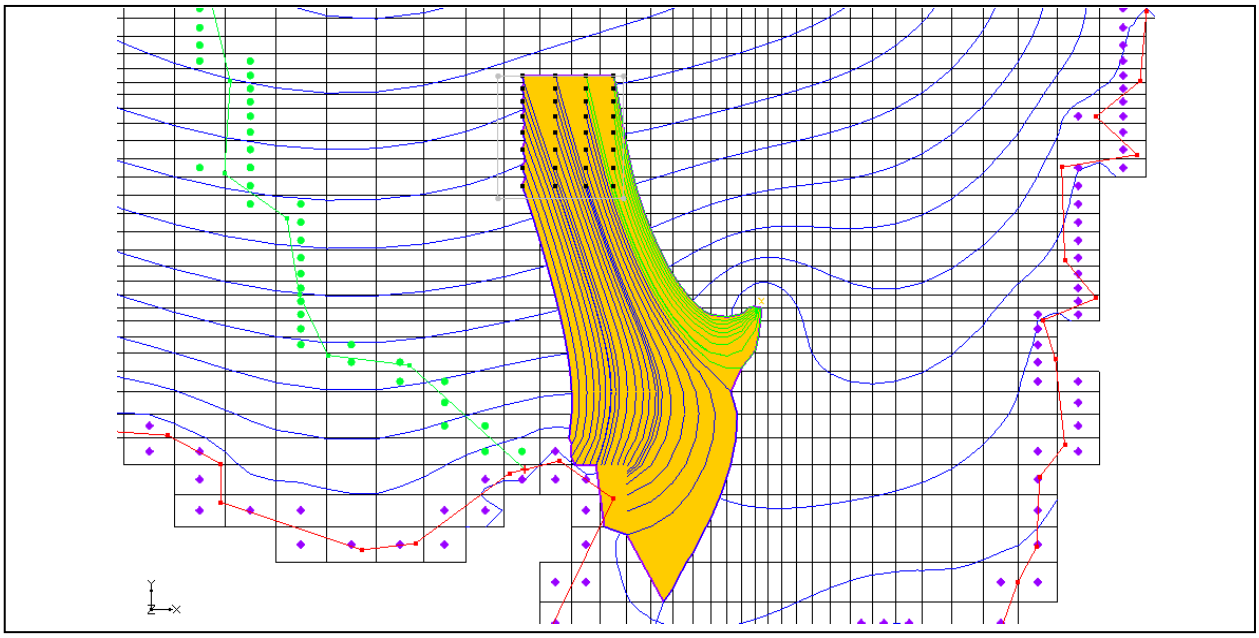
GMS v. 10.0

GMS 10.0 Tutorial

MODPATH

The MODPATH Interface in GMS



Objectives

Setup a MODPATH simulation in GMS and view the results. Learn about assigning porosity, creating starting locations, displaying pathlines in different ways, and displaying capture zones.

Prerequisite Tutorials

- MODFLOW – Conceptual Model Approach I

Required Components

- Grid Module
- Map Module
- MODFLOW
- MODPATH

Time

- 20-40 minutes

AQUAVEO™

1 Introduction 2
1.1 Outline 2
2 Description of Problem 2
3 Getting Started 3
4 Importing the Project 3
5 Assigning the Porosities 3
6 Defining the Starting Locations 4
6.1 Viewing the Pathlines in Cross Section View 4
7 Display Options 5
8 Particle Sets 5
8.1 Particle Sets Dialog 6
8.2 Duplicating Particle Sets 6
8.3 Changing the Display Order 7
9 Tracking Particles from the Landfill 7
9.1 Creating a New Particle Set 7
9.2 Defining the New Starting Locations 8
9.3 Capturing Landfill Particles at the Well 9
10 Color by Zone Code 10
11 Pathline Length/Time 10
12 Capture Zones by Zone Code 11
13 Conclusion 12

1 Introduction

This tutorial describes the steps involved in setting up a MODPATH simulation in GMS. MODPATH is a particle-tracking code developed by the U.S. Geological Survey. MODPATH tracks the trajectory of a set of particles from user-defined starting locations using the MODFLOW solution as the flow field. The particles can be tracked either forward or backward in time. Particle tracking solutions have a variety of applications, including the determination of zones of influence for injection and extraction wells.

1.1 Outline

Here are the steps to this tutorial:

1. Open the MODFLOW Conceptual Model project.
2. Create pathlines from various starting locations.
3. Edit the particle sets.
4. Edit zone codes, and view capture zones.

2 Description of Problem

The problem in this tutorial is an extension of the problem described in the "MODFLOW – Conceptual Model Approach" tutorial; the user should complete that tutorial before continuing if the user has not done so already.

In the "MODFLOW – Conceptual Model Approach" tutorial, a site in East Texas was modeled. The user will be using the solution from this model as the flow field for the particle tracking simulation. The model includes a proposed landfill. For this tutorial, the user will be performing two particle tracking simulations to analyze the long-term effects of contamination from the landfill. First, the user will do reverse particle tracking from the well on the east side of the model to see if the zone of influence of the well overlaps the landfill. Then the user will do forward tracking using an array of particles starting at the landfill to analyze the region of potential contamination for the landfill.

3 Getting Started

Do the following to get started:

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

4 Importing the Project

The first step is to import the East Texas project. This will read in the MODFLOW model and solution and all other files associated with the model.

To import the project, do as follows:

1. Select the **Open** button.
2. In the *Open* dialog, locate and open the directory entitled *Tutorials\MODFLOW\modfmap\sample2.*
3. Select the file entitled "modfmap2.gpr."
4. Click **Open.**

If the MODPATH menu is not visible, then select the *Edit* | **Model Interfaces** command and turn on the MODPATH option.

5 Assigning the Porosities

At this point, it is possible to create particles. First, however, the user will consider the porosity.

In order to calculate the tracking times, a porosity value must be defined for each of the cells in the grid. By default, GMS automatically assigns a porosity of 0.3 to all the cells in the grid. This value is acceptable, so nothing needs to be done.

If the user did want to change the porosity, it could be done in a number of ways. The first way is to assign porosities to the polygons in the conceptual model and then to select the *Map* → **MODFLOW / MODPATH** command. The second way is to select the **Porosity Array** command from the *MODPATH* menu in the 3D Grid module. This allows the user to edit a spreadsheet of values. Another way is to select grid cells and use the *MODPATH* | **Cell Properties** command to edit the porosity of the selected cells.

6 Defining the Starting Locations

Now the user needs to specify the starting locations for the particles. It is necessary to create a set of particle starting locations surrounding the cell containing the well on the east (right) side of the model.

To generate the starting locations:

1. Select and expand the "3D Grid Data" folder in the Project Explorer.
2. Select the *MODPATH* | **Create Particles at Wells** command to open the *Generate Particles at Wells* dialog.
3. Make sure the number of particles is set to "20."
4. Make sure that the *Extraction wells* option is selected.
5. Select the **OK** button.

A number of things happen now. GMS creates particles at every cell that contains an extraction well. It then saves a set of MODPATH input files to a temporary folder. Also, the user should now see a set of pathlines that converge on the east well. Notice that the pathlines extend to the northeast and miss the area covered by the proposed landfill.

This tutorial is not interested in the well on the west (left) side of the model, so the user will delete the particles and pathlines for that well.

6. Select the **Select Starting Locations** tool.
7. Drag a box surrounding the well on the west (left) side of the model.
8. Select the *Delete* button.

6.1 Viewing the Pathlines in Cross Section View

The 3D nature of the pathlines is best seen in cross section view.

1. Select the **Select Cells** tool.
2. Select a cell near the right of the landfill.

3. Select the **Side View** button.

The user may wish to move back and forth through the columns using the arrows in the *Mini-Grid Toolobar*. When finished, do as follows:

4. Select the **Plan View** button.

7 Display Options

In addition to displaying the pathlines, GMS can draw a closed boundary around the pathlines connected to the well. This boundary is referred to as a "capture zone." Capture zones can only be displayed in plan view. GMS has a number of options for the display of pathlines and capture zones.

1. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
2. Turn on *Direction arrows*.
3. Change the *Time interval* to "2000.0" below the *Direction arrows* toggle.
4. Make sure the *Boundary* option in the *Capture zones* section is turned on.
5. Turn on the *Poly fill* option in the *Capture zones* section.
6. Select the **OK** button.

The user should now see arrows on the pathlines pointing in the direction of flow. The user should also see the capture zone filled with a solid color.

8 Particle Sets

GMS organizes starting locations into "particle sets." When the user created the starting locations at the wells, GMS automatically created a particle set and put the new starting locations in it.

1. Expand the "Particle Sets" folder in the Project Explorer under the "3D Grid Data" folder and "grid" item.
2. Right-click on the particle set .
3. Select the **Properties** command from the pop-up menu.

8.1 Particle Sets Dialog

This brings up the *Particle Sets* dialog. Using the *Particle Sets* dialog, the user can change the particle set properties including the tracking direction and the tracking duration.

One particle set is always designated as the active particle set. Whenever new points are created, they are added only to the active particle set. Similarly, the user can only delete points from the active particle set.

By default, the tracking duration is set to track to the end, meaning MODPATH will track the particles until they run into something (a sink, the water table, or the edge of the model, etc.). Now change the tracking duration to a specific value.

1. In the *Track* column, switch the option to "Duration" in the pull-down list.
2. In the *Duration* column, change the value to "3000."
3. Click **OK**.

8.2 Duplicating Particle Sets

The next step is to display a 1,500-day capture zone as well as a 3,000-day capture zone. The user will turn the arrows off so they don't obscure the display of the capture zones.

1. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
2. Turn off the *Direction arrows.*
3. Click **OK**.

Now the user will create another particle set by copying the existing one.

4. Right-click on the particle set.
5. Click **rename** and change the name to "3000 days" so that it goes for 3000 days.
6. Right-click on the particle set.
7. Select the **Duplicate** command from the pop-up menu.
8. Right-click on the duplicate particle set.
9. Click **rename** and change the name to "1500 days."
10. Right-click on the "1500 days"  particle set.
11. Select the **Properties** command from the pop-up menu to open the *Particle Sets* dialog.

12. Change the *Duration* of the "1500 days" particle set to "1500.0."
13. Click **OK**.

8.3 Changing the Display Order

The order of the particle sets in the Project Explorer is the order in which they are displayed. Thus, the particle sets listed on top in the spreadsheet will be displayed on top of the ones underneath. The user can drag the particle sets up and down to change their order. Since the 1,500-day capture zone is smaller than the 3,000-day capture zone, the user needs to make sure that it is displayed on top.

1. In the *Project Explorer*, drag the "1500 days" particle set up so it is above the "3000 days" particle set.

The user should now see two capture zones, the larger one being the 3,000-day capture zone, and the smaller one being the 1,500-day capture zone.

9 Tracking Particles from the Landfill

Next, the user will perform forward tracking from a set of starting locations which coincide with the site of the proposed landfill.

9.1 Creating a New Particle Set

Create a new particle set for the particles that the user will create at the landfill.

1. In the Project Explorer, right-click on the "Particle Sets" folder.
2. Select the **New Particle Set** command.
3. Right-click on the new particle set.
4. Click **rename** and change the name to "Landfill."
5. Right-click on the "Landfill" particle set.
6. Select the **Properties** command to open the *Particle Sets* dialog.
7. Make sure the *Direction* of the "Landfill" particle set is "Forward."
8. Click **OK**.

9.2 Defining the New Starting Locations

Finally, the user will create a new set of starting locations at the site of the proposed landfill. The particles will be placed on the top of the ground water table to simulate leachate entering from the surface.

First, turn off the boundary fill option so the new pathlines are easier to see.

1. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
2. In the *Capture zones* section, turn off the *Poly fill* option.
3. Select the **OK** button.

Before selecting the cells, the user will make the recharge coverage the active coverage so that the landfill polygon is clearly visible.

4. In the Project Explorer, expand the "Map Data" folder and the "East Texas" conceptual module.
5. Select the "Recharge" coverage.

To select the cells:

6. Select the **Select Polygons** tool.
7. Select the landfill polygon.
8. Right-click on the selected polygon.

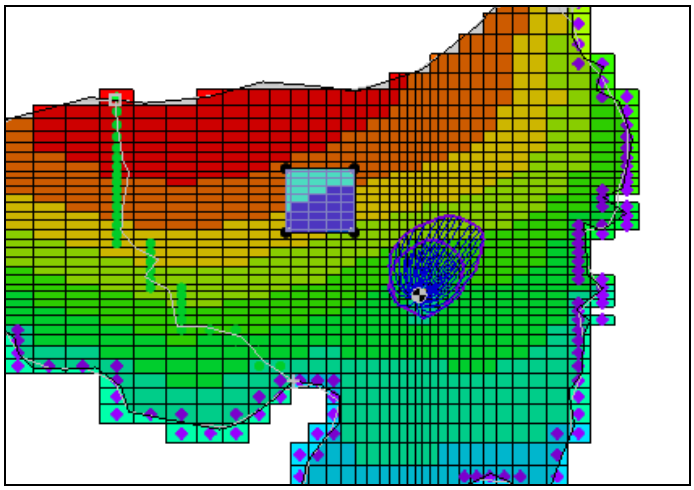


Figure 1 Landfill Polygon

9. Select the **Select Intersecting Objects** command from the menu.
10. Select **OK**.

The 3D grid cells should now be selected.

11. Select the *MODPATH* | **Create Particles at Selected Cells** command.
12. Make sure the *More options* toggle is off.
13. Make sure the *Distribute particles* option is set to "On water table surface.*"*
14. Select the **OK** button.

Now the user should see a set of pathlines starting at the landfill and terminating at the river at the bottom of the model.

15. Click anywhere outside the grid to unselect the cells.

9.3 Capturing Landfill Particles at the Well

Currently, no particles from the landfill are captured by the well. The user will increase the pumping rate in the well so that some of the particles from the landfill are captured.

1. Select the "Sources&Sinks" coverage in the Project Explorer.
2. Double-click on the well on the right side of the model to open the *Attribute Table* dialog.
3. Enter "-600.0" for the flow rate.
4. Select **OK**.
5. Select the *Feature Objects* | **Map→MODFLOW** command.
6. Select **OK** at the prompt.
7. Select the *File* | **Save As**… command.
8. Enter "modpath" as the file name.
9. Select **Save**.
10. Select the *MODFLOW* | **Run MODFLOW** command.
11. Select **Close** when MODFLOW is finished running.

After the MODFLOW computed heads are imported into GMS, MODPATH will run again with the new MODFLOW solution. The user should now see that the capture zones for the well are larger than before and some of the particles from the landfill terminate at the well.

10 Color by Zone Code

The user will want to easily identify the particles from the landfill that terminate at the well. So the user will make them a different color.

First, turn off the display of the particles coming from the well.

1. In the Project Explorer, turn off the display of the "1500 days" and "3000 days" particle sets by unchecking them.

Now change the zone code for the cell containing the well.

2. In the *Mini-grid Toolbar*, change the *Lay (k)* value to "2."
3. With the **Select Cells** tool, select the cell with the well in it. The user may need to zoom in to do this.
4. Right-click on the selected cell.
5. Choose the **Properties** command to open the *3D Grid Cell Properties* dialog.
6. Change the *MODPATH Zone code* to "2."
7. Click **OK**.
8. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
9. In the *Color* pull-down list, change the selection to "Ending code."
10. Select the **OK** button.

The pathlines that go from the landfill to the well should now be drawn in a different color.

11 Pathline Length/Time

One reason to do particle tracking is to find out how long it will take for particles to travel from one place to another. In this case, it is desirable to know how long it will take for particles to travel from the landfill to the well. GMS reports the length and travel time of selected pathlines.

1. Switch to the **Select Starting Locations** tool.
2. Click on one of the pathlines that goes from the landfill to the well.

In the status bar at the bottom of the GMS window, the user should see some statistics for the selected pathline. One of the items is the time. The user will want to know the

minimum time. The user could click on different pathlines one at a time and compare their times, but there's an easier way.

3. Select all the pathlines that go from the landfill to the well by dragging a box around their starting locations (the user may need to zoom in to do this).

In the status bar at the bottom of the GMS window, the user should see the maximum and minimum lengths and total time for all the selected pathlines.

12 Capture Zones by Zone Code

Notice that there is no closed boundary surrounding the pathlines originating from the landfill. By default, GMS only identifies capture zones for particles originating from wells. However, capture zones can be associated with particles originating from all cells with the same zone code. This feature can be used to group several wells together in the same capture zone. For example, if there were several wells located close together in a well field, the user might want to know what the combined capture zone is for all the wells.

The user can also use this feature to show the "capture zone" for the landfill.

1. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
2. Select the *Delineate by zone code* option in the *Capture Zones* section.
3. Turn on the *Poly fill* option.
4. Select the **OK** button.

The user should now see the capture zone for the landfill pathlines. Notice that the capture zone includes areas where there are no pathlines. To fix this, do as follows:

5. Select the *MODPATH* | **Display Options** command to open the *Display Options* dialog.
6. Change the *Thin triangle ratio* to "0.9."
7. Select the **OK** button.

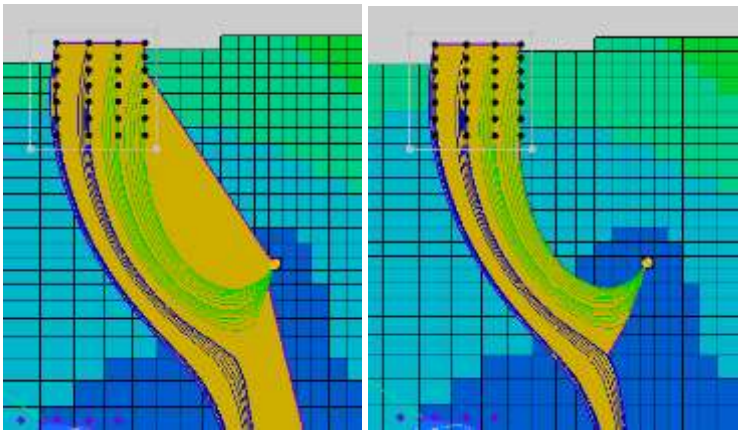


Figure 2 Capture Zones

Notice how the boundary of the capture zone has been "sucked in" so that it corresponds more closely to the pathlines. This is what the *Thin triangle ratio* does. If it is decreased too much, the capture zone will begin to look bad. The default was appropriate for the well capture zone seen earlier, but not for the landfill capture zone. The user will sometimes have to adjust this value to get a good-looking capture zone.

13 Conclusion

This concludes the *MODPATH* tutorial. Here are the key concepts in this tutorial:

- MODPATH is available whenever a MODFLOW model is in memory. MODPATH requires a flow solution before pathlines can be computed.
- It is possible to create particle starting locations in two ways using either the **Generate Particles at Wells** or **Generate Particles at Selected Cells** commands.
- Particles are grouped into particle sets. Particle sets are used to control the tracking direction, the duration, and the display order.
- A number of different display options are available for pathlines, including displaying arrows, coloring by zone code, and displaying filled polygons representing capture zones (in plan view only).