GMS 10.1 Tutorial
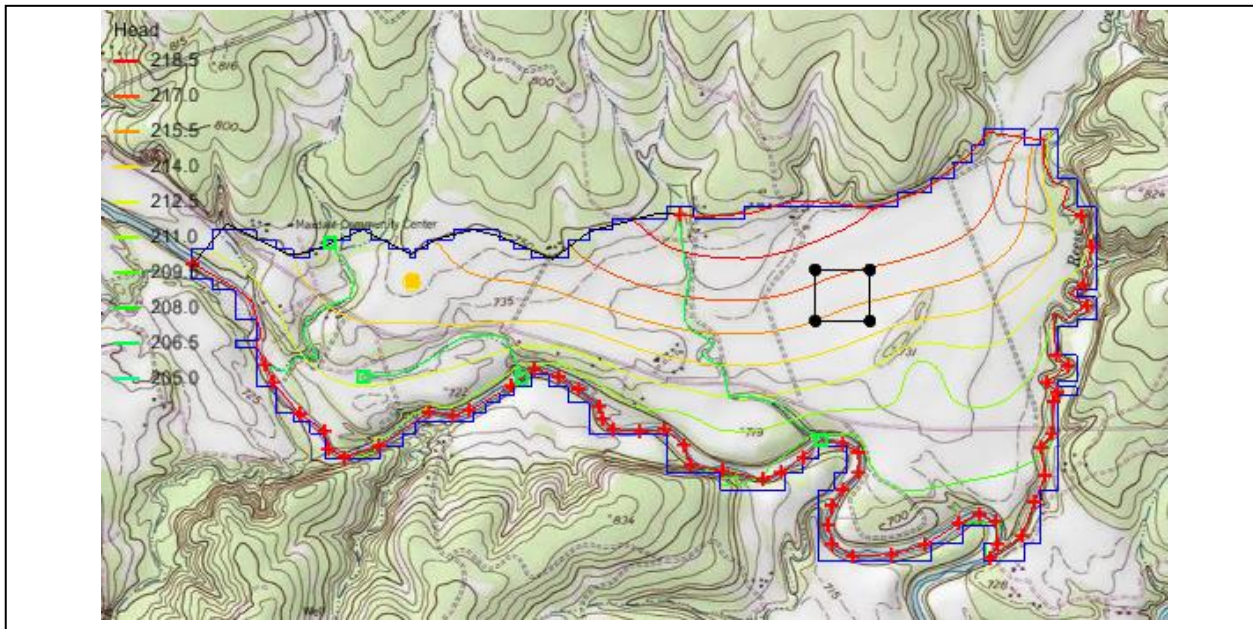# MODFLOW – Conceptual Model Approach II
Build a multi-layer MODFLOW model using advanced conceptual model techniques

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

This tutorial builds on the "MODFLOW – Conceptual Model Approach I" tutorial. In that tutorial, a one-layer model using the conceptual model approach was built. The top and bottom elevations were all the same, meaning the model was 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. One of the wells will be switched to use the MNW2 package with a well screen that partially penetrates both layers. An imported shapefile will be used to define multiple recharge polygons to simulate a landfill.

The problem to be solved in this tutorial is illustrated in   Figure 1. The site is located in East Texas. This tutorial 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.
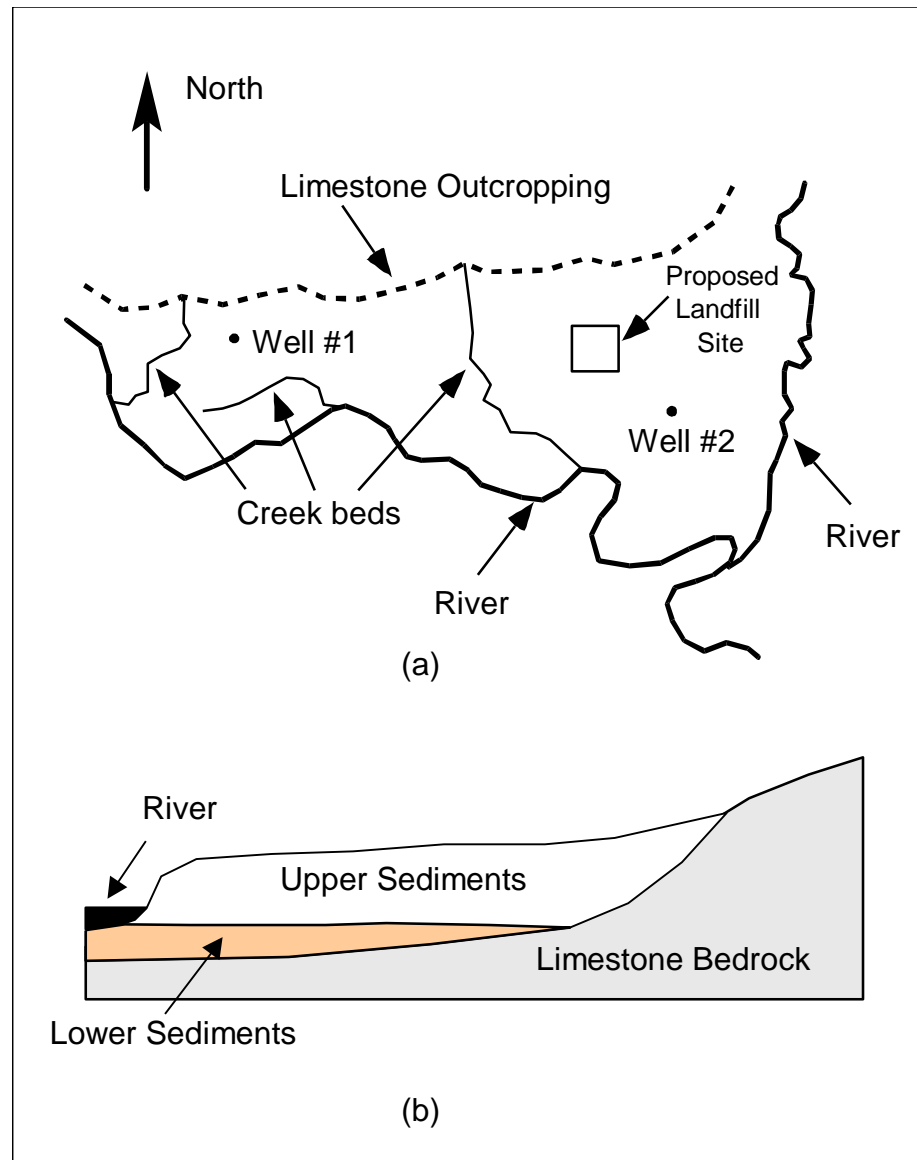
Figure 1   Site to be modeled in this tutorial. (a) Plan view of site. (b) Typical north-south cross section through site

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. 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.

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.

This tutorial will discuss and demonstrate defining a recharge polygon by importing a shapefile and converting it to feature objects, defining well screens in the conceptual model, assigning drain elevations using a raster, creating a two layer grid from the conceptual model, interpolating scatter points to MODFLOW layer data, mapping the conceptual model to MODFLOW, checking the simulation and running MODFLOW, and viewing the results.

1.1   Getting Started

Do the following to get started:

1. If necessary, launch GMS.

2. If GMS is already running, select *File* / **New** to ensure that the program settings are restored to their default state.

2   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. Click **Open** 📂 to bring up the *Open* dialog.

2. Select "Project Files (*.gpr)" from the *Files of type* drop-down.

3. Browse to the *Tutorials\MODFLOW\modfmap\sample1* directory and select "modfmap.gpr"

4. Click **Open** to import the project and close the *Open* dialog. The Main Graphics Window will appear as in Figure 2.
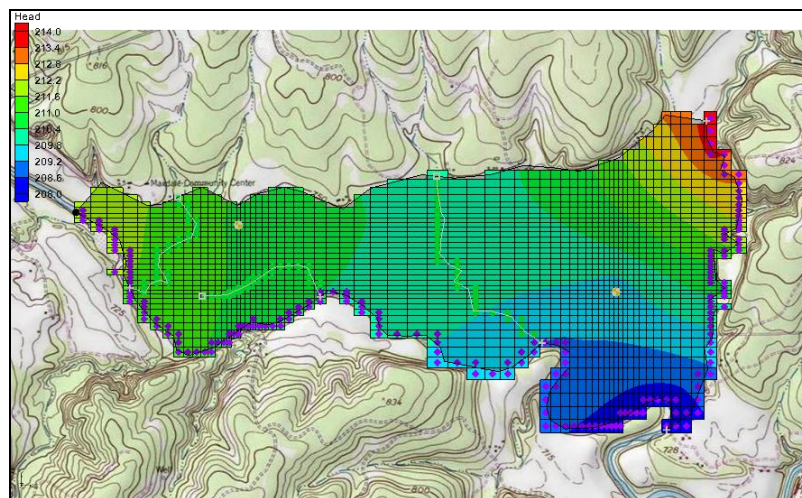

Figure 2   After importing the project

3   Saving the Project

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

1. Select *File* | **Save As…** to bring up the *Save As* dialog.

2. Select "Project Files (*.gpr)" from the *Save as type* drop-down.

3. Enter "modfmapII.gpr" as the *File name*.

4. Click **Save** to save the file under the new name and close the *Save As* dialog.

Be sure to use the **Save** 💾 macro periodically throughout the tutorial.

4   Redefining the Recharge

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.

4.1   Creating the Landfill Boundary and Polygon

The first step is to create the arc delineating the boundary of the landfill. For this tutorial, import the boundary from an existing landfill shapefile.

1. Click on "GIS Layers" 🌐 folder in the Project Explorer to make it active.

2. Select *GIS* | **Add Shapefile Data…** to bring up the *Select shapefile* dialog.

3. Select "Shapefile (*.shp)" from the *Files of type* drop-down.

4. Browse to the *Tutorials\MODFLOW\modfmap* directory and select "landfill_arcs.shp".

5. Click **Open** to import the shapefile and close the *Select shapefile* dialog.

Now that the shapefile is imported, convert it into feature objects in the "🍂 Recharge" coverage.

6. Expand the "Map Data" folder, including the "🐢 East Texas" conceptual model within it.

7. Select the "🍂 Recharge" coverage to make it active.

8. Select "landfill_arcs.shp" under "GIS Layers" in the Project Explorer to make it active.

9. Select *GIS* | **Shapes → Feature Objects** and click **Yes** when asked if all visible shapefiles should be used for mapping. The *GIS to Feature Objects Wizard* dialog will appear.

10. Click **Next** to go to the *Step 1 of 2* page of the *GIS to Feature Objects Wizard* dialog.

11. On the Mapping row of the Mapping preview section, select "Elevation" from the drop-down in the ARC_ELEV column.

12. Click **Next** to go to the *Step 2 of 2* page of the *GIS to Feature Objects Wizard* dialog.

13. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.

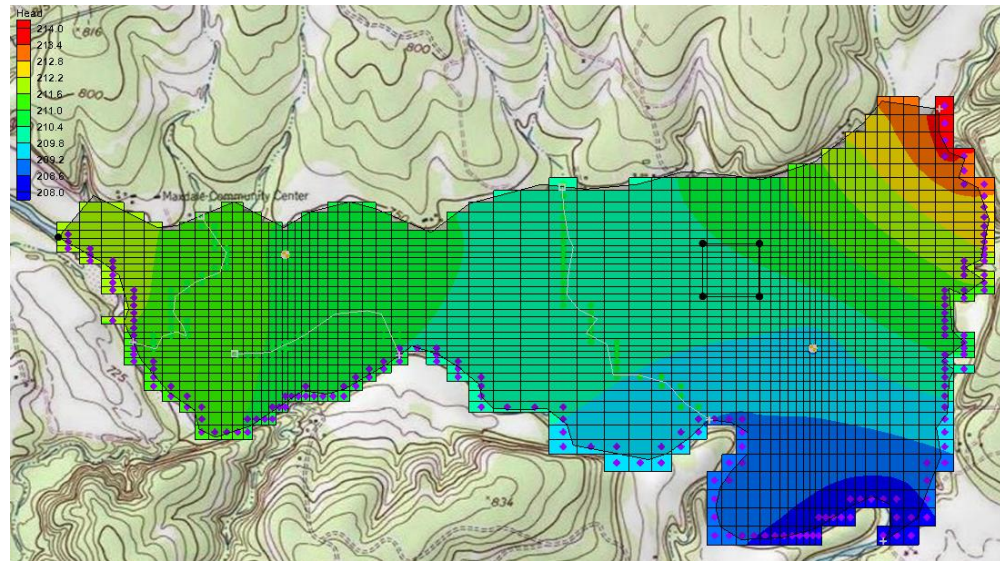The landfill is now created and visible in the Graphics Window (Figure 3).

Figure 3   The landfill is outlined as a black square

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

14. Click the **Build Polygons** macro.

4.2   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 to make it active.

2. Using the **Select Polygons** tool, double-click on the landfill polygon to bring up the *Attribute Table* dialog.

3. Enter "0.00006" as the *Recharge rate*.

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. Click **OK** to close the *Attribute Table* dialog.

5   Redefining the Hydraulic Conductivity

In the previous tutorial, only a one-layer model was simulated. In this tutorial, a two-layer model will be made. 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, constant values will be used for the second layer.

5.1   Turning on Vertical Anisotropy

1. Right-click on the "⬛ Layer 1" coverage and select **Coverage Setup…** to bring up the *Coverage Setup* dialog.

2. In the *Areal Properties* column, turn on *Vertical anis*.

3. Click **OK** to close the *Coverage Setup* dialog.

4. Select "⬛ Layer 1" to make it active.

5. Double-click on the landfill polygon to bring up the *Attribute Table* dialog.

6. Enter "4.0" in the *Vertical anis.* column. This makes the vertical hydraulic conductivity one fourth of the horizontal conductivity.

7. Click **OK** to close the *Attribute Table* dialog.

5.2   Creating the Layer 2 Coverage

Create the "Layer 2" coverage by copying the "Layer 1" coverage.

1. Right-click on the "⬛ Layer 1" coverage and select **Duplicate…** to create a new "⬛ Copy of Layer 1" coverage.

2. Right-click on "⬛ Copy of Layer 1" and select **Coverage Setup…** to bring up the *Coverage Setup* dialog.

3. Enter "Layer 2" as the *Coverage name*.

4. Below the three columns, set the *Default layer range* to go from "2" to "2".

5. Click **OK** to close the *Coverage Setup* dialog.

5.3   Specify the Coverage Attributes

For the new layer, do the following:

1. Select the "⬛ Layer 2" coverage in the Project Explorer to make it active.

2. Using the **Select Polygons** 🔲 tool, double-click on the landfill polygon to bring up the *Attribute Table* dialog.

3. Enter "10.0" for the *Horizonal K*.

4. Click **OK** to close the *Attribute Table* dialog.

6   Modifying the Local Source/Sink Coverage

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

The simplest way is to specify the grid layer in the conceptual model, but that requires knowing how many grid layers will be in the project and where they will be when building the conceptual model. The conceptual model would also need to be changed if grid layers were later added or subtracted.

The second way is to use a well screen with the WEL package. This allows for specifying 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 conceptual model will be modified so that it uses real terrain data for elevations. The terrain elevations will come from a raster. The raster will also be used 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 the elevation for each drain at the arc nodes would otherwise have to be manually determined and entered.

6.1   Turn on MNW2

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

1. Right-click the "<img> Sources & Sinks" coverage in the Project Explorer and select **Coverage Setup…** to bring up the *Coverage Setup* dialog.

2. In the *Sources/Sinks/BCs* column, turn on *Wells (MNW2)*.

3. Click **OK** to close the *Coverage Setup* dialog.

6.2   Modifying the Existing Well

First edit the eastern well.

1. Select the "<img> Sources & Sinks" coverage in the Project Explorer to make it active.

2. Using the **Select Points\Nodes** <img> tool, aelect the well on the eastern (right) side of the model.

3. Click **Properties** <img> to bring up the *Attribute Table* dialog.

4. In the table, select "well (MNW2)" from the drop-down in the *Type* column.

5. Scroll to the right and enter "-300.0" in the *Qdes (m^3/d)* field.

6. Check the box in the *Vertical Boreline* column.

7. Click on the <img> button in the *Boreline* column to bring up the *Table* dialog.

8. In the row marked with a star *, enter "180.0" in the *Z screen begin* column and "165.0" in the *Z screen end* column.

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

9. In the row *1*, enter "0.05" in the *Rw* column.

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

11. Select "THIEM" from the drop-down in the *LOSSTYPE* column.

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

6.3   Importing the Raster

Next, it is possible to use imported elevation data to define the drain elevation. In this case, a raster with the necessary information will be imported.

1. Turn off "easttex.png" in the Project Explorer.

2. Click **Open** 📂 to bring up the *Open* dialog.

3. Select "Raster/DEM Files" from the *Files of type* drop-down.

4. Browse to the *Tutorials/MODFLOW/modfmap* directory and select "elev_10.tif".

5. Click **Open** to import the raster and close the *Open* dialog.

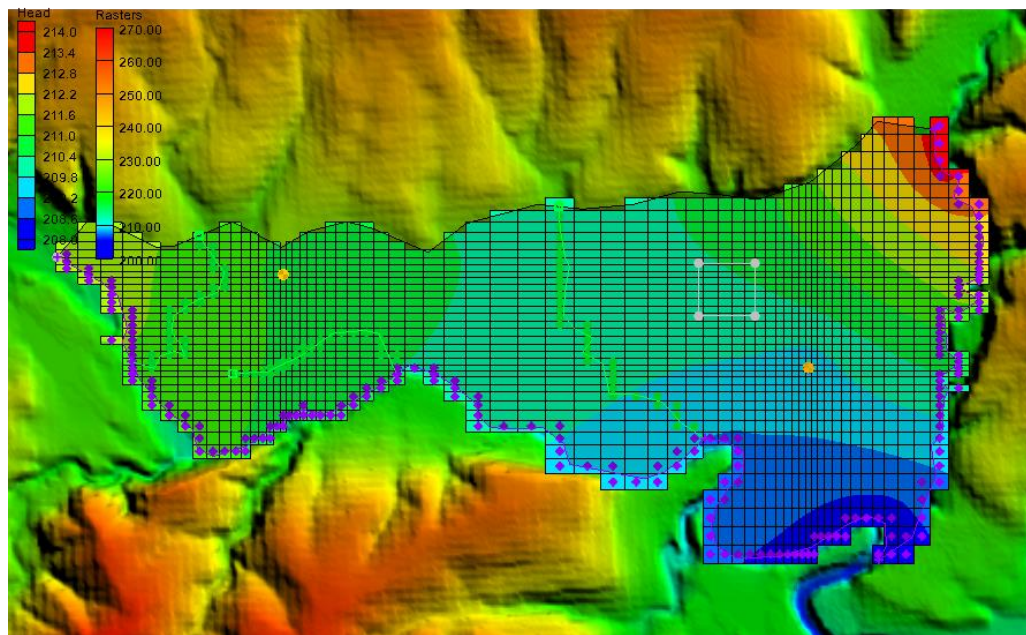An elevation raster will appear in the background of the project (Figure 4).

Figure 4   The elevations raster showing in the background

6.4   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 to make it active.

2. Click on the **Select Points/Nodes** 🖈 tool, then select *Edit* / **Select All**.

3. Click **Properties** 🗐 to bring up the *Attribute Table* dialog.

4. At the top right, select "drain" from the *BC type* drop-down.

5. In the *All* row, select "<Raster>" from the *Bot. Elev. (m)* column to bring up the *Select Raster* dialog.

6. Select "elev_10.tif" and click OK to close the *Select Raster* dialog.

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

7   Recreating the Grid

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

1. Select *Feature Objects* | **Map → 3D Grid** and click **OK** to confirm deletion of the existing grid.

2. Click **OK** to confirm deletion of all MODFLOW and MODPATH data and bring up the *Create Finite Difference Grid* dialog.

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* section, select *Number cells* and enter "2" in the field to the right.

4. Click **OK** to close the *Create Finite Difference Grid* dialog.
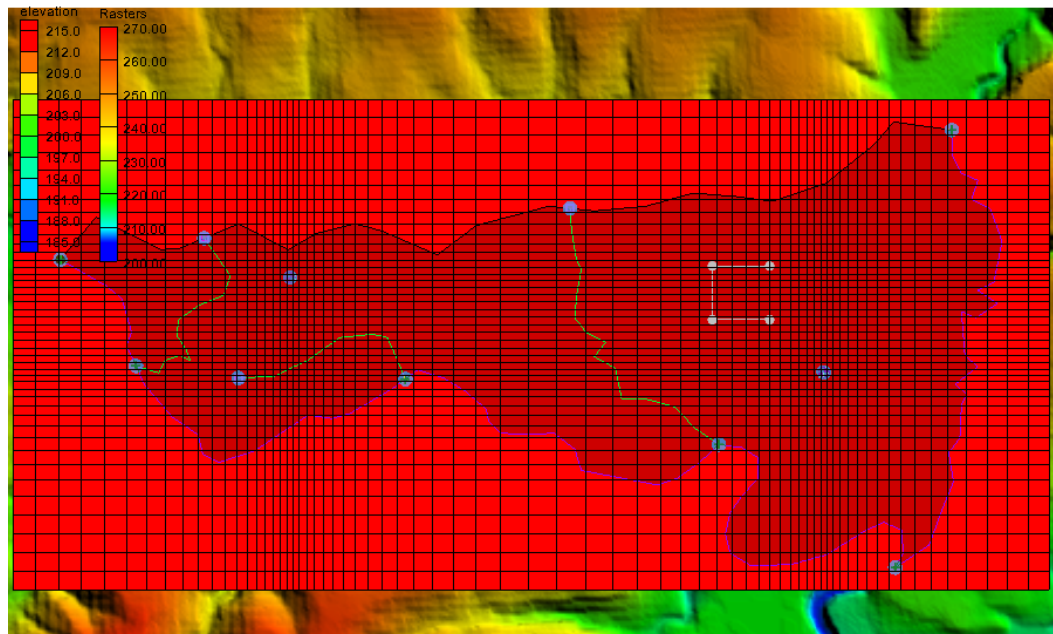
The new grid will appear in the Graphics Window (Figure 5).

Figure 5   The recreated grid

8   Initializing the MODFLOW Data

Now that the grid is constructed, the next step is to convert the conceptual model to a grid-based numerical model. Before doing this, the MODFLOW data must first be initialized:

1. Right-click on the " grid" in the Project Explorer and select **New MODFLOW…** to bring up the *MODFLOW Global/Basic Package* dialog.

2. Click **OK** to accept the defaults and close the *MODFLOW Global/Basic Package* dialog.

9   Defining the Active/Inactive Zones

Now that the grid has been created and MODFLOW has been initialized, 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 " Sources & Sinks" coverage to make it active.

2. Using the **Select Polygons** tool, select one of the three polygons.

3. Click **Properties** to bring up the *Attribute Table* dialog.

4. Scroll to the far right in the table and confirm that the layer assignment for the *From layer* is "1" and the *To layer* is "2".

5. Click **OK** to close the *Attribute Table* dialog.

Feel free to repeat steps 2-4 to verify the same information for the other two polygons.

6. Click anywhere outside the three polygons to unselect all of them.

7. Select *Feature Objects* | **Activate Cells in Coverage(s)**.

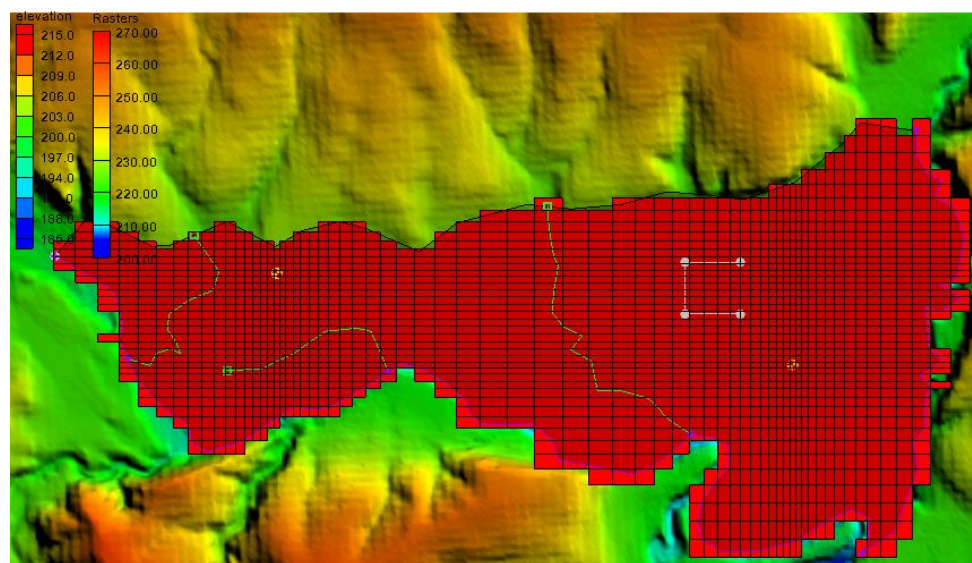The Graphics Window should appear similar to Figure 6.

Figure 6   Only the active zones are highlighted

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.

10   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, use a raster for the top of the gird 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.

10.1   Delete the Layer Elevations Coverage

The initial model 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, interpolating layers from scatter points will create a more realistic, varying terrain. It is therefore necessary to delete the "Layer Elevations" coverage.

- Right-click on the "Layer Elevations" coverage and select **Delete**.

10.2   Interpolating the Heads and Elevations

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

1. Right-click on the " elev_10.tif" raster and select *Interpolate To* | **MODFLOW Layers…** to bring up the *Interpolate to MODFLOW Layers* dialog.

This dialog tells GMS which datasets to interpolate to which MODFLOW arrays. The dialog is explained fully in the "MODFLOW – Interpolating Layer Data" tutorial.

2. Select "elev_10.tif" in the *Rasters* section and "Starting Heads 1" in the MODFLOW data section.

3. Click **Map** to add them to the *Dataset → MODFLOW data* section.

4. Select "elev_10.tif" in the *Rasters* section and "Starting Heads 2" in the MODFLOW data section.

5. Click **Map** to add them to the *Dataset → MODFLOW data* section.

6. Select "elev_10.tif" in the *Rasters* section and "Top Elevations Layer 1" in the MODFLOW data section.

7. Click **Map** to add them to the *Dataset → MODFLOW data* section.

8. Click **OK** to perform the interpolations and close the *Interpolate to MODFLOW Layers* dialog.

10.3   Interpolating the Layer Elevations

To interpolate the layer elevations for the bottom of layers 1 and 2, do the following:

1. In the Project Explorer, expand the " 2D Scatter Data" folder.

2. Right-click on the "elevs" scatter set and select *Interpolate To* | **MODFLOW Layers** to bring up the *Interpolate to MODFLOW Layers* dialog.

Notice that 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. No additional mappings need to be performed here.

3. Click **OK** to perform the interpolations and close the *Interpolate to MODFLOW Layers* dialog.

Now that the interpolation is finished, hide the scatter point sets, GIS layers, and the grid frame.

4. Turn off the " Grid Frame", the " GIS Layers" folder, and the " 2D Scatter Data" folder.

10.4   Viewing the Model Cross Sections

To check the interpolation, 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. Switch to **Side View** .

4. 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 (Figure 7). This must be fixed before running the model.
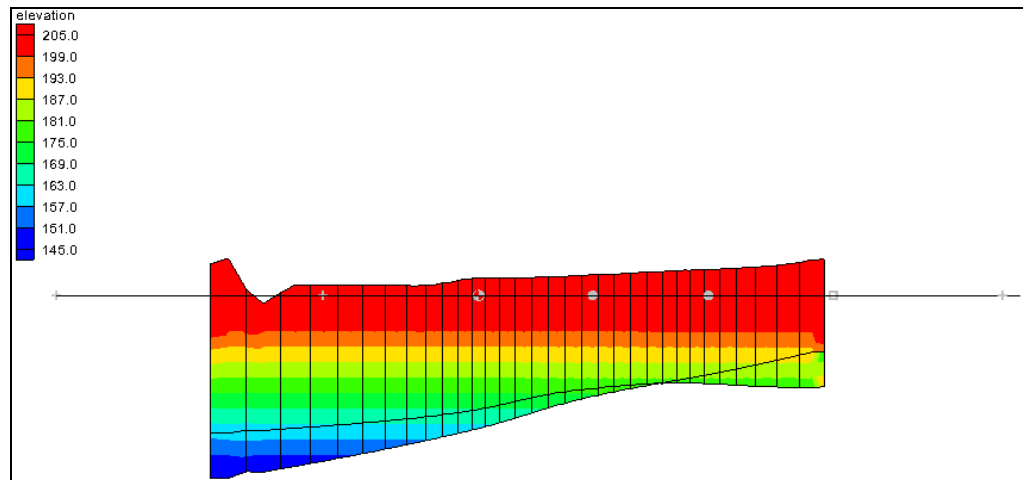
Figure 7   Cross section before elevations are corrected

10.5   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 *MODFLOW* / **Check Simulation…** to bring up the *Model Checker* dialog.

2. Click **Run Check** to check the model for errors.

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.

3. Click **Fix Layer Errors…** to bring up the *Fix Layer Errors* dialog.

4. In the *Correction method* section, select *Truncate to bedrock* and click **Fix Affected Layers**.

5. Click **OK** to exit the *Fix Layer Errors* dialog.

6. Click **Run Check** to check the model for errors.

Notice that all the errors have been fixed. The various warnings are acceptable for the purposes of this tutorial and can be ignored.

7. Click **Done** to exit the *Model Checker* dialog.

Another way to view the layer corrections is in plan view.

8. Switch to **Plan View** .

9. In the *Ortho Grid Toolbar* , click 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 (Figure 8).

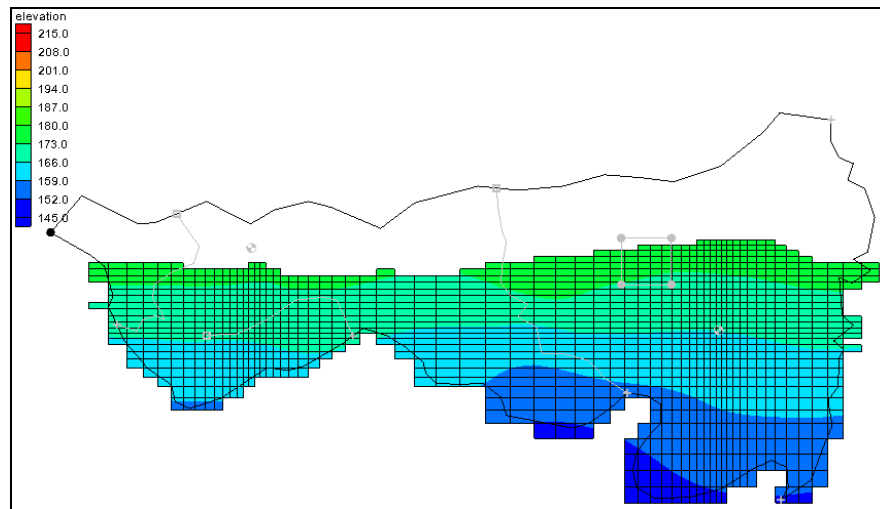10. Switch back to the top layer by clicking the down arrow .

Figure 8   Inactive cells in layer 2

11   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 and select *Map To | **MODFLOW / MODPATH*** to bring up the *Map → Model* dialog.

2. Select *All applicable coverages* and click **OK** to close the *Map → Model* dialog.

Notice that the cells underlying the drains, wells, and specified head boundaries were all identified and assigned the appropriate sources/sinks (Figure 9). 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.
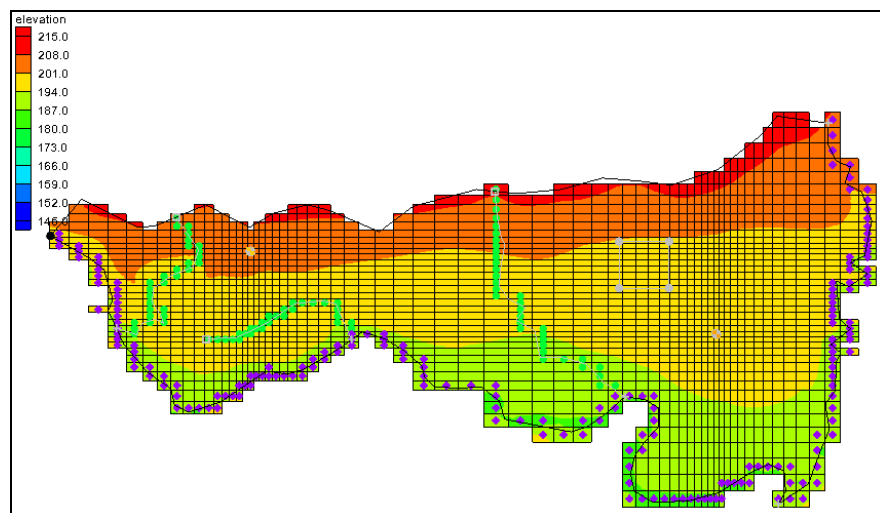

Figure 9   The model after conversion

12   Checking and Saving 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 to switch to the 3D Grid module.

2. Select *MODFLOW* | **Check Simulation…** to bring up the *Model Checker* dialog.

3. Click **Run Check**. There should be no errors.

4. Click **Done** to exit the *Model Checker* dialog.

5. Click **Save** .

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.

13   Running MODFLOW

It is now possible to run MODFLOW.

1. Select *MODFLOW* / **Run MODFLOW** to bring up the *MODFLOW* model wrapper dialog. The process should be completed quickly.

2. When the solution is completed, turn on *Read solution on exit* and *Turn on contours (if not on already)* and click **Close** to exit the *MODFLOW* model wrapper dialog.

The Graphics Window should appear similar to Figure. To view the contours for the second layer, do as follows:

3. Click the up arrow in the *Ortho Grid Toolbar*.

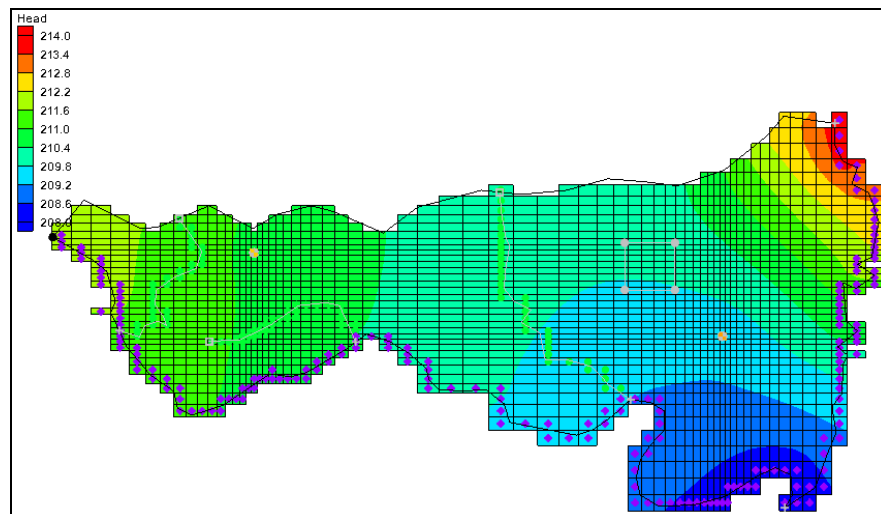4. After viewing the contours, return to the top layer by clicking the down arrow .

Figure 10   Contours of layer 1 after the model run

14   Viewing the Water Table in Side View

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

1.   Using the **Select Cell** 🔲 tool, select a cell somewhere near the well on the right side of the model.

2.   Switch to **Side View** 🔲.

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. A cone of depression should be seen at the well (Figure).

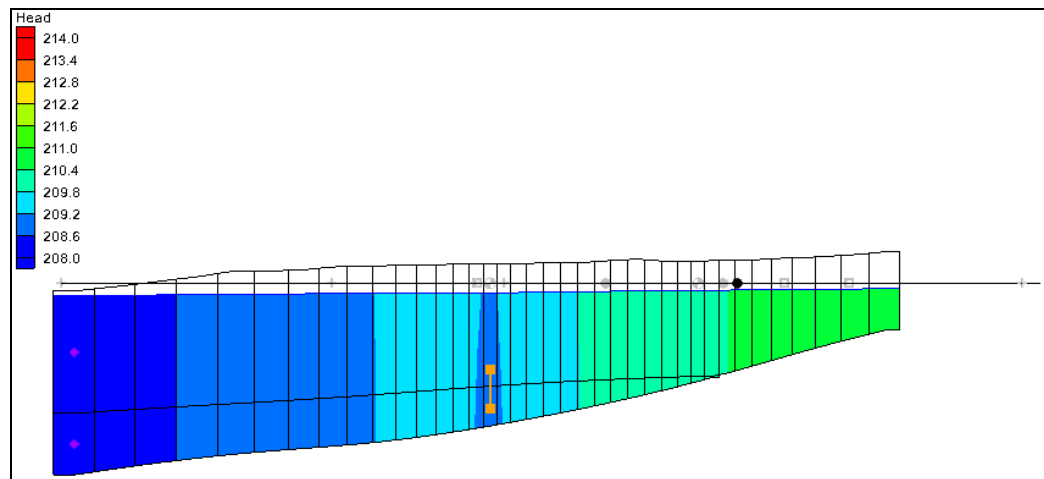3.   When finished reviewing the cells, witch to **Plan View** 🔲.

Figure 11   Side view of the model with well depression visible

15   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, to know if any water exited from the drains, do the following:

1.   Select the "🔘 Sources & Sinks" coverage to make it active.

2.   Using the **Select Arcs** 🔭 tool, select 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, view the flow to the river.

3.   Using the **Select Arcs** 🔭 tool, select one of the specified head arcs at the bottom and view the flow.

4.   Using the **Select Arcs** 🔭 tool while holding down the *Shift* key, select at least two of the specified head arcs.

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

5. Select the "3D Grid Data" folder in the Project Explorer to switch to the 3D Grid module.

6. Using the **Select Cell** tool, select a group of cells by dragging a box around the cells.

7. Select *MODFLOW* | **Flow Budget** to bring up the *Flow Budget* dialog.

This dialog shows a comprehensive flow budget for the selected cells.

8. Click **OK** to exit the *Flow Budget* dialog.

9. Click anywhere outside the model to unselect the cells.

16   Conclusion

This concludes the "MODFLOW – Conceptual Model Approach II" tutorial. The following topics were discussed and demonstrated:

- 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.