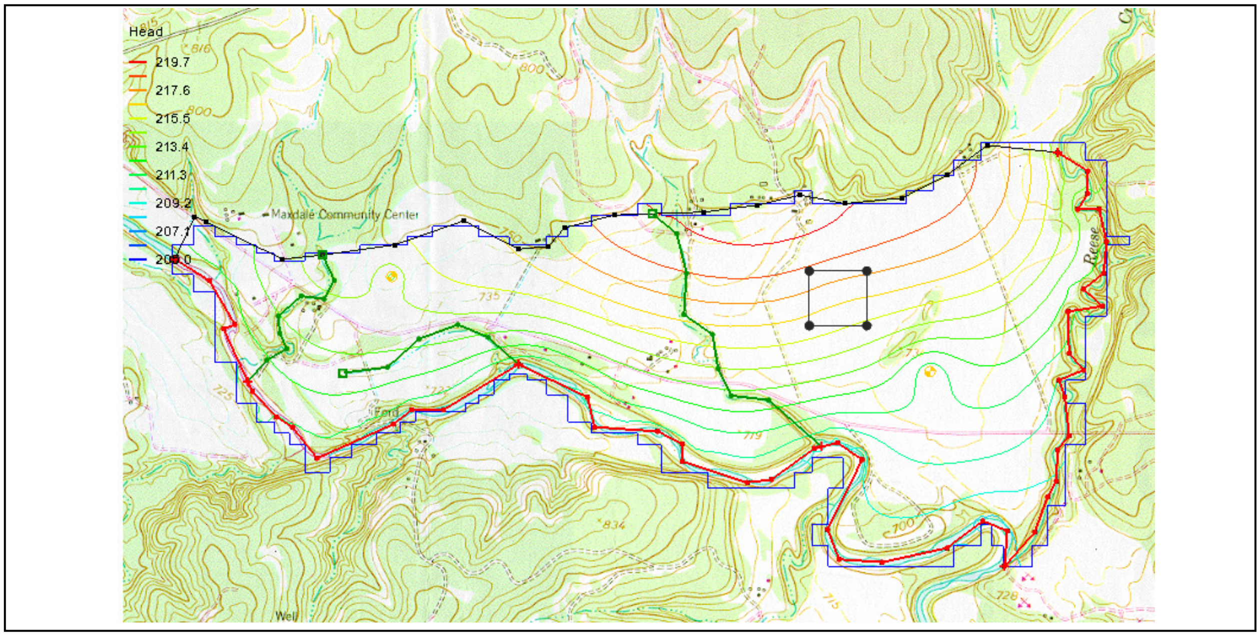
GMS

v. 8.2

GMS 8.2 Tutorial
MODFLOW – Conceptual Model Approach

Build a MODFLOW model using the conceptual model approach



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
- MODAEM Tutorial

Required Components

- Grid
- Geostatistics
- Map
- MODFLOW

Time

- 30-60 minutes

AQUAVEO™

1 Contents

2 Introduction

Two approaches can be used to construct a MODFLOW simulation in GMS: the grid approach or 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 steps involved in the grid approach are described in the tutorial entitled *MODFLOW - Grid Approach*. 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. Once this model is complete, the grid is generated and the conceptual model is converted to the grid model and all of the cell-by-cell assignments are performed automatically. The steps involved in performing a MODFLOW simulation using the conceptual model approach are described in this tutorial.

2.1 Outline

This is what you will do:

1. Import a background image.
2. Create and define coverages.
3. Map the coverages to a 3D grid.
4. Import scatter points and interpolate them to elevation.
5. Convert the conceptual model to MODFLOW.
6. Check the simulation and run MODFLOW.
7. 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.

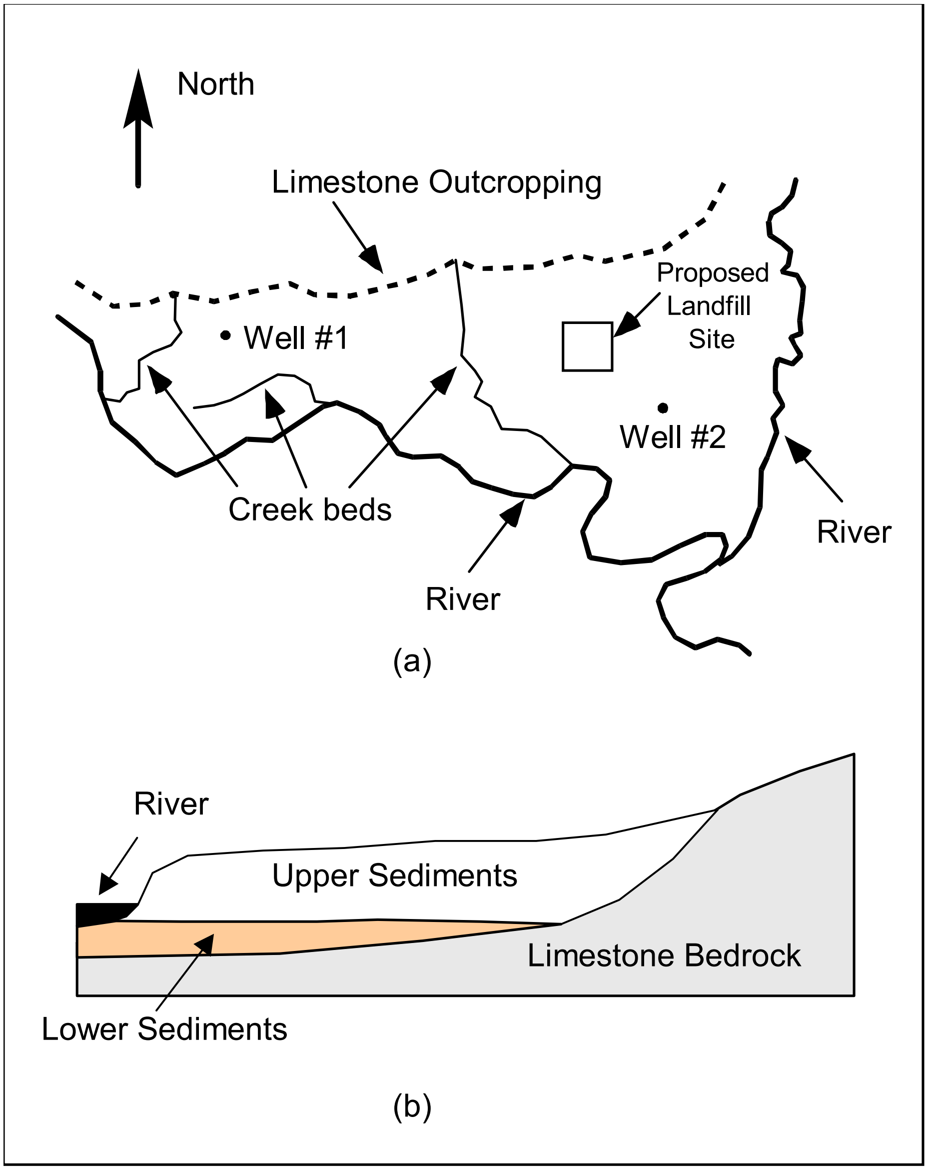


Figure 1. Site to be Modeled in This Tutorial. (a) Plan View of Site. (b) Typical North-South Cross Section Through Site.

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 Background Image

The first step in setting up the simulation is to import a digital image of the site being modeled. This image was created by scanning a portion of a USGS quadrangle map on a desktop scanner. The image was imported to GMS, registered, and a GMS project file was saved. To read in the image, we will open the project file. Once the image is imported to GMS, it can be displayed in the background as a guide for on screen digitizing and placement of model features.

5.1 Reading the Image

To import the image:

1. Select the *Open* button .
2. Locate and open the directory entitled **tutfiles\MODFLOW\modfmap**.
3. Open the file entitled **start.gpr**.

All other objects in GMS are drawn on top of the image. The image only appears in plan view. You may wish to import other images located in the same folder.

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 **easttex.**

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

7 Defining the Units

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

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

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

5. Select the *OK* button.

8 Defining the Boundary

The first step is to define the outer boundary of the model. We will do this by creating an arc which forms a closed loop around the site.

8.1 Create the Coverage

1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New* | *Conceptual Model* command.

2. For the *Name*, enter **East Texas**. For the *Model*, select **MODFLOW**.

3. Click *OK*.

4. Right-click on the **East Texas** conceptual model and select the *New Coverage* command from the pop-up menu.

5. Change the *Coverage name* to **Boundary**. Change the *Default elevation* to **213.** Change the *Default layer range* to go from **1** to **2**.

6. Click *OK*.

8.2 Create the Arc

1. Select the *Create* Arc tool .

2. Begin the arc by clicking once on the left (west) side of the model at the location shown in Figure 2.

3. Create the arc by proceeding around the boundary of the site in a counter-clockwise direction and clicking on a sequence of points around the boundary. Don't worry about the spacing or the exact location of the points; just use enough points to define the approximate location of the boundary. The boundary on the south and east sides of the model should coincide with the rivers. The boundary along the top should coincide to the limestone outcropping as shown in Figure 2.

4. To end the arc, click on the point where you began.

Note: As you are clicking on the points, if you make a mistake and wish to back up a point or two, press the *Backspace* key. If you wish to abort the arc and start over, press the *ESC* key.

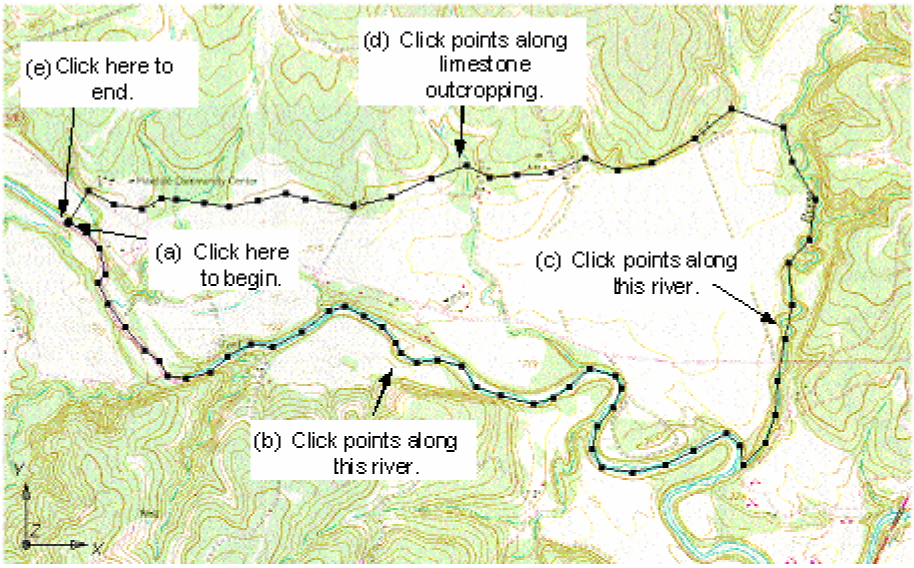


Figure 2. Creating the Boundary Arc.

9 Building the Local Source/Sink Coverage

The next step in building the conceptual model is to construct the local sources/sinks coverage. This coverage defines the boundary of the region being modeled and it defines local sources/sinks including wells, rivers, drains, and general head boundaries.

The properties which can be assigned to the feature objects in a coverage depend on the conceptual model and the options set in the Coverage Setup dialog. Before creating the feature objects, we will change the options in the Coverage Setup dialog.

1. Right-click on the **Boundary** coverage and select the *Duplicate* command from the pop-up menu. Change the new coverage name to **Sources & Sinks**.

2. Right-click on the **Sources & Sinks** coverage and select the *Coverage Setup* command from the pop-up menu.

3. In the list of *Sources/Sinks/BCs*, turn ON the following options which we will need for this tutorial:

   - *Layer range*
   - *Wells*
   - *Refine points*

- *Specified Head (CHD)*
- *Drain*

4. Make sure the *Use to define model boundary (active area)* option is on.
5. Click OK.

9.1 Defining the Specified Head Arcs

The next step is to define the specified head boundary along the south and east sides of the model. Before doing this, however, we must first split the arc we just created into three arcs. One arc will define the no-flow boundary along the top and the other two arcs will define the two rivers. An arc is split by selecting one or more vertices on the arc and converting the vertices to nodes.

1. Select the *Select Vertices* tool .
2. Select the two vertices shown in Figure 3. Vertex #1 is located at the junction of the two rivers. Vertex #2 is located at the top of the river on the east side of the model. To select both vertices at once, select the first vertex and then hold down the *Shift* key while selecting the other vertex.
3. Right-click on one of the selected vertices and select *Vertex -> Node* command.

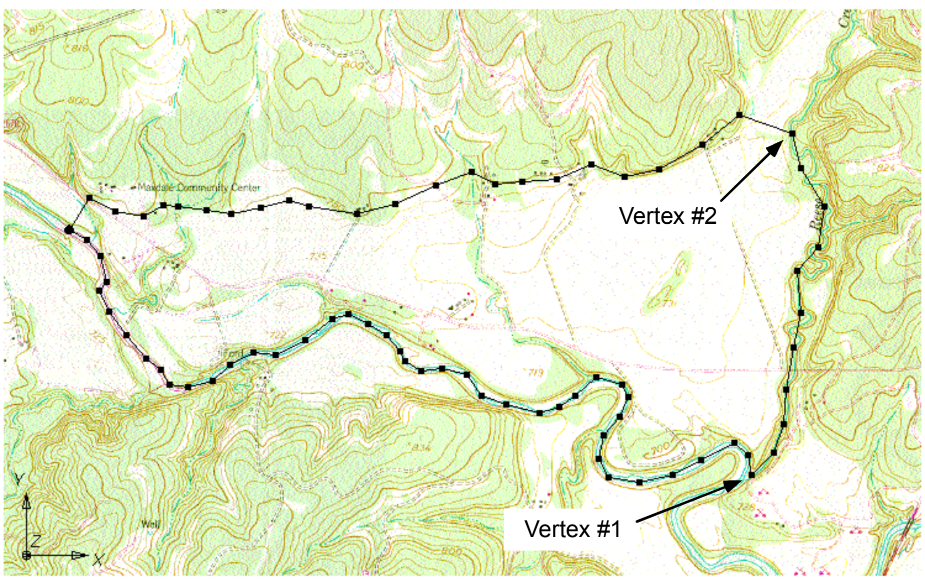


Figure 3. Convert Vertices to Nodes.

Now that we have defined the three arcs, we will specify the two arcs on the rivers as specified head arcs.

4. Select the *Select Arcs* tool .

5. Select the arcs on the south and east and (right and bottom) sides of the model by selecting one arc and holding down the *Shift* key while selecting the other arc.

6. Right-click on one of the selected arcs and select the *Attribute Table* command from the pop-up menu.

7. Find the spreadsheet cell corresponding to the *All* row and the *Type* column. In this cell, select the **spec. head** type. This will change the types for both arcs.

8. Select the *OK* button.

9. Click anywhere on the model other than on the arcs to unselect them.

Note that the color of the arcs has changed indicating the type of the arc.

The next step is to define the head at the nodes at the ends of the arcs. The head along a specified head arc is assumed to vary linearly along the length of the arc.

10. Select the *Select Points/Nodes* tool .

11. Double click on the node on the west (left) end of the arc on the southern (bottom) boundary.

12. Enter a constant value of **212** for the *Head-Stage*.

13. Select the *OK* button.

14. In a similar fashion, assign a value of **208** to the node at the junction of the two rivers and a value of **214** to the node at the top of the arc on the east boundary of the model.

9.2 Defining the Drain Arcs

At this point, we will enter the arcs at the locations of the creek beds to define the drains.

1. Select the *Create Arc* tool .

2. Create the arc labeled as arc #1 in Figure 4. Start by clicking on the bottom arc, create the arc by clicking points along the creek bed, and end the arc by double clicking on the top arc.

Notice that when you click in the vicinity of a vertex on an existing arc or on the edge of an arc, GMS automatically snaps the arc you are creating to the existing arc and makes a node at the junction of the two arcs.

3. Create the arcs labeled arc #2 and arc #3 in Figure 4 the same way you made arc #1.

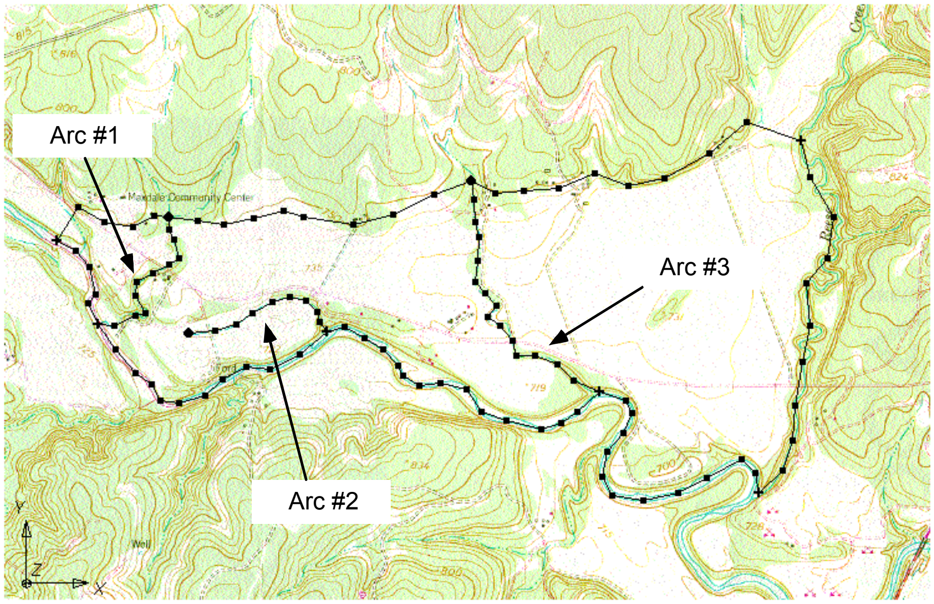


Figure 4. The Drain Arcs.

Next, we will define the arcs as drains and assign the conductance and elevation to the arcs.

4. Select the *Select Arcs* tool.
5. Select all of the drain arcs by clicking on the arcs while holding down the *Shift* key.
6. Right-click on one of the selected arcs and select *Attribute Table* command from the pop-up menu.
7. In the *All* row, *Type* column, select the **drain** option.
8. Enter a conductance of **555** in the *All* row. This represents a conductance per unit length value. GMS automatically computes the appropriate cell conductance value when the drains are assigned to the grid cells.
9. Change the *From layer* and *To layer* properties to be **1** for each of the arcs. This means the drains will only be in layer 1 of the grid.
10. Select the *OK* button.

The elevations of the drains are specified at the nodes of the arcs. The elevation is assumed to vary linearly along the arcs between the specified values.

11. Select the *Select Points/Nodes* tool.

Double click on Node 2 in Figure 5. Notice that this node has 2 properties associated with it since it is attached to 2 arcs of different types.

12. Enter **212** for the *Bot. elev.* of the *drain* property. Do NOT change anything in the *spec. head* property. Click *OK*.

13. Repeat this procedure to assign the drain elevations to the nodes shown in Figure 5. Be sure to change the *drain* property only, and NOT the *spec. head* property.

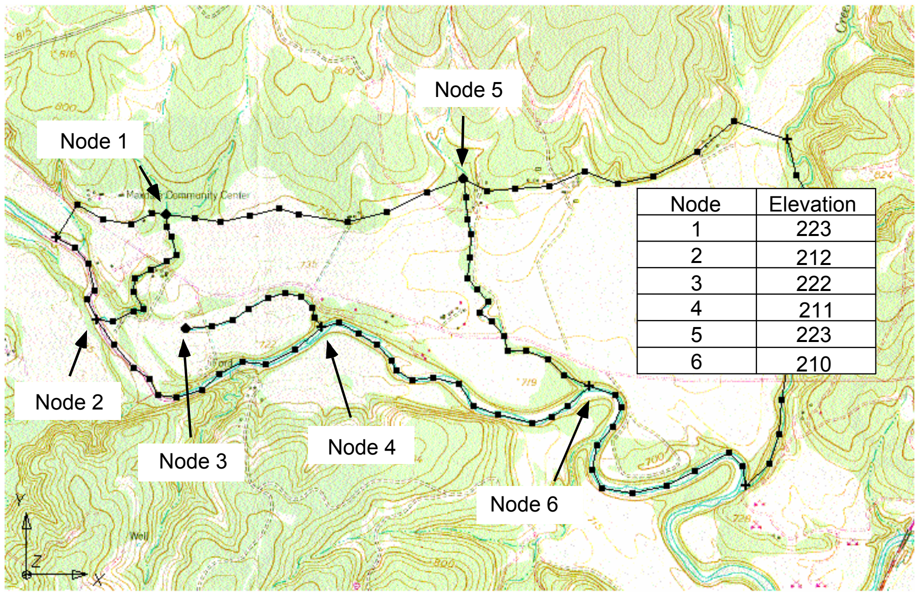


Figure 5. Elevations for Drain Nodes.

9.3 Building the polygons

With the local sources/sinks type coverage, the entire region to be modeled must be covered with non-overlapping polygons. This defines the active region of the grid. In most cases, all of the polygons will be variable head polygons (the default). However, other polygons may be used. For example, to model a lake, a general head polygon can be used. The simplest way to define the polygons is to first create all of the arcs used in the coverage and then select the *Build Polygons* command. This command searches through the arcs and creates a polygon for each of the closed loops defined by the arcs. These polygons are of type “NONE” by default but may be converted to other types by selecting the polygons and using the *Properties* command.

Now that all of the arcs in the coverage have been created, we are ready to construct the polygons. All of our polygons will be variable head polygons.

1. Select the *Feature Objects* | *Build Polygons* command.

Notice that the polygon is now filled. You can change the view of the polygons if you wish by selecting the *Display* | *Display Options* command.

9.4 Creating the Wells

The final step in creating the local sources/sinks coverage is to define the wells. Wells are defined as point type objects. Two wells will be created.

1. Select the *Create Point* tool.
2. Move the cursor to the approximate location of Well #1 shown in Figure 1 and click once with the mouse to create the point.
3. While the new point is selected, type the coordinates **(613250, 3428630)** in the *X* and *Y* edit fields at the top of the GMS window and hit the *Tab* or *Enter* key.
4. Select the *Properties* button.
5. For the *Type*, select the **well** option.
6. For the *Flow rate*, enter a constant value of **-680**.
7. Change the *From layer* and *To layer* properties to be **1**. This means the well will only be in layer 1 of the grid.
8. Select the *OK* button.
9. In a similar fashion, create the other well at the location **(615494, 3428232)** and assign a pumping rate of **–2,830**. However, for this well, change the *From layer* and *To layer* so that the well is applied only to layer two (change both the edit fields to **2**).

Grid Refinement

A well represents a point of convergence in the groundwater flow and causes steep gradients in the head near the well. In order to accurately model the flow near wells, the grid is typically refined in the vicinity of the wells. This type of refinement can be performed automatically in GMS by assigning refinement data directly to the wells in the conceptual model.

1. Select the *Select Points/Nodes* tool.
2. Select both wells by clicking on the wells while holding the *Shift* key.
3. Select *Properties* button.
4. Find the *Refine* column, and in the *All* row, turn on the toggle. This turns on refinement for both points.
5. Change the *Base size* to **25**, the *Bias* to **1.1** and the *Max size* to **150** for both points.
6. Click *OK*.

10 Delineating the Recharge Zones

The next step in constructing the conceptual model is to construct the coverage which defines the recharge zones. 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.

10.1 Copying the Boundary

We'll create our recharge coverage by copying the boundary.

1. Right-click on the **Boundary** coverage and select the *Duplicate* command from the pop-up menu.
2. Change the name of the new coverage to **Recharge**.
3. Right-click on the **Recharge** coverage and select the *Coverage Setup* command.
4. In the *Areal Properties* list, turn on the *Recharge rate* property.
5. Select the *OK* button.

10.2 Creating the Landfill Boundary

Next, we will create the arc delineating the boundary of the landfill. To do this, we will first create a closed loop in the form of a rectangle at the approximate location of the landfill. We will then edit the nodes and vertices so that the arc coincides precisely with the boundary of the landfill.

1. Select the *Create Arc* tool .
2. Create a rectangular polygon representing the landfill as shown in Figure 6. Don't worry about getting the exact coordinates at this point.

Now that the arc is created in the approximate location, we will edit the coordinates of the vertices and nodes to define the precise coordinates.

3. Select the *Select Vertices* tool .
4. Drag a box around the entire landfill polygon rectangle, thus selecting all the vertices.
5. Right-click on one of the selected vertices and select the *Vertex -> Node* command.
6. Select the *Select Points/Nodes* tool .
7. Select one of the nodes.

8. While the node is selected, enter the exact coordinates of the node in the *Edit Window*. Select the *Tab* key after entering each coordinate value.

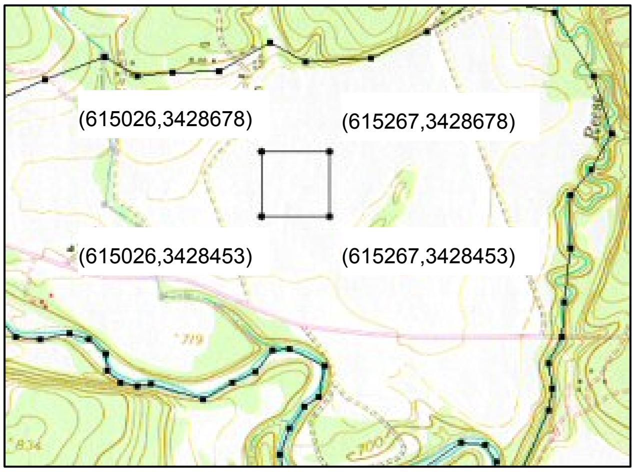


Figure 6. Landfill.

9. Repeat this process for the remaining corners of the landfill polygon.

10.3 Building the Polygons

Now that the arcs are defined, we can build the polygons.

1. Select the *Feature Objects* | *Build Polygons* command.

10.4 Assigning the Recharge Values

Now that the recharge zones are defined, we can assign the recharge values. We will assign one value to the landfill polygon, and another value to the remaining 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.
5. Double click on the outer polygon.
6. Change the *Recharge rate* to **0.00695.**
7. Select the *OK* button.

11 Defining the Hydraulic Conductivity

Next we will enter the hydraulic conductivity for each layer. In many cases, you may wish to define multiple polygons defining hydraulic conductivity zones. For the sake of simplicity, we will use a constant value for each layer.

11.1 Copying the Boundary

We'll create our layer coverage by copying the boundary.

1. Right-click on the **Boundary** coverage and select the *Duplicate* command from the pop-up menu.
2. Change the name of the new coverage to **Layer 1**.
3. Right-click on the **Layer 1** coverage and select the *Coverage Setup* command.
4. In the *Areal Properties* list, turn ON the following options
   - *Horizontal K*
   - *Vertical anis.*
5. Change the *Default layer range* to go from **1** to **1**.
6. Select the *OK* button.
7. Right click on the **Layer 1** coverage and select the *Duplicate* command from the pop-up menu. Change the name of the new coverage to **Layer 2**.
8. Right click on the **Layer 2** coverage and select the *Coverage Setup* command.
9. Change the *Default layer range* to go from **2** to **2**.
10. Select the *OK* button.

11.2 Top Layer

First, we will assign a K value for the top layer.

1. Select the **Layer 1** coverage in the *Project Explorer*.
2. Select the *Feature Objects* | *Build Polygons* command.
3. With the *Select Polygons* tool, double click on the polygon.
4. Change the *Horizonal K* to **5.5**.
5. Change the *Vertical anis.* to **4**.

6. Select the *OK* button.

11.3 Bottom Layer

For the bottom 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. Change the *Vertical anis.* to **4**.
6. Select the *OK* button.

This completes the definition of the coverages in the conceptual model. Before continuing to create the grid, we will make the sources/sinks coverage the active coverage.

7. Select the **Sources & Sinks** coverage in the *Project Explorer*.

12 Locating the Grid Frame

Now that the coverages are complete, we are ready to create the grid. The first step in creating the grid is to define the location and orientation of the grid using the Grid Frame. The Grid Frame represents the outline of the grid. It can be positioned on top of our site map graphically.

1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New | Grid Frame* command.
2. In the *Project Explorer* right-click on the *Grid Frame* and select the *Fit to Active Coverage* command.
3. Double-click on the grid frame in the *Project Explorer* to bring up the properties dialog.
4. Change the *Origin z:* to **170** and the *Dimension z:* to **60**. This provides a set of initial values for the MODFLOW layer elevation arrays. Later, we will interpolate the layer elevations.
5. Select the *OK* button to exit the *Grid Frame* dialog.

13 Creating the Grid

Now that the coverages and the Grid Frame are created, we are now ready to create the grid.

1. Select the *Feature Objects* | *Map → 3D Grid* command.

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.

2. In the *Z-Dimension* change *Number cells* to **2**.

3. Select the OK button.

14 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 (1)* item in the *Project Explorer* and select the *New MODFLOW* command.

2. Select the *OK* button.

15 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 *Select Polygons* tool.

3. Select one of the polygons.

4. Select *Properties* button.

5. Confirm that the layer assignment is 1 to 2 and click *OK*.

6. 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.

16 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 import from web tool available in GMS.

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

16.1 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.

16.2 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** array, and click the *Map* button.

3. Highlight the **ground_elev** data set and the **Top Elevations Layer 1** array, and click the *Map* button.

4. Select the *OK* button to perform the interpolation.

16.3 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

16.4 Adjusting the Display

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

1. Uncheck the scatter point sets in the *Project Explorer*.

2. Uncheck the grid frame in the *Project Explorer*.

16.5 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. Select a cell somewhere near the center of the model.

3. Select the Side *View* button .

To get a better view of the cross section, we will increase the z magnification.

4. Select the *Display Options* command.
5. Enter a value of **5** for the *Z magnification* factor.
6. Select the *OK* button.
7. Select the *Frame* 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.

16.6 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 *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 *Plan View* button.
9. In the mini-grid display, 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.

17 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.

18 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*.

19 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.

20 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.

21 Viewing the Head Contours

A set of contours should appear. To get better contrast between the contours and the background image, we will change the contour color to blue.

1. Select *Contour Options* from the main toolbar.

2. Click on the *Color Ramp* button and select the down arrow on the *Color* item.

3. Select a dark blue color.

4. Select the *OK* button twice to exit the dialogs.

To view the contours for the second layer:

5. Select the up arrow in the mini-grid display.

6. After viewing the contours, return to the top layer by selecting the down arrow .

22 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 .

23 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.

24 Conclusion

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

- A background image can be imported to help you construct the conceptual model.
- It is usually a good idea to define the model boundary in a coverage and copy that coverage whenever you need to create a new coverage.
- You can customize the set of properties associated with points, arcs and polygons by using the *Coverage Setup* dialog.
- Some arc properties, like head, are not specified by selecting the arc but by selecting the nodes at the ends of the arc. That way the property can vary linearly along the length of the arc.

- A grid frame can be used to position the grid, but is not required.
- You must use the *Map* → *MODFLOW / MODPATH* command every time you want to transfer the conceptual model data to the grid.
- 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.

