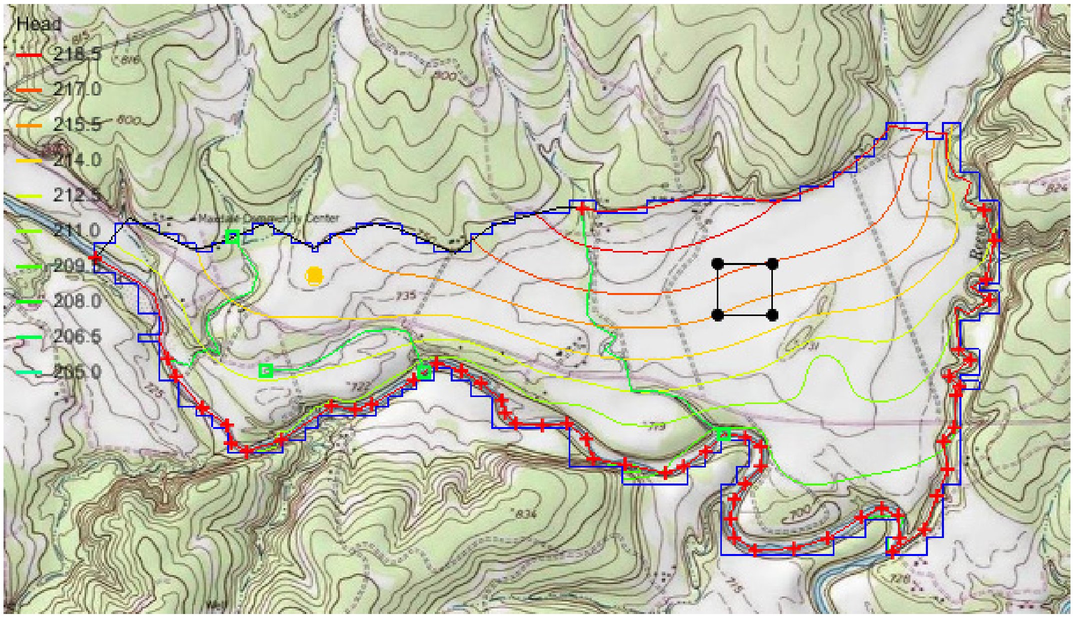
GMS 9.2 Tutorial

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

- Interpolating Layer Elevations Tutorial

Required Components

- Grid
- Geostatistics
- Map
- MODFLOW
- Subsurface Char Module

Time

- 40-60 minutes

1 Contents

2 Introduction

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

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

2.1 Outline

This is what you will do:

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 TIN surface.
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.

3 Description of Problem

The problem we will be solving for this tutorial is illustrated in Figure 1. The site is located in East Texas. We will assume that we are 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.

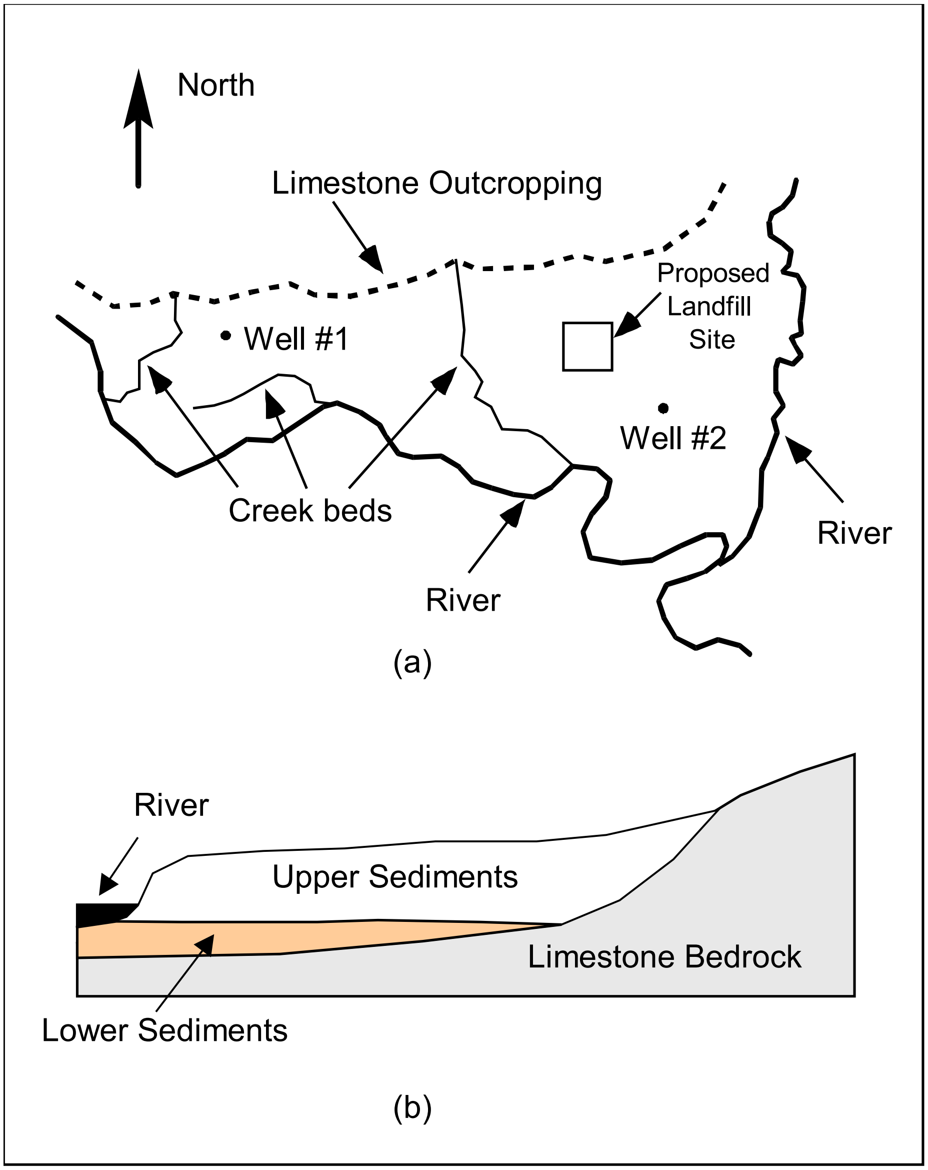


Figure 1. Site to be Modeled in This Tutorial. (a) Plan View of Site. (b) Typical North-South Cross Section Through Site.

We 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. We will assume the influx to the system is primarily through recharge due to rainfall. There are some creek beds in the area which are sometimes dry but occasionally flow due to influx from the groundwater. We 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.

4 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.

5 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\sample1**.
3. Open the file entitled **modfmap.gpr**.

6 Saving the Project

Before we make any changes, lets save the project under a new name.

1. Select the *File* | *Save As* command.
2. Save the project with the name **modfmap.**

Now you can hit the save button periodically as you develop your model.

7 Redefining the Recharge

We 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.

7.1 Creating the Landfill Boundary

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

1. Click on *GIS Layers* in the *Project Explorer*.
2. Select the *GIS | Add Shapefile Data* menu command.
3. Choose **tutfiles\MODFLOW\modfmap\landfill_arcs.shp** and click *Open.*

Now that the shapefile is imported, we will convert this shape file into feature objects in the **Recharge** coverage.

4. Make the **Recharge** coverage the active one by selecting it in the *Project Explorer.*
5. Select *landfill_arcs.shp* in the *Project Explorer*.
6. Select the *GIS | Shapes->Feature Objects* command
7. Select the *Yes* button to use map all visible shapes.
8. Select the *Next* button.
9. In the *GIS to Feature Objects Wizard* dialog, select *Elevation* for the ARC_ELEV.
10. Select *Next* then *Finish* to complete the process.

The landfill is now created.

7.2 Rebuilding the Polygons

Now that the landfill boundary is defined, we need to rebuild the polygons.

1. Select the *Build Polygons* macro.

7.3 Assigning the Recharge Values

Now that the recharge zones are redefined, we can assign the recharge values for the landfill polygon.

1. Select the *Select Polygons* tool.
2. Double click on the landfill polygon.
3. 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.

4. Select the *OK* button.

8 Redefining the Hydraulic Conductivity

In the previous tutorial, we only simulate a one layer model. In this tutorial, we'll make a two layer model. Thus, we need to define the hydraulic conductivity for the second layer and the vertical anisotropy for both layers. Similar to the previous tutorial, we will also use constant values for the second layer.

8.1 Turning on Vertical Anisotropy

1. Right-click on the **Layer 1** coverage and select the *Coverage Setup* command from the pop-up menu. From the *Areal Properties* list, turn **on** *Vertical anis*.
2. Click *OK*.
3. Double click on the polygon in the Layer 1 coverage.
4. Change the *Vertical anis.* to **4**.
5. Select the *OK* button.

8.2 Copying the Layer 1 Coverage

We'll create our layer 2 coverage by copying the layer 1 coverage.

1. Right-click on the **Layer 1** coverage and select the *Duplicate* command from the pop-up menu.
2. Change the name of the new coverage to **Layer 2**.
3. Right click on the **Layer 2** coverage and select the *Coverage Setup* command.
4. Change the *Default layer range* to go from **2** to **2**.
5. Select the *OK* button.

8.3 Specify the Coverage Attributes

For the new layer:

1. Select the **Layer 2** coverage in the *Project Explorer*.
2. Select the *Feature Objects | Build Polygons* command.

3. Double click on the polygon
4. Change the *Horizonal K* to **10**.
5. Select the *OK* button.

9 Modifying the Local Source/Sink Coverage

Since our model will have two layers, when we map the conceptual model to the grid, we will need to specify which grid layer we want the wells to 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 that we know how many grid layers we will have and where they will be when we build the conceptual model. Also if we decide to add or subtract grid layers later, we'll have to remember to change our conceptual model.

The second way is to use a well screen with the WEL package. This allows us 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 in the appropriate grid layer (or layers) automatically 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. We'll use this option in this tutorial.

We'll also modify our conceptual model so that it uses real terrain data for elevations. The terrain elevations are imported and converted to a TIN surface. We'll use the TIN to define the drain elevations. When the drain arcs are discretized onto the model grid, the cells that intersect the arcs are found. Then the drain elevation is interpolated from the TIN to the cell centers.

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

9.1 Turn on MNW2

We need to make the MNW2 properties available in the coverage.

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

9.2 Modifying the existing well

We'll 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. Change the *Type* to **well (MNW2)**.
6. Turn on the **Vertical Boreline** option.
7. Click on the **Boreline** button.
8. 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.

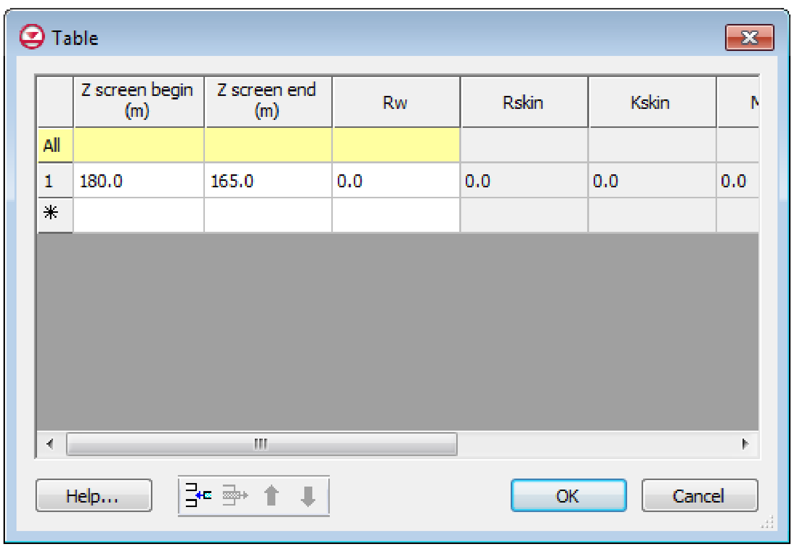


Figure 2. Defining the screened interval.

9. Click *OK.*
10. In the Properties dialog, change the *LOSSTYPE* to **THIEM**.
11. Change *Rw* to **0.05**.
12. Click *OK.*

9.3 Creating the TIN boundary

Next, we use elevation data to define the drain elevation. Elevation data has already been provided for this area. The elevation data can be obtained from many sources such as the National Elevation Dataset (NED), boreholes, etc. First, we need to create a TIN boundary from the conceptual model.

1. Right-click on the **Boundary** coverage and select the *Duplicate* command from the pop-up menu. Change the new coverage name to **TINs**.
2. Select the *Select Arcs* tool.
3. Select the boundary arc.
4. Select the *Feature Objects* | *Redistribute Vertices* command.
5. Select the *Specified Spacing* option.
6. Enter a value of **120** for the spacing.
7. Select the *OK* button.
8. Select the *Build Polygons* macro.
9. Select the *Map->TIN* macro.
10. Select the *OK* button.

You should now see a TIN.

9.4 Interpolating elevation data to TIN

Next, we need to interpolate the elevation data to this TIN.

1. In the *Project Explorer*, expand the *2D Scatter Data* folder.
2. Make the **terrain** scatter point set the active one by selecting it in the *Project Explorer*.
3. Select the *Interpolation* | *Interpolate -> Active TIN* command.
4. Change the name of the data set to **elevation.**
5. Select the *OK* button.

9.5 Modifying the existing TIN

We'll have the drain elevations 3 m below the surface area.

1. Select the **elevation** data set below the **tin (1)**.

2. Select the *Data Calculator* tool.

The currently available data sets are listed in the top of the dialog. Each data set is numbered from "d1" to "dn". Data sets are referenced in the mathematical expression using the "d" numbers. The **elevation** data set should be numbered **d2**.

3. In the *Expression* field, enter **d2-3**.

4. In the *Result* field, enter **drainelev**.

5. Select the *Compute* button.

6. Select the *Done* button exit the dialog.

9.6 Assigning the drain elevation

We are now ready 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. Change the *BC type* to *drain.*

6. In the first row under *Bot. elev.*, click on the drop down box and select *TIN.*

7. In the new window, select the **tin (1).**

8. Select the *OK* button.

GMS will get the elevation from the **drainelev** data and assign it to all drain nodes.

9. Select the OK button exit the dialog.

10 Recreating the Grid

We are now ready 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 *Z-Dimension* change *Number cells* to **2**.
4. Select the OK button.

11 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, we must first initialize the MODFLOW data:

1. Right click on the *grid* item in the *Project Explorer* and select the *New MODFLOW* command.
2. Select the *OK* button.

12 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 *Properties* button.
6. Confirm that the layer assignment is 1 to 2 and click *OK*.
7. 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 cells on the perimeter.

13 Interpolating Layer Elevations

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

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. In some cases, this is done using one set of scatter points. In this case, we will use two scatter point sets: one for the ground surface and one for the elevations of the bottom of layer one and the bottom of layer two. It is often convenient to use two scatter point sets in this fashion due to the source of the points. Ground surface points are often digitized from a map while layer elevations typically come from borehole data. In this case, the ground surface points are obtained from the National Elevation Dataset (NED) using the *Add Online Maps* tool available in GMS.

Layer interpolation is covered in depth in the *Interpolating Layer Data* tutorial.

13.1 Delete the Layer Elevations Coverage

The model we started with included a coverage which defined a constant top and bottom elevation for the entire model area. We no longer want to use that method for defining layer elevations. Instead we will interpolate layer from scatter points to create a more realistic, varying terrain. We'll delete the Layer Elevations coverage.

1. Delete the *Layer Elevations* coverage.

13.2 Importing the Ground Surface Scatter Points

The scatter points have already been read in because they were included in the project file that we read in the beginning. These points came from importing a text file as described in the *2D Geostatistics* tutorial. The scatter sets are hidden so we will unhide them so you can see them.

1. In the *Project Explorer*, expand the *2D Scatter Data* folder.
2. In the *Project Explorer*, check the boxes next to the two scatter point sets named *terrain* and *elevs*.
3. Make the **terrain** scatter point set the active one by selecting it in the *Project Explorer*.

A set of scatter point symbols should appear on the model.

13.3 Interpolating the Heads and Elevations

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

1. Right-click on the *terrain* scatter set and select the *Interpolate To | MODFLOW Layers* menu command.

This is the dialog that allows you to tell GMS which data sets you want to interpolate to which MODFLOW arrays. The dialog is explained fully in the *Interpolating Layer Data* tutorial.

2. Highlight the **ground_elev** data set and the **Starting Heads 1** array, and click the *Map* button.
3. Highlight the **ground_elev** data set and the **Starting Heads 2** array, and click the *Map* button.
4. Highlight the **ground_elev** data set and the **Top Elevations Layer 1** array, and click the *Map* button.
5. Select the *OK* button to perform the interpolation.
6. Turn **off** the *terrain* scatter set .

13.4 Interpolating the Layer Elevations

To interpolate the layer elevations:

1. Select the *elevs* scatter set to make it active.
2. Right-click on the *elevs* scatter set and 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.

3. Select the *OK* button

13.5 Adjusting the Display

Now that we are finished with the interpolation, we can hide the scatter point sets and the grid frame.

1. Turn **off** the scatter point sets .
2. Turn **off** the grid frame .
3. Turn **off** the TIN .

4. Turn **off** the GIS Layers.

13.6 Viewing the Model Cross Sections

To check the interpolation, we 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.

You 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.

13.7 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 *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 two. There are several ways to fix these layers. We will use the *Truncate to bedrock* option. This option makes all cells below the bottom layer inactive.

4. 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*.

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* tool bar, select the up arrow to view the second layer.

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

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

14 Converting the Conceptual Model

We are now ready 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 and select the *Map To | MODFLOW / MODPATH* command.
2. Make sure the *All applicable coverages* option is selected and 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.

15 Checking the Simulation

At this point, we have completely defined the MODFLOW data and we are ready to run the simulation. Let's run the *Model Checker* to see if GMS can identify any mistakes we may have 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*.

16 Saving the Project

Now we are 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.

17 Running MODFLOW

We are now ready 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:

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 .

18 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 display to move back and forth through the grid. You should see a cone of depression at the well. When finished:

4. Select the *Plan View* button .

19 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, we 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, we 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.

This 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.

20 Conclusion

This concludes the *MODFLOW - Conceptual Model Approach II* tutorial. Here are the things that you should have learned in this tutorial:

- You can import shapefiles and convert them to feature objects for use in your conceptual model.

- Well screens can be used to automatically locate the correct 3D grid layer in which the wells are located.

- Elevations for boundary conditions, such as drains, can be specified using a TIN surface.

- You can 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, you can use 2D scatter points and interpolation.