GMS 7.0 TUTORIALS

MODFLOW - Grid Approach

1 Introduction

Two approaches can be used to construct a MODFLOW simulation in GMS: the grid approach and the conceptual model approach. The grid approach involves working directly with the 3D grid and applying sources/sinks and other model parameters on a cell-by-cell basis. The conceptual model approach involves using the GIS tools in the *Map* module to develop a conceptual model of the site being modeled. The data in the conceptual model are then copied to the grid.

The grid approach to MODFLOW pre-processing is described in this tutorial. In most cases, the conceptual model approach is more efficient than the grid approach. However, the grid approach is useful for simple problems or academic exercises where cell-by-cell editing is necessary. It is not necessary to complete this tutorial before beginning the *MODFLOW - Conceptual Model Approach* tutorial.

1.1 Contents

1.2 Outline

This is what you will do:

1. Create a 3D grid.
2. Set up a MODFLOW simulation.
3. Check the simulation and run MODFLOW.
4. Assign zone budgets and view the report.

1.3 Required Modules/Interfaces

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

- Grid
- MODFLOW

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

2 Description of Problem

The problem we will be solving in this tutorial is shown in Figure 1. This problem is a modified version of the sample problem described near the end of the MODFLOW *Reference Manual*. Three aquifers will be simulated using three layers in the computational grid. The grid covers a square region measuring 75000 ft by 75000 ft. The grid will consist of 15 rows and 15 columns, each cell measuring 5000 ft by 5000 ft in plan view. For simplicity, the elevation of the top and bottom of each layer will be flat. The hydraulic conductivity values shown are for the horizontal direction. For the vertical direction, we will use some fraction of the horizontal hydraulic conductivity.

Flow into the system is due to infiltration from precipitation and will be defined as recharge in the input. Flow out of the system is due to buried drain tubes, discharging wells (not shown on the diagram), and a lake which is represented by a constant head boundary on the left. Starting heads will be set equal to zero, and a steady state solution will be computed.

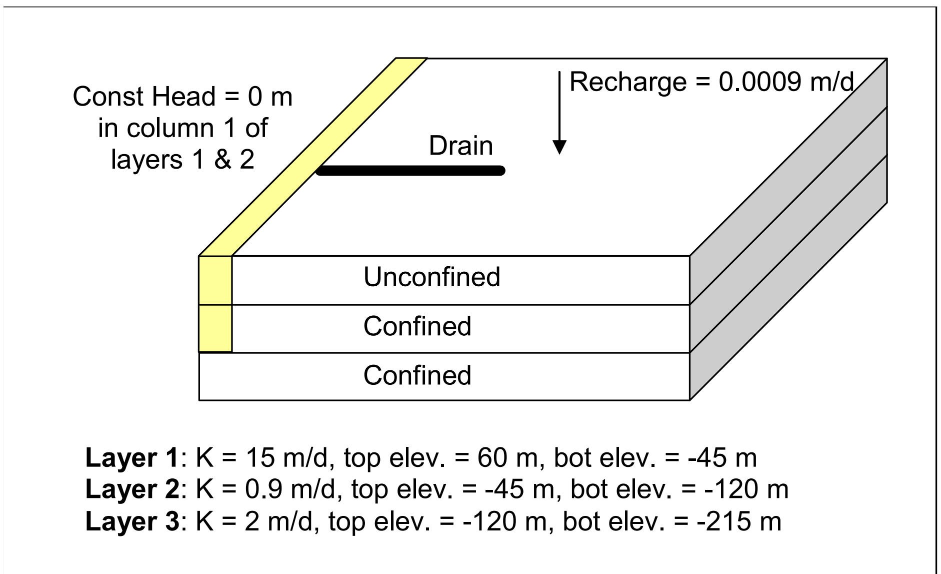


Figure 1. Sample Problem to be Solved.

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 Units

At this point, we can define the units used in the model. The units we choose will be applied to edit fields in the GMS interface to remind us of the proper units for each parameter.

1. Select the *Edit* | *Units* command.

2. For *Length*, enter **m** (for meter). For *Time*, enter **d** (for days). We will ignore the other units (they are not used for flow simulations).

3. Select the *OK* button.

5 Creating the Grid

The first step in solving the problem is to create the 3D finite difference 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 section entitled *X-dimension*, enter **22860** for the *Length* value, and **15** for the *Number cells* value.

3. In the section entitled *Y-dimension*, enter **22860** for the *Length* value, and **15** for the *Number cells* value.

4. In the section entitled *Z-dimension*, enter **3** for the *Number cells value*.

Later, we will enter the top and bottom elevations for each layer of the grid. Thus, the thickness of the cells in the z directions you enter here will not affect the *MODFLOW* computations.

5. Select the *OK* button.

The grid should appear in your window in plan view. A simplified representation of the grid should also appear in the *Mini-Grid Plot*.

Figure 2. Mini-Grid Plot

6 Creating the MODFLOW Simulation

The next step in setting up the model is to initialize the MODFLOW simulation.

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

6.1 The Global Package

The input to *MODFLOW* is subdivided into packages. Some of the packages are optional and some are required. One of the required packages is the Global package. We will begin with this package:

Packages

First, we will select the packages.

1. Select the *Packages* button.

The packages dialog is used to specify which of the packages we will be using to set up the model. The Basic package is always used and, therefore, it cannot be turned off. To select the other packages:

2. In the Point sources/sinks section, turn on the *Drain (DRN1)* and *Well (WEL1)* options.

3. In the Areal sources/sinks section, turn on the *Recharge (RCH1)* option.

4. In the Solver section, select the **Stongly Impl. Procedure (SIP1)** package.

5. Select the *OK* button to exit the *Packages* dialog.

The IBOUND Array

The next step is to set up the IBOUND array. The IBOUND array is used to designate each cell as either active (IBOUND>0), inactive (IBOUND=0), or constant head (IBOUND<0). For our problem, all cells will be active, except for the first two layers in the leftmost column, which will be designated as constant head.

1. Select the *IBOUND* button.

The *IBOUND* dialog displays the values of the IBOUND array in a spreadsheet-like fashion, one layer at a time. The edit field in the upper left corner of the dialog can be used to change the current layer. For our problem, we need all of the values in the array to be greater than zero, except for the left column of the top two layers, which should be less than zero. By default, the values in the array should already be greater than zero. Therefore, all we need to do is change the values for the constant head cells. This can be accomplished by entering a value of -1 for each of the thirty constant head cells. However, there is another way to edit the IBOUND array that is much simpler for this case. This method will be described later in the tutorial. For now we will leave all of the cells active.

2. Select the *OK* button to exit the *IBOUND* dialog.

Starting Heads

The next step is to set up the Starting Heads array.

1. Select the *Starting Heads* button.

The Starting Heads array is used to establish an initial head value when performing a transient simulation. Since we are computing a steady state simulation, the initial head for each cell shouldn't make a difference in the final solution. However, the closer the starting head values are to the final head values, the more quickly MODFLOW will converge to a solution. Furthermore, for certain types of layers, if the starting head values are too low, MODFLOW may interpret the cells as being dry. For the problem we are solving, an initial value of zero everywhere should suffice.

The Starting Heads array is also used to establish the head values associated with constant head cells. For our problem, the constant head values are zero. Since all of the starting head values are already zero by default, we don't need to make any changes.

2. Select the *OK* button to exit the *Starting Heads* dialog.

Top and Bottom Elevations

The next step is to set up the top and bottom elevation arrays.

1. Select the *Top Elevation* button.
2. Make sure the *Layer* is **1**.
3. Select the *Constant→Layer* button.
4. Enter a value of **60** and select the *OK* button.
5. Select the *OK* button to leave the *Top Elevations* dialog.

GMS forces the top of a layer to be at the same location as the bottom of the layer above. Thus, we only need to enter the bottom elevations of all the layers now and the tops of the layers will be set automatically.

6. Select the *Bottom Elevation* button.
7. Make sure the *Layer* is **1**.
8. Select the *Constant→Layer* button.
9. Enter a value of **-45** and select the *OK* button.
10. Change the *Layer* to **2**.
11. Select the *Constant→Layer* button.
12. Enter a value of **-120** and select the *OK* button.
13. Change the *Layer* to **3**.
14. Select the *Constant→Layer* button.

15. Enter a value of **-215** and select the *OK* button.

16. Select the OK button to exit the *Bottom Elevation* dialog.

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

7 Assigning IBOUND Values Directly to Cells

As mentioned above, the IBOUND values can be entered through the IBOUND Array dialog. In some cases, it is easier to assign values directly to cells. This can be accomplished using the *Cell Properties* command. Before using the command, we must first select the cells in the leftmost column of the top two layers.

7.1 Viewing the Left Column

To simplify the selection of the cells, we will change the display so that we are viewing the leftmost layer.

1. Choose the *Side View* button .

The grid appears very thin. To make things easier, we will increase the Z magnification so that the grid appears stretched in the vertical direction.

2. Select the *Display* Options button .
3. Change the *Z magnification* to **15**.
4. Select the *OK* button.

7.2 Selecting the Cells

To select the cells:

1. Choose the *Select Cells* tool .
2. Change the column to **1** in the *Mini-Grid Display* and hit the TAB key.

Notice that we are now viewing column number one (the leftmost column).

3. Drag a box around all of the cells in the top two layers of the grid.

7.3 Changing the IBOUND Value

To edit the IBOUND value:

1. Right-click on one of the selected cells.

2. Select the *Properties* command.

3. Change the *IBOUND* option to **Specified head**.

4. Select the *OK* button to exit the *3D Grid Cell Properties* dialog.

5. Select the *Plan View* button.

Notice that a symbol is displayed in the cells we edited, indicating that the cells are constant head cells.

7.4 Checking the Values

To ensure that the IBOUND values were entered correctly:

1. Select the *MODFLOW | Global Options* command.

2. Select the *IBOUND* button.

3. Choose the up arrow to the right of the layer field in the upper left corner of the dialog to cycle through the layers.

Notices that the leftmost column of cells in the top two layers both have a value of -1. Most of the *MODFLOW* input data can be edited in GMS using either a spreadsheet dialog such as this, or by selecting a set of cells and entering a value directly, whichever is most convenient.

4. Select the *OK* button to exit the *IBOUND Array* dialog.

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

8 The LPF Package

The next step in setting up the model is to enter the data for the Layer Property Flow (LPF) package. The LPF package computes the conductance between each of the grid cells and sets up the finite difference equations for the cell-to-cell flow.

To enter the LPF data:

1. Select the *MODFLOW | LPF Package* command.

8.1 Layer Types

The options in the *Layer Data* section of the dialog are used to define the layer type and hydraulic conductivity data for each layer. For our problem, we have three layers. The top layer is unconfined, and the bottom two layers are confined. The default layer type in GMS is “convertible”, which means the layer can be confined or unconfined. Thus, we don’t need to change the layer types.

8.2 Layer Parameters

The buttons in the *Layer Data* section of the dialog are for entering the parameters necessary for computing the cell-to-cell conductances. MODFLOW requires a set of parameters for each layer depending on the layer type.

8.3 Top Layer

First, we will enter the data for the top layer:

1. Select the *Horizontal Hydraulic Conductivity* button.
2. Select the *Constant → Layer* button.
3. Enter a value of **15**.
4. Select the *OK* button.
5. Select the *OK* button to exit the *Horizontal Hydraulic Conductivity* dialog.
6. Repeat this process to enter a value of **10** for the vertical anisotropy.

8.4 Middle Layer

Next, we will enter the data for the middle layer:

1. Select the up arrow to the right of the layer edit field in the *Layer Data* section to switch to layer **2**.
2. Enter the following values for layer 2:

Parameter	**Value**
Horizontal Hydraulic Conductivity	0.9 m/d
Vertical Anisotropy	5

8.5 Bottom Layer

Now we will enter the data for the bottom layer:

1. Switch to layer 3 and enter the following values:

Parameter	**Value**
Horizontal Hydraulic Conductivity	2 m/d
Vertical Anisotropy	5

This completes the data entry for this dialog.

2. Select the *OK* button to exit the *MODFLOW LPF Package* dialog.

9 The Recharge Package

Next, we will enter the data for the Recharge package. The Recharge package is used to simulate recharge to an aquifer due to rainfall and infiltration. To enter the recharge data:

1. Select the *MODFLOW* | *Source/Sink Packages* | *Recharge Package* command.
2. Select the *Constant→Array* button.
3. Enter a value of **0.0009** and click OK.
4. Select the OK button to exit the Recharge Package dialog.

10 The Drain Package

We will now define the row of drains in the top layer of the model. To define the drains, we must first select the cells where the drains are located, and then select the *Point Sources/Sinks* command.

10.1 Selecting the Cells

The drains are located in the top layer (layer 1). Since this is the current layer, we don't need to change the view.

We need to select the cells on columns 2-10 of row 8. To select the cells:

1. Choose the *Select Cells* tool.
2. Notice that as you move the cursor across the grid, the ijk indices of the cell beneath the cursor are displayed in the *Edit Window* at the bottom of the screen, as shown in Figure 3.

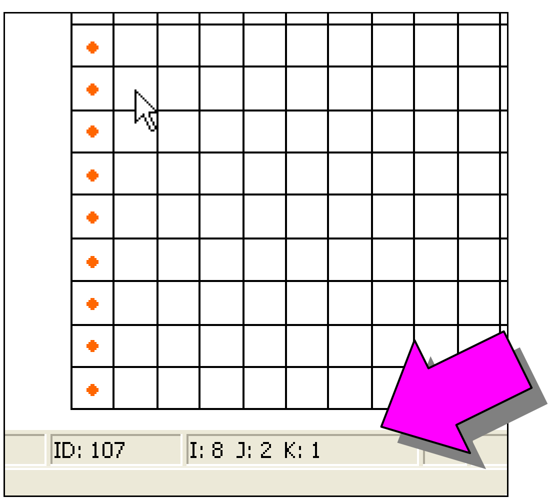


Figure 3. IJK indices of cell under cursor.

3. Select the cell at i=8, j=2, k=1.

4. Hold down the *Shift* key to invoke the multi-select mode and select the cells on columns 3-10 of the same row as the cell you have already selected (Figure 4).

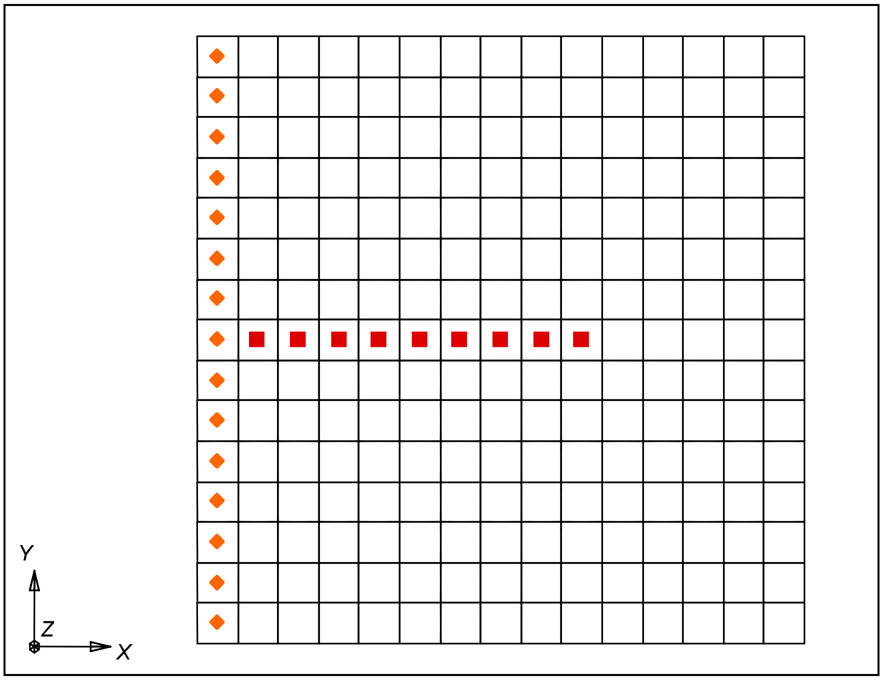


Figure 4. Cells to be Selected.

10.2 Assigning the Drains

To assign drains to the cells:

1. Right-click in the graphics window on the selected cells, and from the pop-up menu select the *Sources/Sinks* command.

2. Select the *Drain* tab.

3. Select the *New* button. This adds a new instance of a drain to each of the selected cells.

At this point we must enter an elevation and a conductance for the selected drains. The drains all have the same conductance but the elevations are not all the same.

4. Enter the following values for the elevations and conductances of the drains:

ID	Elevation	Conductance
107	0	7430
108	0	7430
109	3	7430
110	6	7430

111	9	7430
112	15	7430
113	20	7430
114	27	7430
115	30	7430

5. Select the *OK* button.
6. Unselect the cells by clicking anywhere outside the grid.

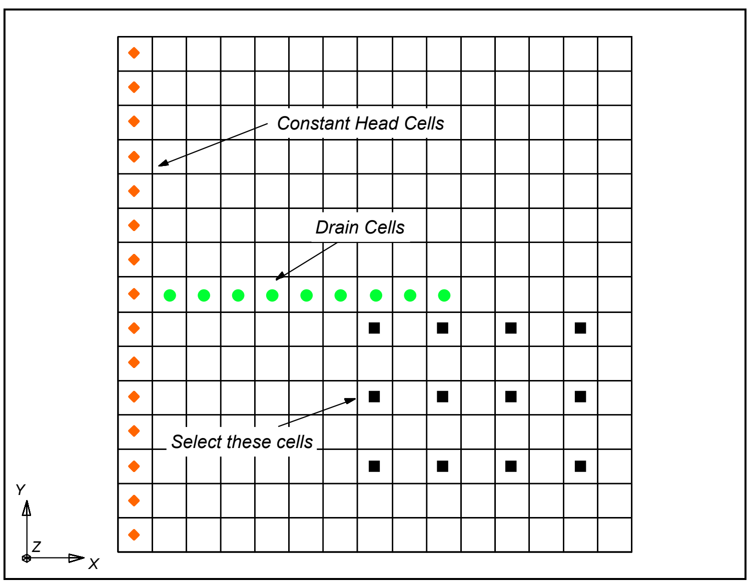
11 The Well Package

Next, we will define several wells by selecting the cells where the wells are located and using the *Point Sources/Sinks* command.

11.1 Top Layer Wells

Most of the wells are in the top layer but some are in the middle and bottom layers. We will define the wells in the top layers first.

1. While holding down the *Shift* key, select the cells shown in **Figure 5**.



row (i)	col (j)	lay (k)
9	8	1
9	10	1
9	12	1
9	14	1
11	8	1
11	10	1
11	12	1
11	14	1
13	8	1
13	10	1
13	12	1
13	14	1

Figure 5. Cells to be Selected on Top Layer

2. Right-click on one of the selected cells and select the *Sources/Sinks* command.
3. Select the *Well* tab.
4. Select the *New* button.

5. Enter a *Flow* value of **-12,230** for all the wells (a negative value signifies extraction).

6. Select the *OK* button.

7. Unselect the cells by clicking anywhere outside the grid.

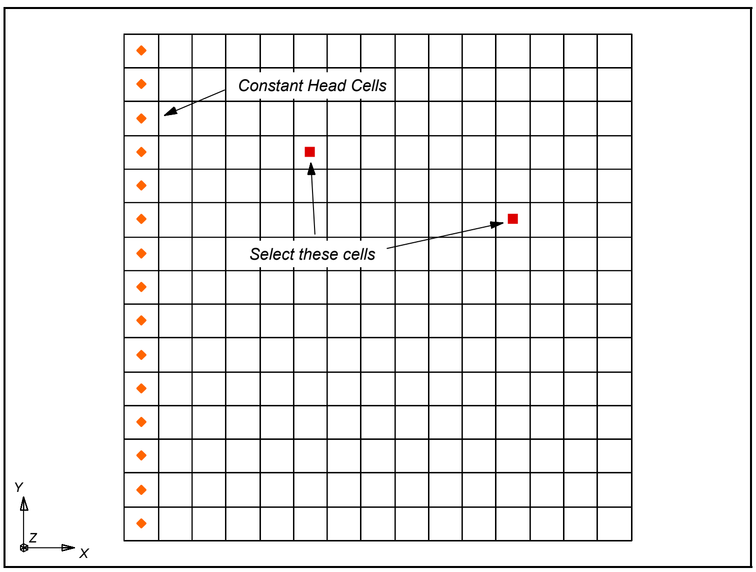
11.2 Middle Layer Wells

Next, we will define some wells on the middle layer. First, we need to view the middle layer.

1. Select the *Decrement* button ⬇ in the *Mini-Grid Plot*.

To select the cells:

2. While holding down the *Shift* key, select the cells are shown in **Figure 6**.



row (i)	col (j)	lay (k)
4	6	2
6	12	2

Figure 6. Cells to be Selected on Middle Layer.

3. Right-click on one of the selected cells and select the *Sources/Sinks* command.

4. Select the *Well* tab.

5. Select the *New* button.

6. Enter a *Flow* value of **-12,230** for both wells.

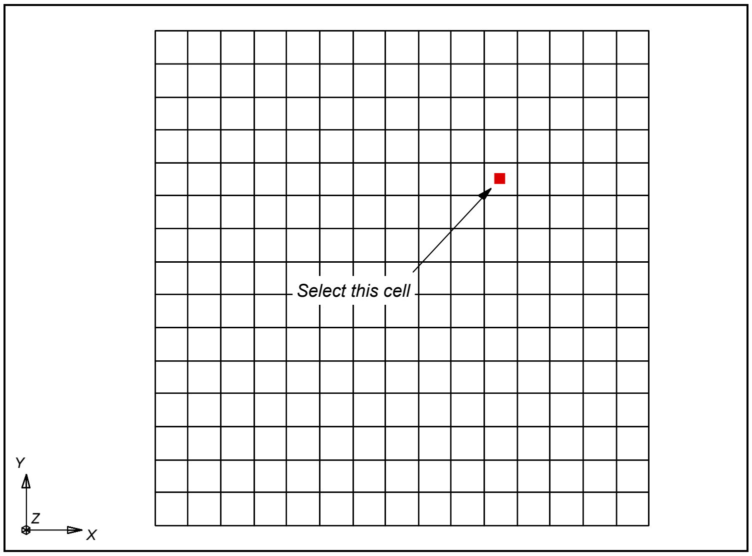
7. Select the *OK* button.

8. Unselect the cells by clicking anywhere outside the grid.

11.3 Bottom Layer Well

Finally, we will define a single well on the bottom layer. To view the bottom layer:

1. Select the *Decrement* button ⬇ in the *Mini-Grid Plot*.
2. Select the cell as shown in **Figure 7**.



row (i)	col (j)	lay (k)
5	11	3

Figure 7. Cell to be Selected on Bottom Layer.

3. Right-click on the selected cell and select the *Sources/Sinks* command.
4. Select the *Well* tab.
5. Select the *New* button.
6. Enter a *Flow* value of **-0.15**.
7. Select the *OK* button.
8. Unselect the cell by clicking anywhere outside the grid.

Now that all of the wells have been defined, we can go back to the top layer.

9. Select the up arrow ⬆ twice in the *Mini-Grid Plot*.

12 Checking the Simulation

At this point, we have completely defined the MODFLOW data and we are ready to run the simulation. However, before saving the simulation and running MODFLOW, we should run the MODFLOW *Model Checker* and check for errors. Because of the significant amount of data required for a MODFLOW simulation, it is often easy to omit some of the required data or to define inconsistent or incompatible options and parameters. Such errors will either cause MODFLOW to crash or to generate an erroneous solution. The purpose of the *Model Checker* is to analyze the input data currently defined for a MODFLOW simulation and report any obvious errors or potential problems. Running the *Model Checker* successfully does not guarantee that a solution will be correct. It simply serves as an initial check on the input data and can save a considerable amount of time that would otherwise be lost tracking down input errors.

To run the *Model Checker*:

1. Select the *MODFLOW | Check Simulation* command.

2. Select the *Run Check* button.

A list of messages is shown for each of the MODFLOW input packages. If you have done everything correctly, there should be no errors for any of the packages. When there is an error, if you select or highlight the error, when appropriate, GMS selects the cells or layers associated with the problem.

3. Select the *Done* button to exit the *Model Checker*.

13 Saving the Simulation

Now we are ready to save the simulation and run MODFLOW.

1. Select the *File | Save As* command.

2. Move to the directory titled **tutfiles\MODFLOW\modfgrid**

3. Save the project with the name **gridmod.gpr**.

14 Running MODFLOW

We are now ready to run MODFLOW:

1. Select the *MODFLOW | Run MODFLOW* command.

At this point MODFLOW is launched in a new window. The super file name is passed to MODFLOW as a command line argument. MODFLOW opens the file and begins the simulation. As the simulation proceeds, you should see some text output in the window reporting the solution progress.

2. When MODFLOW finishes, select the *Close* button.

15 Viewing the Solution

GMS reads the solution in automatically when you close the MODFLOW window. At this point you should see a set of head contours for the top layer. You may also see some cells containing a blue triangle symbol. These cells are flooded, meaning the computed water table is above the top of the cell.

15.1 Changing Layers

To view the solution on the middle layer:

1. Select the down arrow 🠋 in the *Mini-Grid Plot*.

To view the solution on the bottom layer:

2. Select the down arrow 🠋.

To return to the top layer:

3. Select the up arrow 🠉 twice.

15.2 Color Fill Contours

You can also display the contours using a color fill option.

1. Select the *Display* | *Contour Options* command.
2. Change the *Contour Method* to **Color Fill**.
3. Select the *OK* button.

15.3 Color Legend

Next, we will display a color legend.

1. Select the *Display* | *Color Ramp Options* command.
2. Turn on the *Legend* option.
3. Select the *OK* button.

16 Zone Budget

Zone Budget is a program developed by the USGS (Harbaugh 1990) that is used to compute subregional water budgets for MODFLOW ground-water flow models. GMS has incorporated a similar flow budget reporting tool. In GMS, the user defines zones by assigning a *Zone Budget ID* to cells. Once the zones are defined, a report can be generated that shows the flow budget for the zone. The report also includes a component that shows the flow in/out to adjacent zones.

16.1 Assigning Zone Budget Ids

In this model we will make each layer into a zone.

1. Choose the *Select Cells* tool .

2. If necessary, switch to *Plan View* .

3. Make sure you are looking at the second layer of the 3D grid. Adjust the layer by using the up ⬆ or down ⬇ arrow in the *Mini-Grid Plot* to view layer **2** of the grid.

4. Drag a box around all of the cells in layer 2 of the grid.

5. Right-click on one of the selected cells.

6. Selected the *Properties* command in the pop-up menu.

7. Enter **2** for the *Zone budget ID* and select the *OK* button.

8. Switch to layer 3 of the grid by selecting the down ⬇ arrow in the *Mini-Grid Plot*.

9. Repeat steps 4 through 7; except this time you need to enter **3** for the *Zone budget ID*.

16.2 Viewing the Zone Budget Report

We are now ready to view the flow budget for each of the zones.

1. Select the *MODFLOW | Flow Budget* command.

2. Select the *Zones* tab.

You are currently viewing the report for the first zone (the top layer of the grid). The report is divided into two sections: flow into the zone and flow out of the zone. Every source/sink that is present in the model is listed in the report with a flow value. In addition to the sources/sinks, there is a field for the amount of flow that goes between zones.

3. In the *Zone* drop-down box select **2**.

You may also want to view the report for zone 3. When you are done select the *OK* button to exit the dialog.

17 Conclusion

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

- You can specify which units you are using and GMS will display the units next to input fields to help you input the appropriate value. GMS does not do any unit conversions for you.
- The MODFLOW menu is in the 3D Grid module.
- The MODFLOW packages you want to use in your model can be selected by choosing the *MODFLOW | Global Options* command and clicking the Packages button.
- Most MODFLOW array data can be edited in two ways: via a spreadsheet, or by selecting grid cells and using the *MODFLOW | Cell Properties* command.
- Wells, Drains etc. can be created and edited by selecting the grid cell(s) and choosing the *MODFLOW | Sources/Sinks* command or by right-clicking on a selected cell and selecting the *Sources/Sinks* command from the pop up menu.
- You can use the *Model Checker* to analyze the input data and check for errors.
- You can generate a flow budget report for a sub-region of your model by assigning Zone budget ids to the grid and then using the *Flow Budget* command.
- In Ortho mode, only one row, or column, or layer of the 3D grid is visible at a time.