GMS 7.0 TUTORIALS

MODPATH

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 Contents

1.2 Outline

This is what you will do:

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.

1.3 Required Modules/Interfaces

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

- Grid
- Map
- MODFLOW
- MODPATH

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

2 Description of Problem

The problem we will be solving for this tutorial is an extension of the problem described in the previous tutorial entitled *MODFLOW - Conceptual Model Approach*. If you have not yet completed the previous tutorial, you may wish to do so before continuing.

In the previous tutorial, a site in East Texas was modeled. We will be using the solution from this model as our flow field for the particle tracking simulation. The model includes a proposed landfill. For this tutorial, we will be performing two particle tracking simulations to analyze the long term effects of contamination from the landfill. First we 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 we 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

Let's get started.

1. If necessary, launch GMS. 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:

1. Select the *Open* button .
2. In the *Open* dialog, locate and open the directory entitled **tutfiles\MODFLOW\modfmap\sample**.
3. Open the file entitled **modfmap.gpr**.

At this point we are ready to create particles. First, however, we will look at the porosity.

5 Assigning the Porosities

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 we don't need to do anything.

If we did want to change the porosity, we could do it in a number of ways. The first way is to assign porosities to the polygons in the conceptual model and selecting 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 you 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 we need to specify the starting locations for the particles. We want 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* | *Generate Particles at Wells* command.

3. Make sure the number of particles is set to **20** and the *Extraction wells* option is selected.

4. Select the *OK* button.

A number of things happen now. GMS creates particles at every cell that contains a well. It then saves a set of MODPATH input files to a temporary folder and automatically runs MODPATH in the background. When MODPATH is done running, GMS reads in the pathlines that MODPATH computes. This all takes place very quickly – usually in a second or two.

You should now see a set of pathlines that converge on the east well. Notice that the pathlines almost intersect the area covered by the proposed landfill, indicating a potential for leachate from the landfill to appear in the water pumped from the well.

We are not interested in the well on the west (left) side of the model, so we will delete the particles and pathlines for that well.

5. Select the Select Starting Locations tool .

6. Drag a box surrounding the well on the west (left) side of the model.

7. Hit 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 Cell* tool .

2. Select a cell near the right landfill.

3. Select the *Side View* button .

You may wish to move back and forth through the columns using the arrows ⬅ ➡ in the *Mini-grid plot*. When finished:

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 if you are in plan view. GMS has a number of options for the display of pathlines and capture zones.

1. Select the *MODPATH | Display Options* command.

2. Turn **on** *Direction arrows*.

3. Change the *Time interval* to **500.0** 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.

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

8 Particle Sets

GMS organizes starting locations into "particle sets". When we 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*.
2. Right-click on the particle set and 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 you 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, you 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.). Let's 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 **1,500**.
3. Click *OK*.

8.2 Duplicating Particle Sets

Let's display a 1500 day capture zone, and a 500 day capture zone. We'll turn the arrows off so they don't obscure the display of the capture zones.

1. Select the *MODPATH* | *Display Options* command.

2. Turn **off** the *Direction arrows* and click OK.

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

3. Using the *Project Explorer*, change the name of the particle set to **1500 days** so that we know it goes for 1500 days.

4. Right-click on the particle set and select the *Duplicate* command from the pop-up menu.

5. Rename the new particle set **500 days**.

6. Right-click on the 500 days particle set and select the *Properties* command from the pop-up menu.

7. Change the *Duration* of the **500 days** particle set to **500.0** and 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. You can drag the particle sets up and down to change their order. Since the 500 day capture zone is smaller than the 1,500 day capture zone, we need to make sure that it is displayed on top.

1. In the *Project Explorer*, drag the *500 days* particle set up so it is above the *1,500 days* particle set.

You should now see two capture zones, the larger one being the 1,500 day capture zone, and the smaller one being the 500 day capture zone.

8.4 Particle Sets within Cells

Notice that none of the particle sets thus far have directly intersected the landfill. Changing the location of the particles in a cell can result in very different MODPATH solutions. We will now create particles on the faces of the well cell whose pathlines will intersect the landfill.

1. Right-click on the *1500 days* particle set in the *Project Explorer* and select the *Duplicate* command.

2. Rename the new particle set to **end**.

3. Double-click on the *end* particle set.

4. In the *Track* column, select the **To end** option in the pull-down list and click *OK*.

5. In the mini-grid display, select the down arrow ⬇ to view the second layer.

6. Select the *Select Cell* tool and select the cell containing the well (you may need to zoom in to do this).
7. Select the *MODPATH* | *Generate Particles at Selected Cells* command.
8. Select *On cell faces* from the *Distribute particles* pull-down menu.
9. Turn on the *More options* toggle.
10. Change the *NZ* value to **4** for faces *1*, *2*, *3*, and *4*. This will create four rows of particles on each face.
11. Select *OK***.**
12. In the mini-grid display, select the up arrow ⬆ to return to the first layer.

You should now see a new capture zone representing the pathlines of particles distributed on the faces of the well cell instead of on the surface. Notice the new capture zone now intersects the right side of the landfill area.

9 Tracking Particles from the Landfill

Next, we 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 we will create at the landfill.

1. In the *Project Explorer*, right-click on the *Particle Sets* folder and select the *New Particle Set* command.
2. Change the name of the new particle set to **Landfill**.
3. Right-click on the *Landfill* particle set and select the *Properties* command.
4. Make sure the direction of the *Landfill* particle set is *Forward* and click *OK*.

9.2 Defining the New Starting Locations

Finally, we 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.

We'll turn off the boundary fill option so we can see the new pathlines easier.

1. Select the *MODPATH* | *Display Options* command.

2. In the *Capture zones* section, turn off the *Poly fill* option.

3. Select the *OK* button.

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

4. In the *Project Explorer,* expand the *East Texas* conceptual module and select the *Recharge* coverage.

To select the cells:

5. Select the *3D Grid Data* folder in the *Project Explorer*.

6. Select the *Select Cells* tool.

7. Select the cells covered by the landfill by dragging a rectangle that coincides with the landfill boundary.

8. Select the *MODPATH* | *Generate Particles at Selected Cells* command.

9. Turn off the *More options* toggle.

10. Change the *Distribute particles* option to *On water table surface.*

11. Select the *OK* button.

Now you should see a set of pathlines starting at the landfill and terminating in the well, the creek bed, and in the river at the bottom of the model. If none of your landfill pathlines go to the well, you may need to add particles to the column of cells just to the right of the cells you currently have particles in. You may wish to view this solution in cross section view as well.

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

10 Color by Zone Code

Some of the pathlines that start at the landfill should terminate at the well. We want to easily identify these so we will make them a different color.

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

1. In the *Project Explorer*, turn off the display of the 5**00 days**, **1,500 days**, and **end** particle sets by unchecking them.

Now we'll change the zone code for the cell containing the well.

2. Select the *Decrement* button ⬇ in the *Mini-Grid* Plot to view layer 2 of the grid.

3. With the *Select Cells* tool, select the cell with the well in it. You may need to zoom in to do this.

4. Select the *MODPATH* | *Cell Properties* command.

5. Change the *MODPATH Zone code* to **2** and click *OK*.

6. Select the *MODPATH* | *Display Options* command.

7. In the *Color* pull-down list, change the selection to **Ending code**.

8. 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 we want 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, you should see some statistics for the selected pathline. One of the items is the time. We want to know the minimum time. We 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 (you may need to zoom in to do this).

In the status bar at the bottom of the GMS window, you should see the maximum and minimum lengths and times 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, you might want to know what the combined capture zone is for all the wells.

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

1. Select the *MODPATH | Display Options* command.
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.

You 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:

5. Select the *MODPATH | Display Options* command.
6. Change the *Thin triangle ratio* to **0.85** and select the *OK* button.

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 you decrease it too much, the capture zone will begin to look bad. The default was appropriate for the well capture zone we saw earlier, but not for the landfill capture zone. You will sometimes have to adjust this value to get a good looking capture zone.

13 Conclusion

This concludes the *MODPATH* tutorial. Here are the things that you should have learned in this tutorial:

- MODPATH is available whenever a MODFLOW model is in memory. MODPATH requires a flow solution before pathlines can be computed.
- Unlike most models in GMS, MODPATH is much more automated, and GMS runs MODPATH in the background as soon as starting locations are created.
- You can 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. You use particle sets to control the tracking direction, the duration, and the display order.
- There are a number of different display options available for pathlines, including displaying arrows, coloring by zone code, and displaying filled polygons representing capture zones (in plan view only).

