project Explorer.
2. Select Display | Animate… to bring up the Options page of the Animation Wizard dialog.
3. In the Options section, turn on Dataset.
4. Click Next to go to the Datasets page of the Animation Wizard dialog.
5. Turn on Display clock.
6. Click Finish to close the Animation Wizard dialog and launch the external Play AVI Application.
7. After clicking the Stop button, the Step button can be used to view the animation one frame at a time.

Feel free to experiment with some of the other playback controls.
8. When finished, close the Play AVI Application and return to GMS.

9  Modeling Sorption and Decay

The solution that was just computed can be thought of as a worst-case scenario because sorption and decay were neglected. Sorption will retard the movement of the plume and decay (due to biodegradation) will reduce the concentration. For the second part of the tutorial, modify the model so that sorption and decay are simulated. Then compare this solution with the first solution.

9.1  Turning on the Chemical Reactions Package

Sorption and decay are simulated in the Chemical Reactions Package. This package must be turned on before it can be used.

1. Select MT3DMS | Basic Transport Package… to open the Basic Transport Package dialog.
2. Click Packages… to open the MT3DMS/RT3D Packages dialog.
3. Turn on Chemical reaction package.
4. Click OK to exit the MT3DMS/RT3D Packages dialog.
5. Click OK to exit the Basic Transport Package dialog.

9.2  Entering the Sorption and Biodegradation Data

Next, enter the sorption and biodegradation data in the Chemical Reactions Package dialog.

1. Select MT3DMS | Chemical Reaction Package… to bring up the Chemical Reaction Package dialog.
2. Select “Linear isotherm” from the Sorption drop-down.
3. Select “First-order irreversible kinetic reaction” from the Kinetic rate reaction drop-down.
4. Enter “1” as the Layer.
5. Enter “53500.0” as the Bulk density.
6. In the spreadsheet, enter “0.00000585” in the leachate column of the 1st sorption const. row.
7. Enter “0.0001” in the leachate column on both the Rate const. (dissolved) and the Rate const. (sorbed) rows.
8. Enter “2” as the Layer.
9. Enter “51500.0” as the **Bulk density**.

10. In the spreadsheet, enter “0.00000585” in the *leachate* column of the *1st sorption const.* row.

11. Enter “0.00005” in the *leachate* column on both the *Rate const. (dissolved)* and the *Rate const. (sorbed)* rows.

12. Click **OK** to exit the *Chemical Reaction Package* dialog.

### 9.3 Run Options

It is nearly time to save the project under a new file name. However, the problem described earlier with the HFF file will be an issue because MT3D will look for a HFF file with the same name as the one about to be used to save the project. Because that file doesn’t exist, MODFLOW could be rerun to create it. Another way to address the issue is by doing the following:

1. Select **MT3DMS | Run Options…** to bring up the *Run Options* dialog.
2. Select **Single run with selected MODFLOW solution**.
3. Select “run1 (MODFLOW)” from the drop-down.
4. Click **OK** to close the *Run Options* dialog.

With this option, GMS tells MT3D to use the HFF file that was generated previously.

### 10 Saving and Running MT3DMS

It is now possible to save the new simulation.

1. Select **File | Save As…** to bring up the *Save As* dialog.
2. Select “Project Files (*.gpr)” from the *Save as type* drop-down.
3. Enter “run2.gpr” as the *File name*.
4. Click **Save** to save the project with the new name and close the *Save As* dialog.
5. Select **MT3DMS | Run MT3DMS…** to bring up the *MT3DMS* model wrapper dialog.
6. When the simulation is finished, turn on *Read solution on exit*.
7. Click **Close** to import the solution and close the *MT3DMS* dialog.

Notice again that it is possible to view the sorbed dataset.
11 Viewing the Solution

After the simulation finishes and the solution is read in to GMS:

1. Select the "leachate" dataset under the "run2 (MT3DMS)" in the Project Explorer.

2. In the Time Step list below the Project Explorer, select the last time step.

Notice that at the end of the simulation the plume is smaller and less advanced than the same time step in the first simulation.

11.1 Generating a Time History Plot

A useful way to compare two transient solutions is to generate a time history plot. The fastest way to do this is to create an “Active Data Set Time Series” plot.

1. Click Plot Wizard to bring up the Step 1 of 2 page of the Plot Wizard dialog.

2. Select “Active Data Set Time Series” from the list on the left.

3. Click Finish to generate the plot and close the Plot Wizard dialog.

4. Select a cell in the grid near the landfill. Notice that the plot shows the concentration v. time.

5. Select a different cell and notice that the plot updates. If no cell is selected, then the plot will not show any data.

To export the plot data and put it into a spreadsheet program:

6. Right-click on the plot and select View Values… to bring up the View Values dialog.

7. Select the desired data and press Ctrl-C.

8. Go into the desired spreadsheet program, select a cell, and press Ctrl-V to paste the copied data.

9. Back in GMS, click Close to exit the View Values dialog.

12 Conclusion

This concludes the “MT3DMS – Conceptual Model Approach” tutorial. The following key concepts were discussed and demonstrated in this tutorial:

- If starting with a MODFLOW conceptual model and wanting to do a transport model, first turn on transport in the conceptual model properties.
• It is possible to use the MT3D | Run Options… command to tell MT3D which MODFLOW solution to use.

Continue experimenting with MT3DMS or close the program.