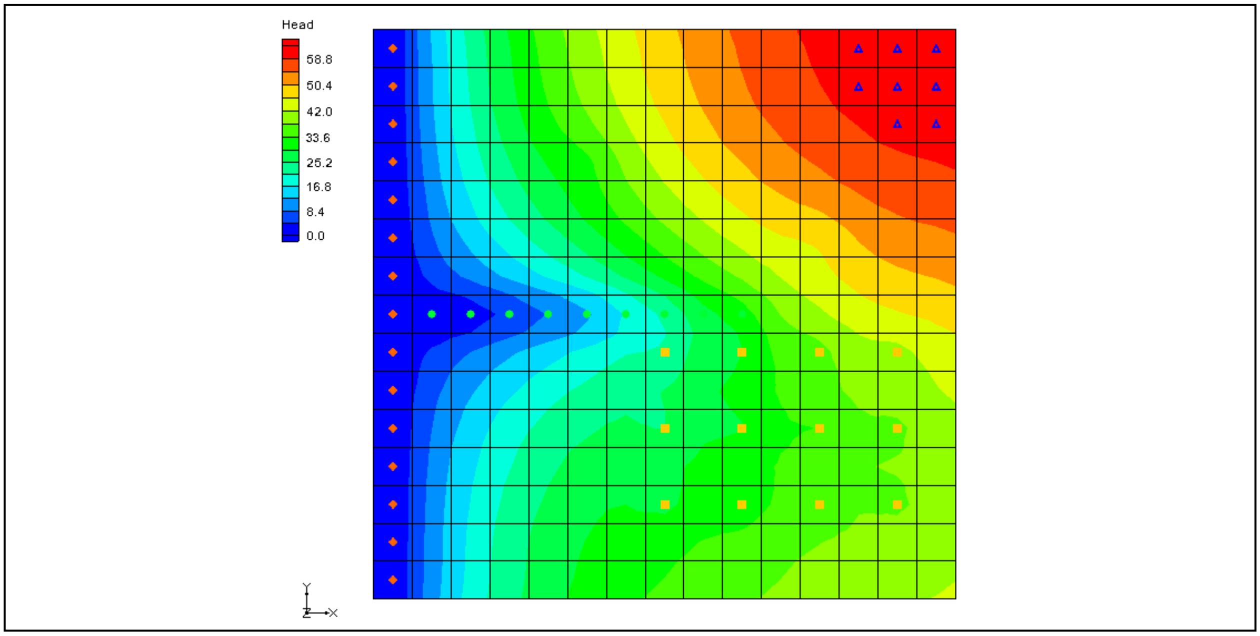
v. 10.0

GMS 10.0 Tutorial

MODFLOW – Grid Approach

Build a MODFLOW model on a 3D grid



Objectives

The grid approach to MODFLOW pre-processing is described in this tutorial. In most cases, the conceptual model approach is more powerful than the grid approach. However, the grid approach is useful for simple problems or academic exercises where cell-by-cell editing is necessary.

Prerequisite Tutorials

- None

Required Components

- Grid Module
- MODFLOW

Time

- 35-55 minutes

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 first and second "MODFLOW – Conceptual Model Approach" tutorials.

1.1 Outline

Here are the steps of this tutorial:

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.

2 Description of Problem

The problem that this tutorial will be solving 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 feet by 75000 feet. The grid will consist of 15 rows and 15 columns, each cell measuring 5000 feet by 5000 feet 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, the tutorial will use some fraction of the horizontal hydraulic conductivity.

Flow into the system is due to infiltration from precipitation and is 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.

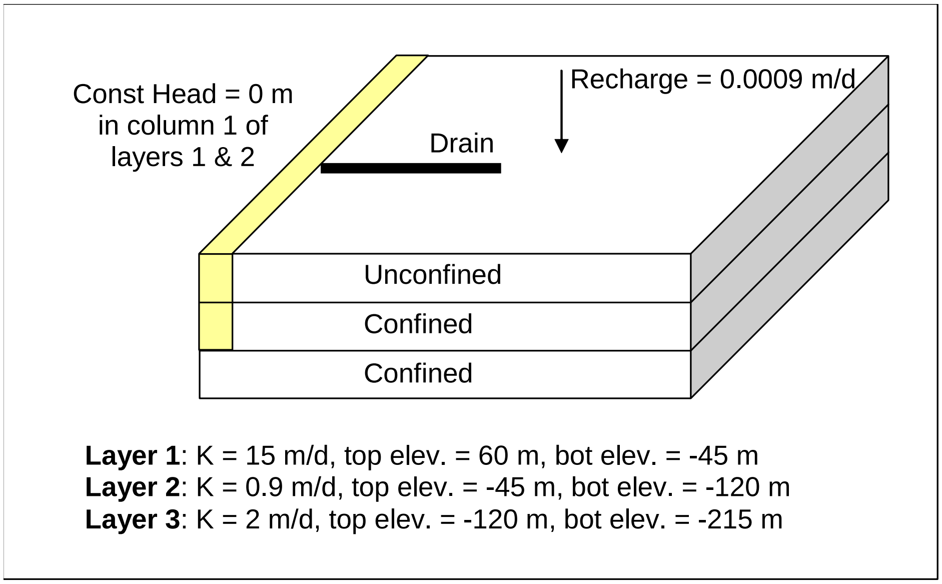


Figure 1 Sample problem to be solved

3 Getting Started

Do as follows 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 Units

At this point, it is possible to define the units used in the model. The units that are chosen 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*, click the "…" Button next to the length field.
3. In the *Display Projection* dialog, set the units for both horizontal and vertical to "Meters."
4. Click **OK**.
5. For *Time*, enter "d" (for days). Ignore the other units (they are not used for flow simulations).
6. Select the **OK** button to exit the *Units* dialog.

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

5. Select the **OK** button.

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

Figure 2 Mini-Grid toolbar

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" .

2. From the pop-up menu, select the **New MODFLOW** command.

The *MODFLOW Global/Basic Package* dialog will appear.

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.

Packages

First, select the packages.

1. Select the **Packages** button.

The *MODFLOW Packages* dialog will appear. It is used to specify which of the packages will be used to set up the model. The Basic package is always used, and, therefore, it cannot be turned off. To select the other packages, do as follows:

2. In the Optional packages section, turn on the *Drain (DRN1), Recharge (RCH1)*, and *Well (WEL1)* options.

3. In the Solver section, select the *Stongly Impl. Proc. (SIP1)* package.

4. Select the **OK** button to exit the *MODFLOW 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 this problem, all cells will be active, except for the first two layers in the leftmost column, which will be designated as constant head.

1. In the *MODFLOW Global/Basic Package* dialog, select the **IBOUND** button.

The *IBOUND Array* 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 this problem, all of the values in the array need 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, changing the values for the constant head cells is all that needs to be done. 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, leave all of the cells active.

2. Select the **OK** button to exit the *IBOUND Array* 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. In the *Top Elevation* dialog, make sure the *Layer* is "1."

3. Select the **Constant → Layer** button.

4. In the *Layer Value* dialog, enter a value of "60," and select the **OK** button.

5. Select the **OK** button to leave the *Top Elevation* dialog.

GMS forces the top of a layer to be at the same location as the bottom of the layer above. Thus, it is only necessary 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. In the *Bottom Elevation* dialog, make sure the *Layer* is "1."
8. Select the **Constant → Layer** button.
9. In the *Layer Value* dialog, enter a value of "-45," and select the **OK** button.
10. In the *Bottom Elevation* dialog, change the *Layer* to "2."
11. Select the ***Constant → Layer*** button.
12. In the *Layer Value* dialog, enter a value of "-120," and select the **OK** button.
13. In the *Bottom Elevation* dialog, change the *Layer* to "3."
14. Select the ***Constant → Layer*** button.
15. In the *Layer Value* dialog, 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

Starting Heads

The Starting Heads array is used to establish an initial head value when performing a transient simulation. Since the user is 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 at hand, the initial values are set to 60 because the *Starting heads equal grid top elevation* toggle is turned on.

The Starting Heads array is also used to establish the head values associated with constant head cells. For this problem, the constant head values should be zero. Since all of the starting head values are already 60 by default, it will be necessary to make these changes later in the tutorial.

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, it is necessary to 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, the user will change the display so that he or she is viewing the leftmost layer.

1. Choose the **Side View** button.

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

2. Select the **Display Options** button.
3. In the *Display Options* dialog, change the *Z magnification* to “15.”
4. Select the **OK** button.

7.2 Selecting the Cells

Do the following to select the cells:

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

Notice that the user is 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. In the *3D Grid Cell Properties* dialog, change the *IBOUND* option to “Specified head.”
4. Change the *Starting head* to “0.0.”
5. Select the **OK** button to exit the *3D Grid Cell Properties* dialog.
6. Select the *Plan View* button .

Notice that a symbol is displayed in the cells that were 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. In the *MODFLOW Global/Basic Package* dialog, 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.

Notice 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 – Layer Property Flow** command.

8.1 Layer Types

The options in the *Layer Data* section of the *LPF Package* dialog are used to define the layer type and hydraulic conductivity data for each layer. This problem has 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, it isn’t necessary 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, enter the data for the top layer:

1. Select the **Horizontal Hydraulic Conductivity** button.
2. In the *Horizontal Hydraulic Conductivity* dialog, select the **Constant → Layer** button.
3. In the *Layer Value* dialog, enter a value of "15" and click **OK**.
4. Select the **OK** button to exit the *Horizontal Hydraulic Conductivity* dialog.
5. Select the **Vertical Anisotropy** button.
6. In the *Vertical Anisotropy* dialog, select the **Constant → Layer** button.
7. In the *Layer Value* dialog, enter a value of "10" and click **OK**.
8. Select the **OK** button to exit the *Vertical Anisotropy* dialog.

8.4 Middle Layer

Next, 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. Using the **Constant → Layer** button, enter the following values for layer 2:

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

8.5 Bottom Layer

Now enter the data for the bottom layer:

1. Switch to layer 3 and, using the **Constant → Layer** button, enter the following values:

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

This completes the data entry for the *LPF Package* dialog.

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

9 The Recharge Package

Next, it is necessary to 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, do the following:

1. Select the *MODFLOW* | *Optional Packages* | **RCH – Recharge** command.
2. In the *MODFLOW Recharge Package* dialog, select the ***Constant → Array*** button.
3. In the *Grid Value* dialog, enter a value of “0.0009,” and click **OK**.
4. Select the **OK** button to exit the *MODFLOW Recharge Package* dialog.

10 The Drain Package

The next step is to define the row of drains in the top layer of the model. To define the drains, first select the cells where the drains will be 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, it isn’t necessary to change the view.

First, select the cells on columns 2-10 of row 8. To select the cells, do the following:

1. Choose the **Select Cells**  tool.
2. Notice that as the user moves 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.

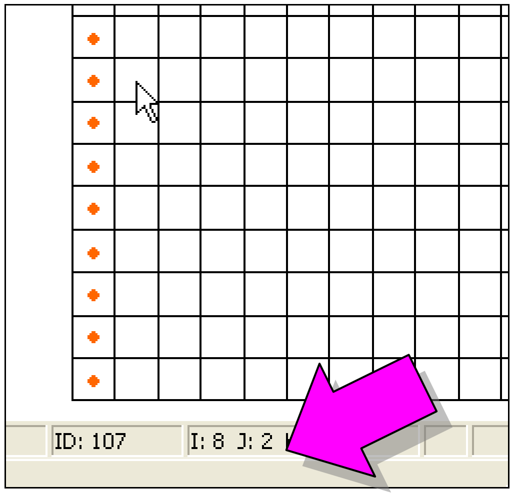


Figure 3 IJK indices of cell under cursor

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 that was already selected (Figure 4).

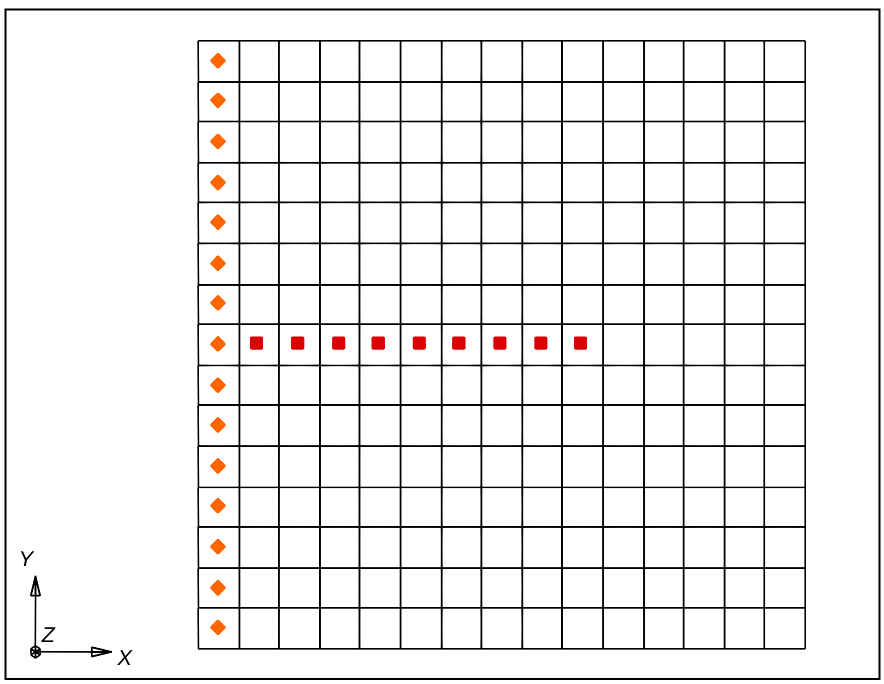


Figure 4 Cells to be selected

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 *Drain* from the list on the left.

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

At this point, the user 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. In the *MODFLOW Sources/Sinks* dialog, 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.

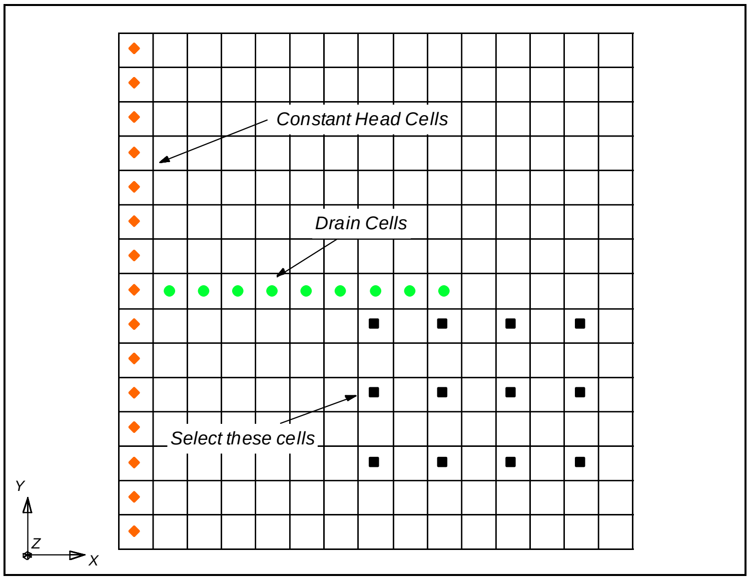
11 The Well Package

Next, the user 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. Define the wells in the top layers first:

1. While holding down the *Ctrl* 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. In the *MODFLOW Sources/Sinks* dialog, select *Wells* from the list on the left.
4. Select the **Add BC** button.
5. Enter a *Flow* value of "-12230" for all the wells (a negative value signifies extraction).
6. Select the **OK** button.
7. Unselect the cells by clicking anywhere outside the grid.

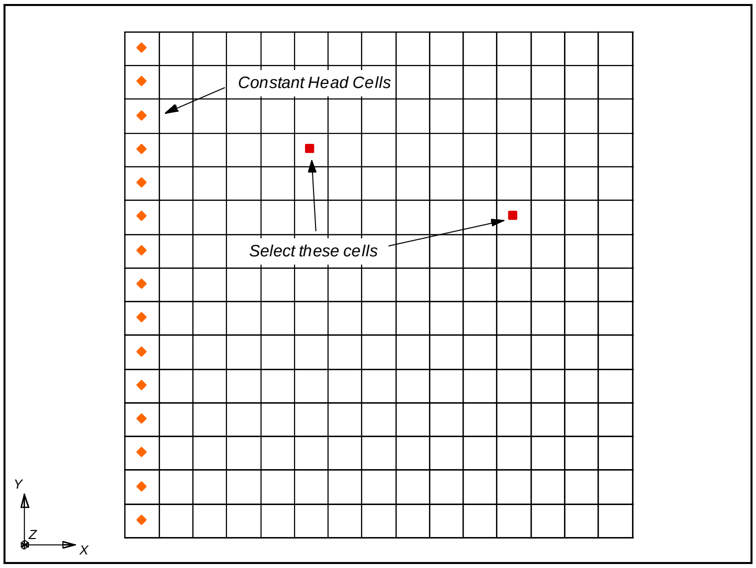
11.2 Middle Layer Wells

Next, the user will define some wells on the middle layer. First, it is necessary to view the middle layer.

1. Select the *Up* arrow in the *Mini-Grid Toolbar* to view layer 2.

To select the cells:

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



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

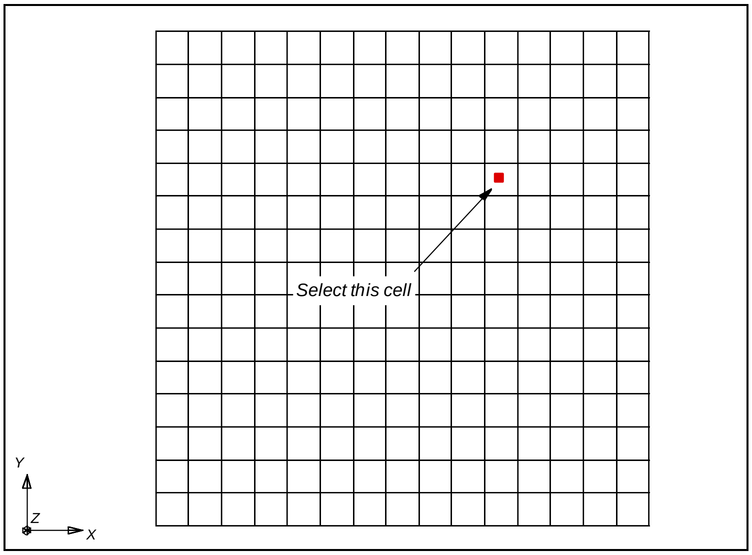
Figure 6. Cells to be selected on middle layer.

3. Right-click on one of the selected cells and select the **Sources/Sinks** command.
4. In the *MODFLOW Sources/Sinks* dialog, select *Wells* from the list on the left.
5. Select the **Add BC** button.
6. Enter a *Flow* value of "-12230" for both wells.
7. Select the **OK** button.
8. Unselect the cells by clicking anywhere outside the grid.

11.3 Bottom Layer Well

Finally, the user will define a single well on the bottom layer. Do the following to view the bottom layer:

1. Select the *Up* arrow in the *Mini-Grid Toolbar* to view layer 3.
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. In the *MODFLOW Sources/Sinks* dialog, select *Wells* from the list on the left.
5. Select the **Add BC** 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, it is possible to go back to the top layer.

9. Select the down arrow twice in the *Mini-Grid Toolbar* to view layer 1.

12 Checking the Simulation

At this point, the user has completely defined the MODFLOW data and is ready to run the simulation. However, before saving the simulation and running MODFLOW, the user 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.

Do the following to run the *Model Checker*:

1. Select the *MODFLOW* | **Check Simulation** command.
2. In the *Model Checker* dialog, select the **Run Check** button.

A list of messages is shown for each of the MODFLOW input packages. If the user has done everything correctly, the packages should have no errors. When an error exists, if the user selects or highlights 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 the user is ready to save the simulation and run MODFLOW.

1. Select the *File* | **Save As** command.
2. Move to the directory titled *Tutorials\MODFLOW\modfgrid*.
3. Save the project with the name "gridmod.gpr."

14 Running MODFLOW

It is now possible 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, the user 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 the MODFLOW window is closed. At this point, the user should see a set of head contours for the top layer. The user 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

Do the following to view the solution on the middle layer:

1. Select the up arrow in the *Mini-Grid Toolbar*.

Do the following to view the solution on the bottom layer:

2. Select the up arrow .

Do the following to return to the top layer:

3. Select the down arrow twice.

15.2 Color Fill Contours

It is also possible to display the contours using a color fill option.

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

2. In the *Dataset Contour Options – 3D Grid – Head* dialog, change the *Contour Method* to *Color Fill*.

15.3 Color Legend

Next, display a color legend:

3. In the *Dataset Contour Options – 3D Grid – Head* dialog, turn on the *Legend* option.

4. 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 groundwater 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, the user will make each layer into a zone.

1. Choose the **Select Cells** tool.

2. If necessary, switch to plan view.

3. Be sure to look at the second layer of the 3D grid. Adjust the layer by using the up or down arrow in the *Mini-Grid Toolbar* 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. Select 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 Toolbar*.

9. Repeat steps 4 through 7, except, this time, enter "3" for the *Zone budget ID*.

16.2 Viewing the Zone Budget Report

It is now possible to view the flow budget for each of the zones.

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

2. Select the *Zones* tab.

The user is 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."

The user may also want to view the report for zone 3.

4. When finished, select the **OK** button to exit the dialog.

17 Conclusion

This concludes the tutorial. Here are the key points of this tutorial:

- It is possible to specify which units are being used and GMS will display the units next to input fields to help the user input the appropriate value. GMS does not do any unit conversions for the user.

- The MODFLOW menu is in the 3D Grid module.

- The desired MODFLOW packages for use in the 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* | *Advanced* | **Cell Properties** command.
- Wells, drains, etc. can be created and edited by selecting the grid cell(s) and choosing the *MODFLOW* | *Advanced* | **Sources/Sinks** command or by right-clicking on a selected cell and selecting the **Sources/Sinks** command from the pop up menu.
- The *Model Checker* can be used to analyze the input data and check for errors.
- A flow budget report can be generated for a sub-region of the model by assigning Zone budget IDs to the grid and then using the **Flow Budget** command.
- In Ortho mode, only one row, column, or layer of the 3D grid is visible at a time.