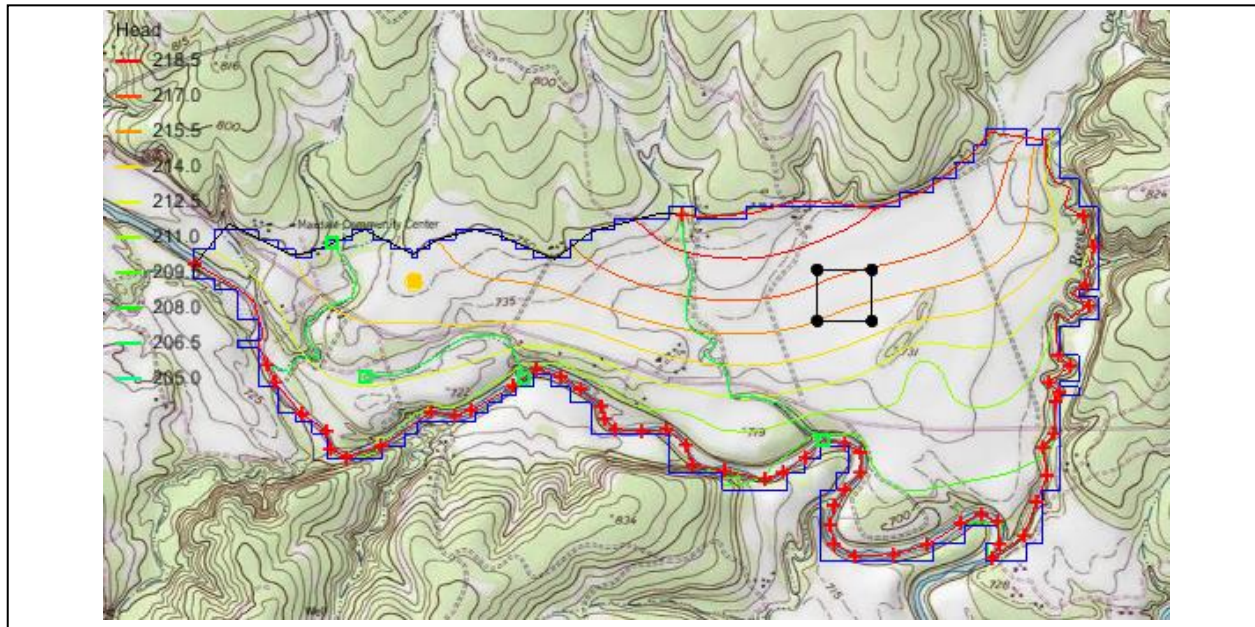


GMS 10.0 Tutorial

MODFLOW – Conceptual Model Approach II

Build a multi-layer MODFLOW model using advanced conceptual model techniques



Objectives

The conceptual model approach involves using the GIS tools in the Map Module to develop a conceptual model of the site being modeled. The location of sources/sinks, layer parameters such as hydraulic conductivity, model boundaries, and all other data necessary for the simulation can be defined at the conceptual model level without a grid.

Prerequisite Tutorials

- MODFLOW – Conceptual Model Approach I
- MODFLOW – Interpolating Layer Data

Required Components

- Grid Module
- Geostatistics
- Map Module
- MODFLOW

Time

- 35-50 minutes



1	Introduction	2
1.1	Outline	3
2	Description of Problem	3
3	Getting Started	5
4	Importing the Project	5
5	Saving the Project	5
6	Redefining the Recharge	5
6.1	Creating the Landfill Boundary	6
6.2	Rebuilding the Polygons	6
6.3	Assigning the Recharge Values	6
7	Redefining the Hydraulic Conductivity	7
7.1	Turning on Vertical Anisotropy	7
7.2	Copying the Layer 1 Coverage	7
7.3	Specify the Coverage Attributes	8
8	Modifying the Local Source/Sink Coverage	8
8.1	Turn on MNW2	9
8.2	Modifying the existing well	9
8.3	Importing the raster	10
8.4	Assigning the drain elevation	10
9	Recreating the Grid	11
10	Initializing the MODFLOW Data	11
11	Defining the Active/Inactive Zones	12
12	Interpolating Layer Elevations	12
12.1	Delete the Layer Elevations Coverage	13
12.2	Interpolating the Heads and Elevations	13
12.3	Interpolating the Layer Elevations	13
12.4	Adjusting the Display	14
12.5	Viewing the Model Cross Sections	14
12.6	Fixing the Elevation Arrays	14
13	Converting the Conceptual Model	15
14	Checking the Simulation	15
15	Saving the Project	16
16	Running MODFLOW	16
17	Viewing the Water Table in Side View	16
18	Viewing the Flow Budget	17
19	Conclusion	17

1 Introduction

This tutorial builds on the “MODFLOW – Conceptual Model Approach I” tutorial. In that tutorial, the user builds a one-layer model using the conceptual model approach. The top and bottom elevations are all the same, meaning the model is completely flat.

This tutorial will start with that model and turn it into a more complex and realistic model. The model will end up with two layers and varying top and bottom elevations that match the terrain and geology. The user will switch one of the wells to use the MNW2 package with a well screen that partially penetrates both layers. The user will also use an imported shapefile to define multiple recharge polygons to simulate a landfill.

1.1 Outline

Here are the steps of this tutorial:

1. Define a recharge polygon by importing a shapefile and converting it to feature objects.
2. Define well screens in the conceptual model.
3. Assign drain elevations using a raster.
4. Create a two layer grid from the conceptual model.
5. Interpolate scatter points to MODFLOW layer data.
6. Map the conceptual model to MODFLOW.
7. Check the simulation and run MODFLOW.
8. View the results.

2 Description of Problem

The problem to be solved in this tutorial is illustrated in Figure 1. The site is located in East Texas. This tutorial will assume that the user is evaluating the suitability of a proposed landfill site with respect to potential groundwater contamination. The results of this simulation will be used as the flow field for a particle tracking and a transport simulation in the MODPATH tutorial and the MT3DMS tutorial.

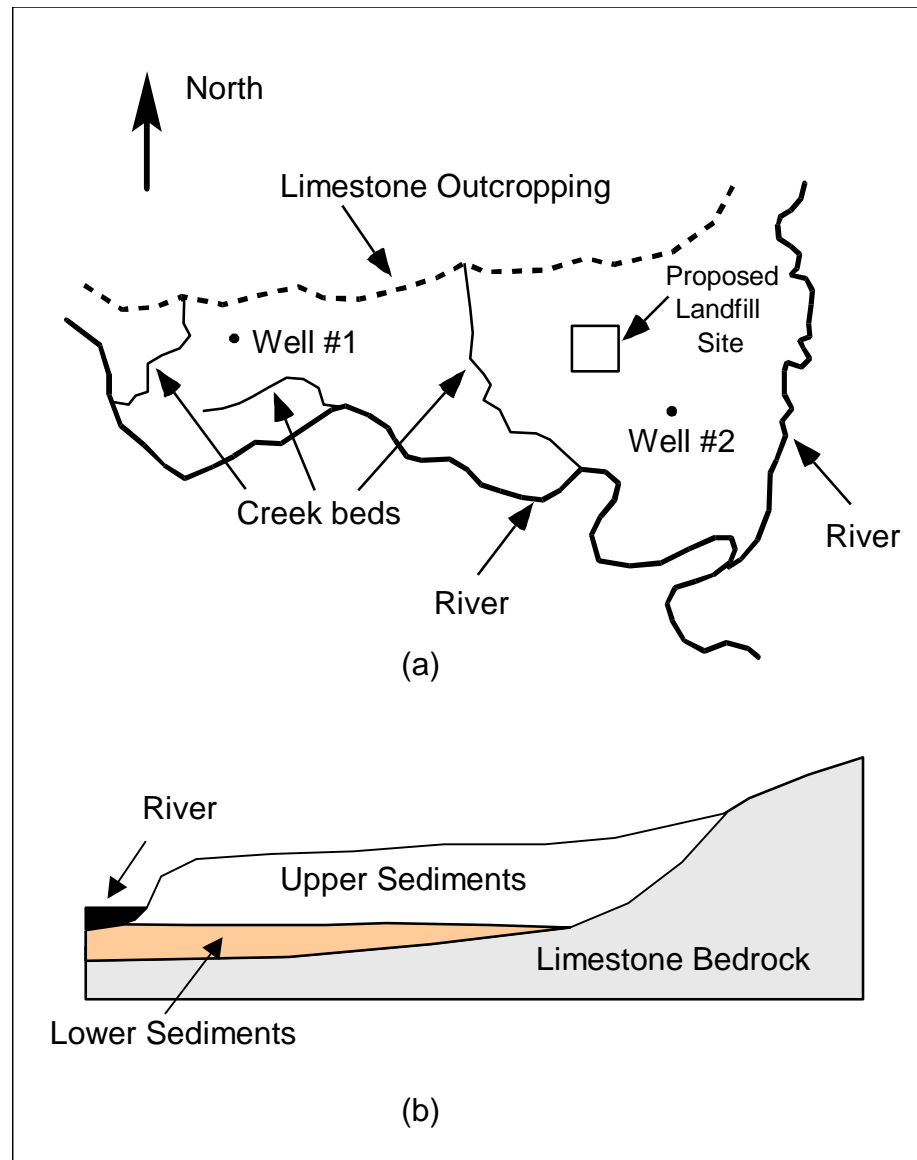


Figure 1 Site to be modeled in this tutorial. (a) Plan view of site. (b) Typical north-south cross section through site

The tutorial will be modeling the groundwater flow in the valley sediments bounded by the hills to the north and the two converging rivers to the south. A typical north-south cross section through the site is shown in Figure 1b. The site is underlain by limestone bedrock which outcrops to the hills at the north end of the site. There are two primary sediment layers. The upper layer will be modeled as an unconfined layer, and the lower layer will be modeled as a confined layer.

The boundary to the north will be a no-flow boundary, and the remaining boundary will be a specified head boundary corresponding to the average stage of the rivers. The user will assume the influx to the system is primarily through recharge due to rainfall. Some creek beds in the area are sometimes dry but occasionally flow due to influx from the groundwater. The tutorial will represent these creek beds using drains. There are also two production wells in the area that will be included in the model.

NOTE: Although the site modeled in this tutorial is an actual site, the landfill and the hydrogeologic conditions at the site have been fabricated. The stresses and boundary conditions used in the simulation were selected to provide a simple yet broad sampling of the options available for defining a conceptual model.

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, the 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\sample1*.
3. Select the file entitled “modfmap.gpr.”
4. Click the **Open** button.

5 Saving the Project

Before making any changes, save the project under a new name.

1. Select the *File* | **Save As** command.
2. Save the project with the name “modfmapII.”


Be sure to hit the **Save**  button periodically as the model is developed.


6 Redefining the Recharge



The user will assume that the recharge over the area being modeled is uniform except for the landfill. The recharge in the area of the landfill will be reduced due to the landfill liner system.

6.1 Creating the Landfill Boundary

Next, the user will create the arc delineating the boundary of the landfill. In this tutorial, the user will import the boundary from an existing landfill shapefile.

1. Click on “GIS Layers”  folder in the Project Explorer.
2. Select the *GIS / Add Shapefile Data* menu command.
3. Choose *Tutorials\MODFLOW\modfmap*
4. Select the “landfill_arcs.shp” file.
5. Click **Open**.

Now that the shapefile is imported, the user will convert this shape file into feature objects in the “Recharge”  coverage.

6. Expand the “Map Data” folder.
7. Expand the “East Texas”  conceptual model.
8. Make the “Recharge”  coverage the active one by selecting it in the Project Explorer.
9. Select “landfill_arcs.shp” in the “GIS Layers” folder in the Project Explorer.
10. Select the *GIS / Shapes → Feature Objects* command
11. Select the **Yes** button to map all visible shapes.
12. In the *GIS to Feature Objects Wizard* dialog, select the **Next** button.
13. Select *Elevation* for the ARC_ELEV.
14. Select **Next** then **Finish** to complete the process.

The landfill is now created.



6.2 Rebuilding the Polygons

Now that the landfill boundary is defined, it is necessary to rebuild the polygons.

1. Select the **Build Polygons**  macro.

6.3 Assigning the Recharge Values

Now that the recharge zones are redefined, it is possible to assign the recharge values for the landfill polygon.

1. Click on the “Recharge”  coverage.
2. Select the **Select Polygons**  tool.
3. Double-click on the landfill polygon.
4. In the *Attribute Table* dialog, change the *Recharge rate* to “0.00006.”


Note: This recharge rate is small relative to the rate assigned to the other polygons. The landfill will be capped and lined and thus will have a small recharge value. The recharge essentially represents a small amount of leachate that escapes from the landfill.

5. Select the **OK** button.

7 Redefining the Hydraulic Conductivity


In the previous tutorial, the user only simulated a one-layer model. In this tutorial, the user will make a two-layer model. Thus, it is necessary to define the hydraulic conductivity for the second layer and the vertical anisotropy for both layers. Similar to the previous tutorial, the user will also use constant values for the second layer.


7.1 Turning on Vertical Anisotropy

1. Right-click on the “Layer 1”  coverage.
2. Select the **Coverage Setup** command from the pop-up menu.
3. In the *Coverage Setup* dialog, from the *Areal Properties* list, turn on *Vertical anis.*
4. Click **OK**.
5. Double-click on the landfill polygon in the “Layer 1” coverage.
6. In the *Attribute Table* dialog, change the *Vertical anis.* to “4.” This means that the vertical hydraulic conductivity will be one fourth of the horizontal conductivity.
7. Select the **OK** button.

7.2 Copying the Layer 1 Coverage


The user will create the “Layer 2” coverage by copying the “Layer 1” coverage.

1. Right-click on the “Layer 1”  coverage.
2. Select the **Duplicate** command from the pop-up menu.
3. Right-click on the “Copy of Layer 1” coverage.

4. Select the **Properties** command.
5. In the *Properties* dialog, change the name of the new coverage to “Layer 2.”
6. Click **OK**.
7. Right-click on the “Layer 2”  coverage.
8. Select the **Coverage Setup** command.
9. In the *Coverage Setup* dialog, change the *Default layer range* to go from “2” to “2.”
10. Select the **OK** button.

7.3 Specify the Coverage Attributes

For the new layer, do the following:

1. Select the “Layer 2”  coverage in the Project Explorer.
2. Double-click on the landfill polygon
3. In the *Attribute Table* dialog, change the *Horizontal K* to “10.”
4. Select the **OK** button.

8 Modifying the Local Source/Sink Coverage

Since our model will have two layers, when mapping the conceptual model to the grid, it will be necessary to specify which grid layer the wells should be placed in. There are three ways to do this.

The simplest way is to specify the grid layer in the conceptual model, but that requires the user to know how many grid layers he or she will have and where they will be when building the conceptual model. Also, if the user decides to add or subtract grid layers later, he or she will have to remember to change the conceptual model.

The second way is to use a well screen with the WEL package. This allows the user to specify the top and bottom of the screened interval of the well. When the conceptual model is mapped to the grid, the well will be placed automatically in the appropriate grid layer (or layers) based on which grid layers intersect the well screen. If multiple grid layers are intersected by the well screen, multiple wells will be created.


The third way is to use the MNW2 package and define the screened interval. The MNW2 package is a more realistic well package that better models partially penetrating wells and wells that have more than one screened interval and/or draw from multiple grid layers. This tutorial will use this option.

The user will also modify the conceptual model so that it uses real terrain data for elevations. The terrain elevations will come from a raster. Also, the user will use the raster to define the drain elevations. When the drain arcs are discretized onto the model grid, the cells that intersect the arcs will be found. Then the drain elevation is interpolated from the raster to the cell centers.

This method is particularly helpful with large models where otherwise the user would have to manually determine and enter the elevation for each drain at the arc nodes.




8.1 Turn on MNW2

It is necessary to make the MNW2 properties available in the coverage.

1. Right-click the “Sources & Sinks”  coverage in the Project Explorer.
2. Select the **Coverage Setup** command from the pop-up menu.
3. In the *Coverage Setup* dialog, in the list of *Sources/Sinks/BCs*, turn on *Wells (MNW2)*.
4. Click **OK** to exit.

8.2 Modifying the existing well

First edit the eastern well.

1. Select the “Sources & Sinks”  coverage in the Project Explorer.
2. Switch to the **Select Points\Nodes**  tool.
3. Select the well on the eastern (right) side of the model.
4. Select **Properties**  button.
5. In the *Attribute Table* dialog, change the *Type* to “well (MNW2).”
6. Scroll to the right and enter “-300.0” in the *Qdes* field.
7. Make sure the *Vertical Boreline* option is on.
8. For the *LOSSTYPE* field, choose the “THIEM” option.
9. Click on the ... button under *Boreline*.
10. In the *Table* dialog, enter “180” (m) for *Z screen begin* and “165” (m) for *Z screen end*.

These values make the well screen go through both layer 1 and layer 2.

11. Change R_w to “0.05.”

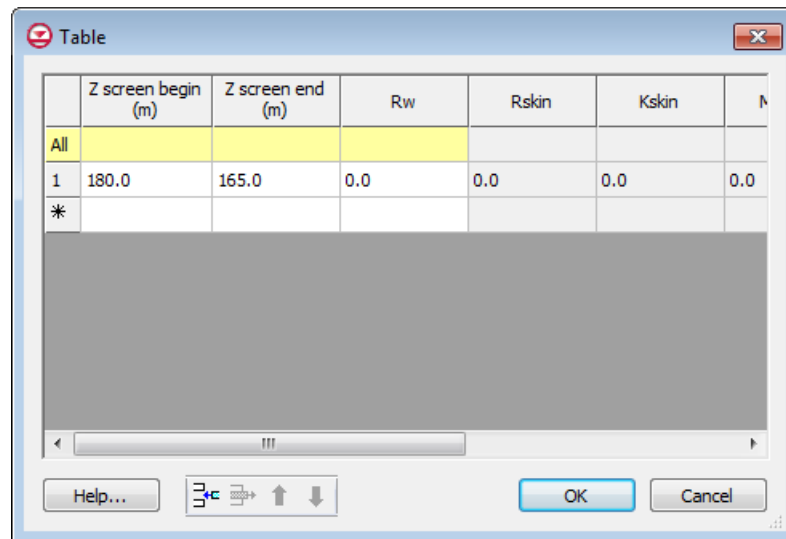



Figure 2 Defining the screened interval

12. Click **OK** to exit the *Table* dialog.

13. Click **OK** to exit the *Attribute Table* dialog.

8.3 Importing the raster



Next, it is possible to use elevation data to define the drain elevation. It is necessary to import the raster so it can be used in our model.


1. Select the **Open**  button.
2. In the *Open* dialog, locate the file entitled “elev_10.tif.”
3. Click the **Open** button.

The user should now see an elevation raster appear in the background of the project.

8.4 Assigning the drain elevation

It is now possible to assign drain elevation using these data.

1. Select the “Sources & Sinks”  coverage in the Project Explorer
2. Select the **Select Points/Nodes**  tool.
3. Select *Edit* / **Select All** command.

4. Select **Properties**  button.
5. In the *Attribute Table* dialog, change the *BC type* to “drain.”
6. In the first row under *Bot. elev.*, click on the drop-down box and select “<Raster>.”
7. In the *Select Raster* window, select the “elev_10.tif.”
8. Select the **OK** button.
9. Select the **OK** button exit the *Attribute Table* dialog.

9 Recreating the Grid

It is now possible to create a new two layer grid.


1. Select the *Feature Objects | Map → 3D Grid* command.
2. Click **OK** twice to confirm deletion of the grid and the current MODFLOW model.

Notice that the grid is dimensioned using the data from the Grid Frame. If a Grid Frame does not exist, the grid is defaulted to surround the model with approximately 5% overlap on the sides. Also note that the number of cells in the x and y dimensions cannot be altered. This is because the number of rows and columns and the locations of the cell boundaries will be controlled by the refine point data entered at the wells.

3. In the *Create Finite Difference Grid* dialog, in the *Z-Dimension* field, change *Number cells* to “2.”
4. Select the **OK** button.





10 Initializing the MODFLOW Data

Now that the grid is constructed and the active/inactive zones are delineated, the next step is to convert the conceptual model to a grid-based numerical model. Before doing this, however, the user must first initialize the MODFLOW data:

1. Right-click on the “grid”  item in the Project Explorer.
2. Select the **New MODFLOW** command.
3. When the *MODFLOW Global/Basic Package* dialog appears, select the **OK** button.

11 Defining the Active/Inactive Zones

Now that the grid is created, the next step is to define the active and inactive zones of the model. This is accomplished automatically using the information in the local sources/sinks coverage.

1. Select the “Map Data”  folder in the Project Explorer.
2. Select the “Sources&Sinks”  coverage to make it active.
3. Select the **Select Polygons**  tool.
4. Select one of the polygons.
5. Select the **Properties**  button.
6. In the *Attribute Table* dialog, confirm that the layer assignment for the *From layer* is “1” and the *To layer* is “2.”
7. Click **OK**.
8. Select the *Feature Objects* / **Activate Cells in Coverage(s)** command.

Each of the cells in the interior of any polygon in the local sources/sinks coverage is designated as active and each cell which is outside of all of the polygons is designated as inactive. Notice that the cells on the boundary are activated such that the no-flow boundary at the top of the model approximately coincides with the outer cell edges of the cells on the perimeter while the specified head boundaries approximately coincide with the cell centers of the perimeter cells.

12 Interpolating Layer Elevations

Now it is necessary to define the layer elevations and the starting head. Since this tutorial is using the LPF package, top and bottom elevations are defined for each layer regardless of the layer type. For a two layer model, it is necessary to define a layer elevation array for the top of layer 1 (the ground surface), the bottom of layer 1, and the bottom of layer 2. It is assumed that the top of layer 2 is equal to the bottom of layer 1.

One way to define layer elevations is to import a set of scatter points defining the elevations and interpolate the elevations directly to the layer arrays. This can be done with scatter points as well as rasters. In this case, the user will use a raster for the top of the grid and scatter points for the elevations of the bottom of layer 1 and the bottom of layer 2.

Layer interpolation is covered in depth in the “MODFLOW – Interpolating Layer Data” tutorial.

12.1 Delete the Layer Elevations Coverage

The model that the user started with included a coverage that defined a constant top and bottom elevation for the entire model area. It is no longer desirable to use that method for defining layer elevations. Instead, the user will interpolate layers from scatter points to create a more realistic, varying terrain. It is therefore necessary to delete the Layer Elevations coverage.

1. Right-click on the the “Layer Elevations” coverage.
2. Select the **Delete** command.

12.2 Interpolating the Heads and Elevations

Next, the user will interpolate the ground surface elevations and starting heads to the MODFLOW grid.


1. Right-click on the “elev_10.tif”  raster.
2. Select the *Interpolate To* / **MODFLOW Layers** menu command.

This is the dialog that allows the user to tell GMS which datasets to interpolate to which MODFLOW arrays. The dialog is explained fully in the “MODFLOW – Interpolating Layer Data” tutorial.

3. In the *Interpolate to MODFLOW Layers* dialog, highlight the “elev_10.tif” raster and the “Starting Heads 1” array, and click the **Map** button.
4. Highlight the “elev_10.tif” raster and the “Starting Heads 2” array, and click the **Map** button.
5. Highlight the “elev_10.tif” raster and the “Top Elevations Layer 1” array, and click the **Map** button.
6. Select the **OK** button to perform the interpolation.

12.3 Interpolating the Layer Elevations

Do as follows to interpolate the layer elevations for the bottom of layers 1 and 2:



1. In the Project Explorer, expand the “2D Scatter Data”  folder.
2. Right-click on the “elevs” scatter set.
3. Select the *Interpolate To* / **MODFLOW Layers** command.

GMS automatically mapped the *Bottom Elevations Layer 1* and *Bottom Elevations Layer 2* arrays to the appropriate data sets based on the data set name.

4. Select the **OK** button to exit the *Interpolate to MODFLOW Layers* dialog.




12.4 Adjusting the Display

Now that the interpolation is finished, the user can hide the scatter point sets and the grid frame.

1. Turn off the “grid frame” .
2. Turn off the “GIS Layers”  folder.

12.5 Viewing the Model Cross Sections

To check the interpolation, the user will view a cross section.

1. Select the “3D Grid Data”  folder in the Project Explorer.
2. Using the **Select Cell**  tool, select a cell somewhere near the center of the model.
3. Select the **Side View**  button.

The user may wish to use the arrow buttons in the *Tool Palette* to view different columns in the grid.

Note that on the right side of the cross section, the bottom layer pinches out and the bottom elevations are greater than the top elevations. This must be fixed before running the model.

12.6 Fixing the Elevation Arrays

GMS provides a convenient set of tools for fixing layer array problems. These tools are located in the Model Checker and are explained fully in the “MODFLOW – Interpolating Layer Data” tutorial.




1. Select the *MODFLOW* / **Check Simulation** command.
2. Select the **Run Check** button.
3. Select the **Fix Layer Errors** button at the right of the dialog.

Notice that many errors were found for layer 2. There are several ways to fix these errors. This tutorial will use the *Truncate to bedrock* option. This option makes all cells below the bottom layer inactive.


4. In the *Fix Layer Errors* dialog, select the *Truncate to bedrock* option.
5. Select the **Fix Affected Layers** button.

6. Select the **OK** button to exit the *Fix Layer Errors* dialog.
7. Select the **Done** button to exit the *Model Checker* dialog.

Notice that the layer errors have been fixed. Another way to view the layer corrections is in plan view.


8. Select the **Plan View**  button.
9. In the *Ortho Grid Toolbar*,  select the up arrow  to view the second layer.

Notice that the cells at the upper (northern) edge of the model in layer 2 are inactive.

10. Switch back to the top layer by selecting the down arrow .

13 Converting the Conceptual Model

It is now possible to convert the conceptual model from the feature object-based definition to a grid-based MODFLOW numerical model.

1. Right-click on the “East Texas”  conceptual model.
2. Select the *Map To / MODFLOW / MODPATH* command.
3. In the *Map → Model* dialog, make sure the *All applicable coverages* option is selected.
4. Select **OK**.

Notice that the cells underlying the drains, wells, and specified head boundaries were all identified and assigned the appropriate sources/sinks. The heads and elevations of the cells were determined by linearly interpolating along the specified head and drain arcs. The conductances of the drain cells were determined by computing the length of the drain arc overlapped by each cell and multiplying that length by the conductance value assigned to the arc. In addition, the recharge and hydraulic conductivity values were assigned to the appropriate cells.

14 Checking the Simulation

At this point, the MODFLOW data has been completely defined, and it is possible to run the simulation. First run the Model Checker to see if GMS can identify any mistakes that may have been made.

1. Select the “3D Grid Data”  folder in the Project Explorer.
2. Select the *MODFLOW | Check Simulation* command.

3. Select the **Run Check** button. There should be no errors.
4. Select the **Done** button to exit the *Model Checker* dialog.

15 Saving the Project

Now the user is ready to save the project and run MODFLOW.

1. Select the **Save**  button.



Note: Saving the project not only saves the MODFLOW files but it saves all data associated with the project including the feature objects and scatter points.

16 Running MODFLOW

It is now possible to run MODFLOW.



1. Select the *MODFLOW* / **Run MODFLOW** command. At this point, MODFLOW is launched and the *Model Wrapper* appears.
2. When the solution is finished, select the **Close** button.


A set of contours should appear. To view the contours for the second layer, do as follows:

3. Select the up arrow  in the *Ortho Grid Toolbar*.
4. After viewing the contours, return to the top layer by selecting the down arrow .

17 Viewing the Water Table in Side View

Another interesting way to view a solution is in side view.

1. Select the **Select Cell**  tool.
2. Select a cell somewhere near the well on the right side of the model.
3. Select the **Side View**  button.

Notice that the computed head values are used to plot a water table profile. Use the arrow buttons  in the *Mini-grid Toolbar* to move back and forth through the grid. The user should see a cone of depression at the well. When finished, do as follows:

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

18 Viewing the Flow Budget


The MODFLOW solution consists of both a head file and a cell-by-cell flow (CCF) file. GMS can use the CCF file to display flow budget values. For example, the user may want to know if any water exited from the drains. This can be accomplished simply by clicking on a drain arc.

1. Select the “Map Data”  folder in the Project Explorer.
2. Choose the **Select Arcs**  tool.
3. Click on the rightmost drain arc.

Notice that the total flow through the arc is displayed in the strip at the bottom of the window. Next, the user will view the flow to the river.

4. Click on one of the specified head arcs at the bottom and view the flow.
5. Hold down the *Shift* key and select each of the specified head arcs.

Notice that the total flow is shown for all selected arcs. Flow for a set of selected cells can be displayed as follows:

6. Select the “3D Grid Data”  folder in the Project Explorer.
7. Select a group of cells by dragging a box around the cells.
8. Select the *MODFLOW* | **Flow Budget** command.

The *Flow Budget* dialog shows a comprehensive flow budget for the selected cells.

9. Select **OK** to exit the dialog.
10. Click anywhere outside the model to unselect the cells.

19 Conclusion

This concludes the “MODFLOW – Conceptual Model Approach II” tutorial. Here are the main points from this tutorial:

- It is possible to import shapefiles and convert them to feature objects for use in the conceptual model.
- Well screens can be used to automatically locate the correct a 3D grid layer in which the wells are located.
- The MNW2 package more accurately models wells that are screened across multiple layers.

- Elevations for boundary conditions, such as drains, can be specified using a raster.
- It is possible to specify things like layer elevations and hydraulic conductivities using polygons in the conceptual model, but that will result in stair-step-like changes. For smoother transitions, it is possible to use 2D scatter points and interpolation.