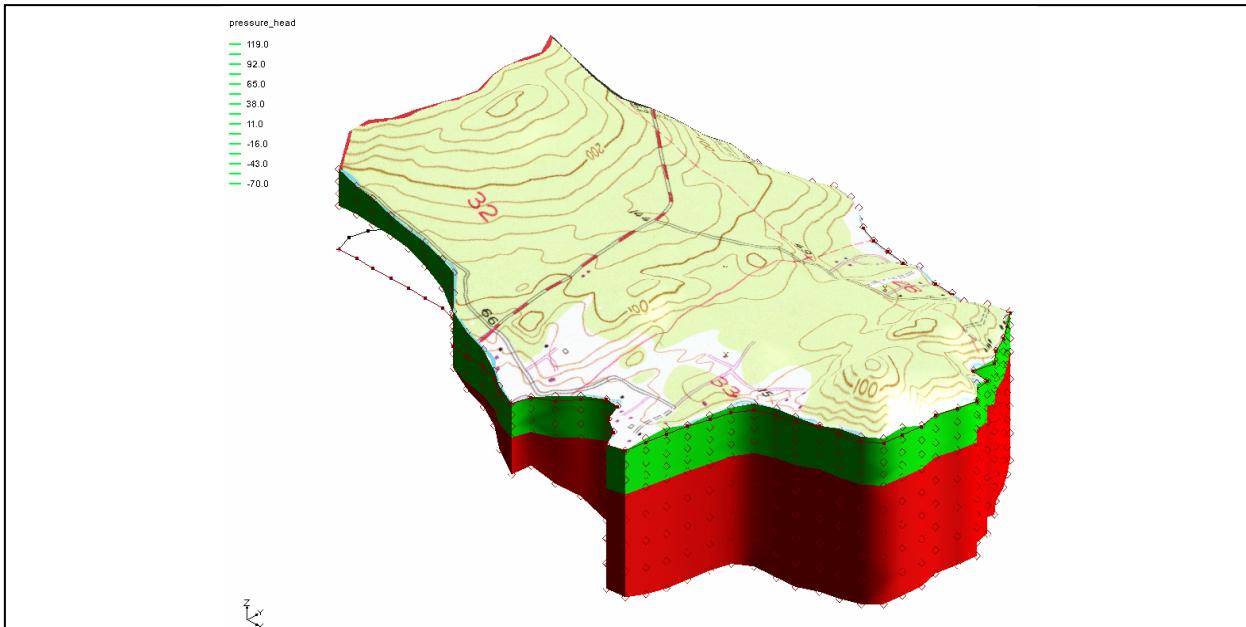


## GMS 9.0 Tutorial

### **FEMWATER – Flow Model**

Build a FEMWATER model to simulate flow



### Objectives

Build a 3D mesh and a FEMWATER flow model using the conceptual model approach. Run the model and examine the results.

#### Prerequisite Tutorials

- None

#### Required Components

- FEMWATER
- Geostatistics
- Map
- Mesh
- Sub-surface characterization

#### Time

- 30-60 minutes

**AQUAVEO™**



# 1 Contents

---

<b>1</b>	<b>Contents .....</b>	<b>2</b>
<b>2</b>	<b>Introduction.....</b>	<b>2</b>
2.1	Outline .....	3
<b>3</b>	<b>Description of Problem.....</b>	<b>3</b>
<b>4</b>	<b>Getting Started .....</b>	<b>4</b>
<b>5</b>	<b>Building the Conceptual Model .....</b>	<b>4</b>
5.1	Importing the Background Image .....	4
5.2	Saving With a New Name .....	5
5.3	Defining the Units .....	5
5.4	Initializing the FEMWATER Coverage .....	5
5.5	Creating the Boundary Arcs .....	6
5.6	Redistributing the Arc Vertices .....	6
5.7	Defining the Boundary Conditions.....	7
5.8	Building the Polygon .....	7
5.9	Assigning the Recharge .....	7
5.10	Creating the Wells .....	8
<b>6</b>	<b>Building the 3D Mesh.....</b>	<b>8</b>
6.1	Defining the Materials .....	9
6.2	Building the 2D Projection Mesh .....	9
6.3	Building the TINs .....	9
6.4	Interpolating the Terrain Data .....	10
6.5	Interpolating the Layer Elevation Data.....	10
6.6	Building the 3D Mesh.....	11
<b>7</b>	<b>Hiding Objects.....</b>	<b>12</b>
<b>8</b>	<b>Converting the Conceptual Model.....</b>	<b>12</b>
<b>9</b>	<b>Selecting the Analysis Options .....</b>	<b>12</b>
9.1	Entering the Run Options .....	12
9.2	Setting the Iteration Parameters .....	13
9.3	Selecting Output Control .....	13
9.4	Defining the Fluid Properties .....	13
<b>10</b>	<b>Defining Initial Conditions .....</b>	<b>13</b>
10.1	The Scatter Point Set .....	14
10.2	Creating the Data Set.....	15
<b>11</b>	<b>Defining the Material Properties .....</b>	<b>15</b>
<b>12</b>	<b>Saving and Running the Model.....</b>	<b>16</b>
12.1	Viewing Head Contours .....	16
12.2	Viewing a Water Table Iso-Surface.....	17
12.3	Draping the TIFF Image on the Ground Surface .....	17
<b>13</b>	<b>Conclusion .....</b>	<b>18</b>

## 2 Introduction

---

FEMWATER is a three-dimensional finite element ground water model. It can be used to simulate flow and transport in both the saturated and the unsaturated zone. Furthermore, the flow and transport can be coupled to simulate density dependent problems such as salinity intrusion. This tutorial describes how to build a FEMWATER model to simulate flow only.

## 2.1 Outline

---

This is what you will do:

1. Import a background image.
2. Define coverages and map them to a 2D mesh.
3. Create tins from the mesh.
4. Interpolate elevations from scatter points to the tins.
5. Build a 3D mesh from the tin horizons.
6. Map the conceptual model to a FEMWATER simulation.
7. Define additional conditions and run FEMWATER.
8. View the water table as an iso-surface.
9. Drape the TIFF image on the ground surface.

## 3 Description of Problem

---

The site to be modeled in this tutorial is shown in Figure 1. The site is a small coastal aquifer with three production wells, each pumping at a rate of 2,830 m<sup>3</sup>/day. The no flow boundary on the upper left corresponds to a parallel flow boundary and the no flow boundary on the left corresponds to a thinning of the aquifer due to a high bedrock elevation. A stream provides a specified head boundary on the lower left and the remaining boundary is a coastal boundary simulated with a specified head condition.

The stratigraphy of the site consists of an upper and lower aquifer. The upper aquifer has a hydraulic conductivity of 3 m/day and the lower aquifer has a hydraulic conductivity of 9 m/day. The wells extend to the lower aquifer. The recharge to the aquifer is about one foot per year. The objective of this tutorial is to create a steady state flow model of the site.

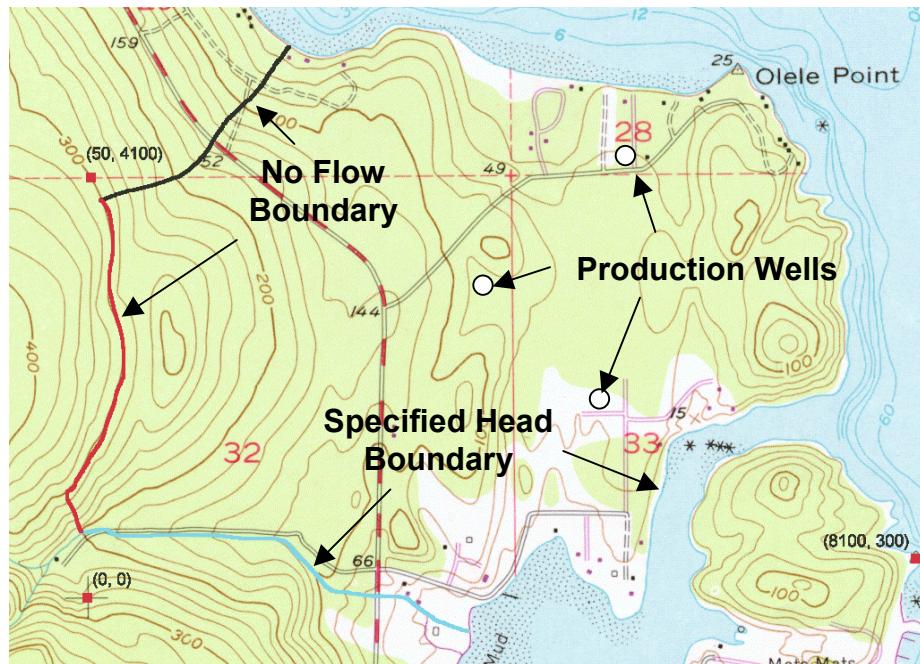


Figure 1. Site to be modeled with FEMWATER.

## 4 Getting Started

If you have not yet done so, launch GMS. If you have already been using GMS, you may wish to select the *File | New* command to ensure the program settings are restored to the default state.

## 5 Building the Conceptual Model

FEMWATER models can be constructed using the direct approach or the conceptual model approach. With the direct approach, a mesh is constructed and the boundary conditions are assigned directly to the mesh by interactively selecting nodes and elements. With the conceptual model approach, feature objects (points, arcs, and polygons) are used to define the model domain and boundary conditions. The mesh is then automatically generated and the boundary conditions are automatically assigned. We will use the conceptual model approach for this tutorial.

### 5.1 Importing the Background Image

Before creating the feature objects, we will import a scanned image of the site. The image was generated by scanning a section of a USGS quadrangle map on a desktop scanner. The image has previously been imported to GMS and registered to real world coordinates. The registered image was saved to a GMS project file. To read the image:

1. Select the *Open* button .
2. Locate and open the directory entitled: **Tutorials\FEMWATER\femwater**
3. Select the file entitled **start.gpr** and select the *Open* button.

## 5.2 Saving With a New Name

---

We'll create a new project by saving it with a new name.

1. Select the *File | Save As* command.
2. Enter **femmod.gpr** for the project name.
3. Select the *Save* button.

You should save your work periodically throughout the tutorial by clicking the save button .

## 5.3 Defining the Units

---

We will now define the units. GMS uses the units we select to plot helpful labels next to input edit fields.

1. Select the *Edit | Units* command.
2. Check to ensure that the *Length* unit is **m** (meters) and the default *Time* unit is **d** (days).
3. Change the *Mass* unit to **kg** (kilograms). The remaining units are for a transport simulation and can be ignored.
4. Select the *OK* button to exit the *Units* dialog.

## 5.4 Initializing the FEMWATER Coverage

---

Before creating the feature objects, we first need to create a FEMWATER 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. In the *Conceptual Model Properties* dialog, change the name to “**femmod**”.
3. Change the *Type* to **FEMWATER** and click *OK*.
4. Right-click on the **femmod** conceptual model  and select the *New Coverage* command from the pop-up menu.
5. Change the *Coverage Name* to “**femwater**”.
6. Turn on the following properties:
  - *Flow BC*
  - *Wells*
  - *Refinement*
  - *Meshing options*
7. Select the *OK* button.

## 5.5 Creating the Boundary Arcs

We are now ready to begin creating the arcs defining the boundary of the model. Notice that the three boundaries on the left have been marked on the background image and are color-coded.

1. Select the *Create Arc* tool .
2. Using the boundary lines shown on the background image, create one arc for each of the three marked boundaries on the left side of the model. Click on a series of points along each line to trace the arc and double click at the end of the boundary to end the arc. Make sure the arcs are connected by starting one arc precisely at the ending point of the previous arc.
3. Create an arc for the coastline boundary. Once again, be sure the arc is connected to the other boundary arcs.

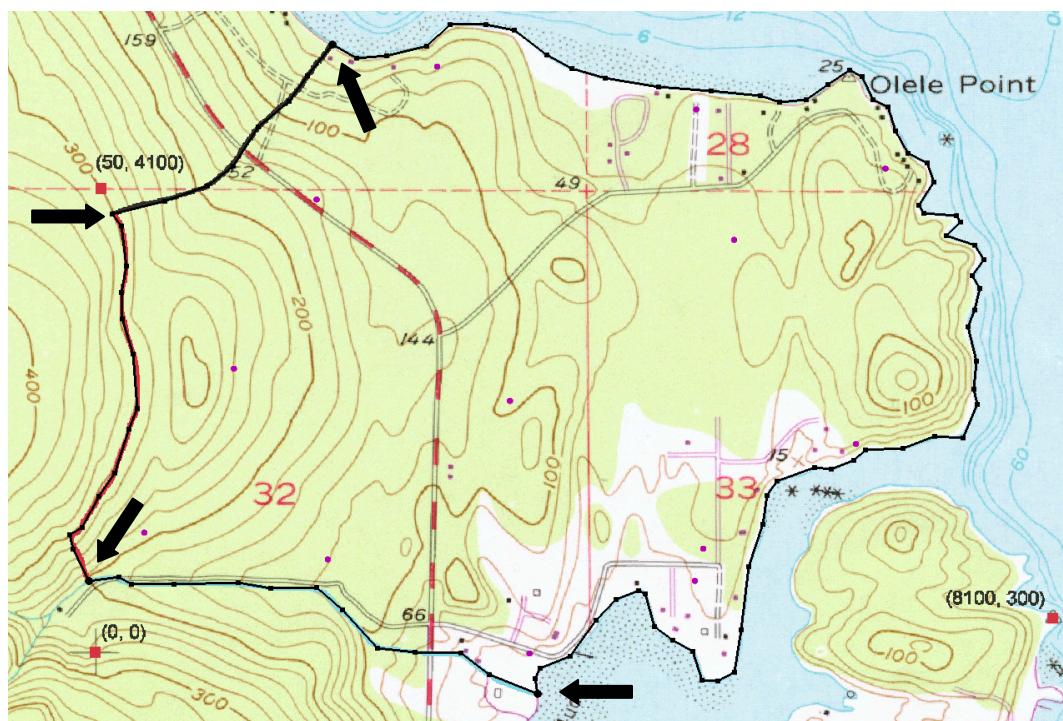


Figure 2. Boundary arcs. Arrows show endpoints of arcs.

## 5.6 Redistributing the Arc Vertices

The two endpoints of each arc are called *nodes* and the intermediate points are called *vertices*. These arcs will be used to generate a 2D mesh that will be converted into a 3D mesh. The spacing of the line segments defined by the vertices will control the size and number of elements. Thus, we need to redistribute the vertices along each arc to ensure that they are evenly spaced and of the correct length.

1. Select the *Select Arcs* tool .
2. Select all of the arcs by dragging a box that encloses all of the arcs.
3. Select the *Feature Objects | Redistribute Vertices* command.

4. Change the *Spacing* to **90**.
5. Select the *OK* button.

## 5.7 Defining the Boundary Conditions

---

Now that the arcs are defined, we can assign boundary conditions to the arcs. The two no-flow boundaries do not need to be altered since the default boundary type is no-flow. However, we need to mark the stream arc and the coastline arc as constant head arcs.

1. Select the *Select Arcs* tool .
2. While holding down the *Shift* key, click on both the stream arc and on the coastline arc.
3. Select the *Properties* button .
4. In the *All* row in the spreadsheet, change the *Flow bc* to **spec. head**. This assigns this type to both arcs.
5. Select the *OK* button.

Note that no head values were assigned to the arcs. The head values are assigned to the nodes at the ends of the arc. This allows the head to vary linearly along the length of the arc. For the coastline arc, the head at both ends is the same. Since the default head value is zero, no changes need to be made. However, we do need to enter a head value for the upper end of the stream. The head will vary linearly along the stream from the specified value at the top to zero at the coast.

6. Select the *Select Points/Nodes* tool .
7. Double click on the node at the top (left) end of the stream arc.
8. Enter a value of **60** for the head.
9. Select the *OK* button.

## 5.8 Building the Polygon

---

Now that the arcs are defined, we are ready to build a polygon defining the model domain. The polygon is necessary for two reasons: 1) it is used to define the model domain when the mesh is generated, and 2) it is used to assign the recharge. In many cases, the model domain is subdivided into multiple recharge zones, each defined by a polygon. In this case, we will use one polygon since we have a single recharge value.

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

## 5.9 Assigning the Recharge

---

Next, we will assign the recharge value. There are two ways to assign recharge in FEMWATER: using a specified flux boundary or using a variable boundary. The variable boundary is more accurate but it is more time-consuming and less stable. To ensure that the tutorial can be completed in a timely fashion, we will use the simpler specified flux approach.

1. Select the *Select Polygons* tool .

2. Double-click anywhere in the interior of the model domain.
3. Change the *Flow bc* to be **spec. flux**.
4. Enter a value of **0.0009** for the *Flux rate* (this value is in m/d and corresponds to about 0.34 m/yr).
5. Select the *OK* button.
6. Click anywhere outside the polygon to unselect it.

## 5.10 Creating the Wells

---

The final step in defining the conceptual model is to create the wells. We want to create three wells with the properties shown in the following table.

Well	X	Y	Z	Top scr.	Bot. scr.	Flow rate	Elem. size
1	1612	1282	14	-44	-55	-2830	45
2	1175	925	23	-50	-62	-2830	45
3	1532	571	14	-75	-90	-2830	45

To create the first well, do the following:

1. Select the *Create Point* tool .
2. Create a point anywhere in the upper right corner of the model.
3. Using the edit fields at the top of the screen, change the xyz coordinates of the point you just created to **(1612, 1282, 14)**. Hit the *Tab* or *Enter* key after entering each value.
4. With the point still selected, select the *Properties* button .

First, we will mark the point as a well point. For a well, we define the pumping rate, and the elevation of the screened interval. The screened interval is used to determine which nodes in the 3D mesh to assign the pumping rate to. The pumping rate is factored among all nodes lying within the screened interval. In our case, there will only be one node.

5. Change the *Type* to **well**.
6. Enter **-44** for the *Top scr.*, **-55** for the *Bot. scr.* and **-2830** for the *Flow rate*.

Next, we will set the "Refine" option so that the mesh is refined around the well.

7. Turn on the *Refine* option.
8. Enter **45** for the *Elem. size* (this controls the element size at the well).
9. Select *OK* to exit the dialog.
10. Repeat this entire process to create two more wells with the properties given in the above table.

## 6 Building the 3D Mesh

---

At this point, the conceptual model is complete and we are ready to build the 3D finite element mesh. The mesh will consist of two zones, one for the upper aquifer and one for the lower aquifer. To build the mesh, we will first create a 2D "projection" mesh using

the feature objects in the conceptual model. We will then create three triangulated irregular networks (TINs): one for the top (terrain) surface, one for the bottom of the upper aquifer, and one for the bottom of the lower aquifer. We will then create the 3D elements by using the horizons to 3D mesh command.

## 6.1 Defining the Materials

---

Before building the mesh, we will define a material for each of the aquifers. The materials are assigned to the TINs and eventually to each of the 3D elements.

1. Select the *Edit | Materials* command.
2. Change the name of the default material to **Upper Aquifer**.
3. Change the *Color/Pattern* to green.
4. Create a new material by typing entering the name **Lower Aquifer** in the last row (the row with a \*).
5. Change the new *Color/Pattern* to red.
6. Select the *OK* button.

## 6.2 Building the 2D Projection Mesh

---

The 2D projection mesh can be constructed directly from the conceptual model:

1. Select the *Feature Objects | Map → 2D Mesh* command.

After a few seconds, the mesh should appear.

## 6.3 Building the TINs

---

To build the TINs defining the stratigraphic horizons, we will make three TINs where each TIN is a copy of the 2D mesh. At first, these three TIN's will have the same elevations (zero) as the 2D mesh. We will then use a set of scatter points and interpolate the proper elevations to the TINs.

To create the top TIN:

1. Select the *2D Mesh Data* folder  in the *Project Explorer*.
2. Select the *Mesh | Mesh → TIN* command.
3. Enter **terrain** for the name of the TIN.
4. In the *Tin material* list, select *Upper Aquifer* (this defines the material below the TIN).
5. Enter **2** for the *Horizon id*.
6. Select *OK*.

To create the second TIN:

7. Select the *Mesh | Mesh → TIN* command.
8. Enter **bottom upper aquifer** for the name of the TIN.
9. In the *Tin material* list, select *Lower Aquifer*.

10. Enter **1** for the *Horizon id*.

11. Select *OK*.

To create the third TIN:

12. From the *Mesh | Mesh → TIN* command.

13. Enter **bottom lower aquifer** for the name of the TIN.

14. In the *Tin material* list, select *Lower Aquifer* (this value is ignored for the bottom TIN).

15. Enter **0** for the *Horizon id*.

16. Select *OK*.

## **6.4 Interpolating the Terrain Data**

---

Next, we will use the scatter points defining the terrain elevations to interpolate to the top TIN. The terrain points were created by digitizing elevations from the contour map. Before interpolating to the TIN, we need to make sure the top TIN is the active TIN.

1. From the *Project Explorer*, select the TIN named **terrain** .

Before interpolating, we will make some adjustments to the interpolation options (based on our experience with these scatter point data).

2. Select the scatter point set entitled **terrain**  in the *Project Explorer* to make it the active scatter set.

3. Select the *Interpolation | Interpolation Options* command.

4. Select the *Options* button next to the *Inverse distance weighted* item.

5. In the *Nodal function* section, make sure the *Constant (Shepard's method)* option is selected.

6. Select the *OK* button twice to exit both dialogs.

To interpolate from the scatter points to the TIN.

7. In the *Project Explorer* right-click on the *terrain* scatter set  and then, from the pop-up menu, select the *Interpolate To | Active TIN* command.

8. Select the *OK* button.

To view the interpolated elevations:

9. Select the *Oblique View* macro .

10. Select the *Display Options* macro .

11. Enter a *Z magnification* factor of **4.0**.

12. Select the *OK* button.

## **6.5 Interpolating the Layer Elevation Data**

---

Next, we will interpolate the elevations defining the bottom of the upper and lower aquifers. These elevations were obtained from a set of exploratory boreholes. Once

again, before interpolating to the TIN, we need to make sure the desired TIN is the active TIN.

1. Select the TIN named **bottom upper aquifer**  from the *Project Explorer*.
2. Select the *2D Scatter Data Folder*  in the *Project Explorer*.

This scatter point set has two data sets: one set of elevations for the bottom of the upper aquifer and one set for the bottom of the lower aquifer. We will first interpolate the elevations for the bottom of the upper aquifer.

3. Select the data set entitled **elevs**  in the *Project Explorer* to make it the active scatter point set.
4. If necessary, expand the **elevs** item so you can see the data sets associated with the scatter point set.
5. Select the **bot of layer 1** data set  to make it active.

To interpolate from the scatter points to the TIN.

6. In the *Project Explorer* right-click on the *elevs* scatter set  and then, from the pop-up menu, select the *Interpolate To | Active TIN* command.
7. Select the *OK* button.

Finally, we will interpolate the elevations for the bottom TIN.

8. Select the TIN named **bottom lower aquifer**  in the *Project Explorer*.
9. Select the data set named **bot of layer 2**  in the *Project Explorer*.
10. In the *Project Explorer* right-click on the *elevs* scatter set  and then, from the pop-up menu, select the *Interpolate To | Active TIN* command.
11. Select the *OK* button.
12. At this point, you should see the correct elevations on all three TINs.

## 6.6 Building the 3D Mesh

We are now ready to build the 3D mesh. The 3D mesh is constructed using the horizon method

1. Click on the *TIN Data* folder  in the *Project Explorer*.
2. Select the *TINs | Horizons -> 3D Mesh* command.
3. We will use the defaults on the first page of the wizard. Select *Next >*.
4. In the *Top elevation* section select *Tin elevations* and select the **terrain** TIN from the *TIN Data* list.
5. Also, in the *Bottom elevation* section select *Tin Elevations* and select the **bottom lower aquifer** TIN from the *TIN Data* list.
6. Select *Next >*.
7. Turn on the *Refine elements* option and select *refine all elements*.

8. Turn on the *Subdivide material layers* option and make sure *Target layer thickness* is selected
9. Change the *Upper Aquifer* maximum layer thickness to **10** and the *Lower Aquifer* thickness to **20**.
10. Select the *Finish* button.

A 3D mesh will now be constructed between the TINS from the options that we selected.

## 7 Hiding Objects

---

Before continuing, we will unclutter the display by hiding the objects we are finished with. We will hide everything but the feature objects and the 3D mesh.

To hide the TINs:

1. Uncheck all three TINs in the *Project Explorer*.

We can also hide the scatter points:

2. Uncheck all the scatter point sets in the *Project Explorer*.

## 8 Converting the Conceptual Model

---

Now we are ready to convert the conceptual model to the 3D mesh model. This will assign all of the boundary conditions using the data defined on the feature objects.

1. Right-click on the 3D mesh  and select the *New FEMWATER* command from the pop-up menu.
2. Select the *OK* button.
3. In the *Project Explorer* right-click on the *femmod* conceptual model  and select *Map to → FEMWATER* command from the pop-up menu.
4. Select *OK* at the prompt (since we only have one coverage, the choice here does not matter).

A set of symbols should appear indicating that the boundary conditions have been assigned.

## 9 Selecting the Analysis Options

---

Next, we will switch to the *3D Mesh* module and select the analysis options.

### 9.1 Entering the Run Options

---

First, we will indicate that we wish to run a steady state flow simulation:

1. Select the *FEMWATER | Run Options* command.
2. For the *Type of simulation*, ensure the **Flow Only** option is selected.
3. In the *Steady State vs. Transient* section, select **Steady state solution**.

The problem we are solving has a very large partially saturated region, mainly in the upper left corner of the model. The larger the unsaturated zone, the more difficult it is to get FEMWATER to converge. For these types of problems, the *Nodal/Nodal* option is a good choice for quadrature. It is not as accurate as the default option (*Gaussian/Gaussian*), but it is more stable.

4. In the *Quadrature selection* section, select the **Nodal/Nodal** option.
5. Select the *OK* button.

## 9.2 Setting the Iteration Parameters

---

Next, we will adjust the iteration parameters.

1. Select the *FEMWATER | Iteration Parameters* command.
2. In the *Flow simulation* section set the *Max iterations for non-linear equation* to **100**.
3. Set the *Max iterations for linear equation* to **1000**.
4. Set the *Steady state convergence criterion* to **0.003**.
5. Set the *Transient convergence criterion* to **0.003**.
6. Select the *OK* button.

## 9.3 Selecting Output Control

---

Next, we will select the output options. We will choose to output a pressure head file only.

1. Select the *FEMWATER | Output Control* command.
2. Make sure that the *Save nodal moisture content file*, *Save velocity file*, and *Save flux file* options are turned **off**.
3. Select the *OK* button.

## 9.4 Defining the Fluid Properties

---

Finally, we will define the fluid properties.

1. Select the *FEMWATER | Fluid Properties* command.

The items in this dialog are dependent on the units we have selected. Since this is a steady state solution, we can ignore the viscosity and compressibility options.

2. Enter **1000** for the *Density of water*.
3. Select the *OK* button.

# 10 Defining Initial Conditions

---

Because of the non-linear equations used by FEMWATER to model flow in the unsaturated zone, FEMWATER is more sensitive to initial conditions than saturated flow models such as MODFLOW. If a set of initial conditions is defined that is significantly

different than the final head distribution, FEMWATER may be slow or even unable to converge.

For a flow simulation, FEMWATER requires a set of pressure heads as an initial condition. The FEMWATER interface in GMS includes a command that can be used to automatically create a set of pressure heads from a user-defined water table surface. The water table surface is defined by a set of scatter points. A total head data set is generated by interpolating head values from the scatter points to the nodes of the 3D mesh. Finally, the pressure head data set is created by subtracting the node elevations from the total head values.

## 10.1 The Scatter Point Set

---

To define the initial condition, we will use a small set of points at the expected elevation of the computed water table surface. In the interest of time, these points have already been created for you and are already in the project. To learn more about how to create scatter points, refer to the tutorial in Volume I entitled *2D Geostatistics*.

Before digitizing the points, we will turn off the flux boundary condition display options.

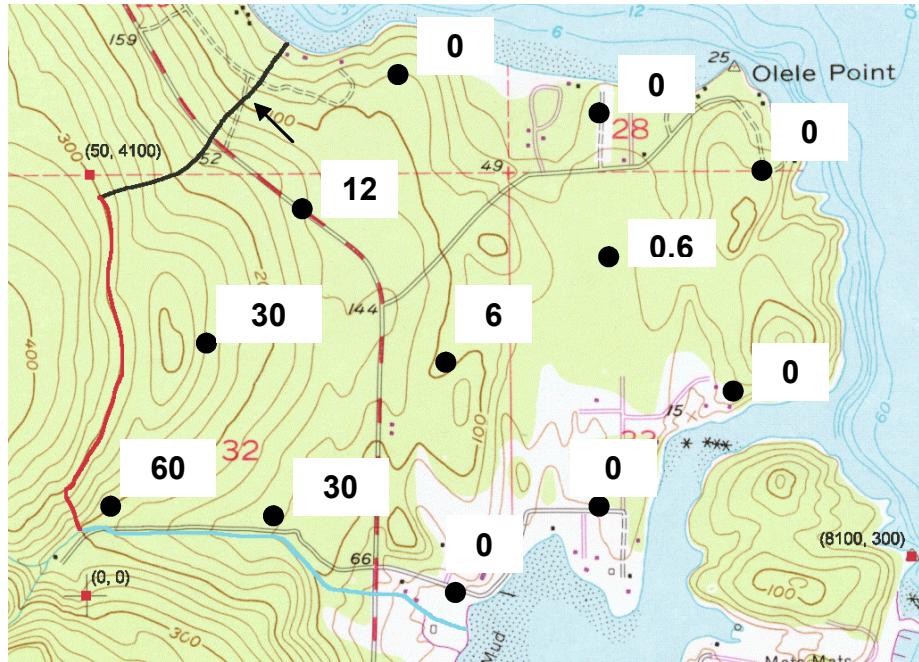
1. Select the *FEMWATER | BC Display Options* command.
2. Turn off the *Flux fluid (CB1)* option.
3. Select the *OK* button.

To view the scatter point set.

4. Switch to *Plan View* .
5. In the *Project Explorer*, uncheck the *2D Mesh data*  and *3D Mesh data*  folders.
6. Turn on the scatter point set named **starheads** .

The scatter points should be located as shown in Figure 3. The values will not be visible by default, but you could turn them on if desired.

7. When you are finished viewing the scatter points, turn the *3D Mesh data*  back on by checking it in the *Project Explorer*.



*Figure 3. Locations and elevations for new points.*

## 10.2 Creating the Data Set

Finally, to define the initial condition, we must create a pressure head data set. The data set will be saved to a data set file. The path to this file will be passed to FEMWATER when the simulation is launched.

1. Select the *FEMWATER | Initial Conditions* command.
  2. In the *Cold start* section on the upper left side of the dialog, select the *Read from data set file* option.
  3. Select the *Generate IC* button.
  4. Select **starheads** for the *Active 2D scatter point set*.
  5. Enter **-60** for the *Minimum pressure head*.
  6. Select the *OK* button.
  7. When prompted for the file name, enter **starthd.phd** and select the *Save* button.
  8. Select the *OK* button to exit the *Initial Conditions* dialog.

## 11 Defining the Material Properties

The final step in setting up the model is to define the material properties.

1. Select the *Edit | Materials* menu command.

We need to enter a hydraulic conductivity and a set of unsaturated zone curves for each aquifer.

To define the material properties for the upper aquifer:

2. For the *Upper aquifer* material enter a value of **3.0** for  $K_{xx}$ ,  $K_{yy}$  and  $K_{zz}$ .
3. With one of the cells in the “Upper Aquifer” material row selected, select the *Generate Unsat Curves* button.
4. Set the *Curve type* to **van Genuchten equations**.
5. Enter **1.8** for the *Max. height above water table*.
6. Select the *Preset parameter values* section and select *Silt*.
7. Select the *Compute Curves* button.
8. Select the *OK* button.

To define the material properties for the lower aquifer:

9. For the *Lower aquifer* enter a value of **9.0** for  $K_{xx}$ ,  $K_{yy}$  and  $K_{zz}$ .
10. With one of the cells in the “Lower Aquifer” material row selected, select the *Generate Curves* button.
11. Select the *Compute Curves* button.
12. Select the *OK* button.
13. Select the *OK* button to exit the *Materials* dialog.

## 12 Saving and Running the Model

---

We are now ready to save and run the model.

1. Select the *Save* button .

To run FEMWATER:

2. Select the *FEMWATER | Run FEMWATER* command.

The FEMWATER window should appear and you will see some information on the progress of the model convergence. The model should converge in a few minutes.

3. When the model converges, select the *Close* button.

### 12.1 Viewing Head Contours

---

First, we will view a color fringe plot.

1. Select the *Display Options* button .
2. Verify that the *Contours* option is turned on.
3. Select the *Options* button next to the *Contours* item.
4. Under the *Contour method* section of the dialog change the color of the line to blue by selecting *Single Color* from the second drop down and then selecting blue from the color list on the right.
5. Select *OK* to exit the dialog.
6. Select the *FEMWATER* tab.
7. Turn off the *Specified Head (DB1)* option.

8. Select the *OK* button.

## **12.2 Viewing a Water Table Iso-Surface**

---

Another way to view the solution is to generate an iso-surface at a pressure head of zero. This creates a surface matching the computed water table. Further, if we cap the iso-surface on the side greater than zero, we will get a color-shaded image of the pressure variation in the saturated zone.

1. Expand the **femmod** solution  in the *Project Explorer*.
2. Select **pressure\_head**  to make it the active data set.
3. Select the *Display Options* button .
4. Turn **off** the *Element edges* and the *Contours* options.
5. Turn **on** the *Iso-surfaces* option.
6. Select the *Options* button next to the *Iso-surfaces* item.
7. Make sure that the *number of iso-surfaces* is **1**.
8. For the upper value on the first line of the spreadsheet enter a value of **0.0**.
9. Turn **on** the *Fill between* toggle on the second line of the spreadsheet to fill between **0.0** and the maximum value.
10. Turn **on** the *Iso-surface faces*.
11. Turn **on** the *Contour specified range* option.
12. Set the *Minimum* to **0** and the *Maximum* to **140**.
13. Select the *OK* button to exit both dialogs.
14. Select the *Oblique View* macro .

## **12.3 Draping the TIFF Image on the Ground Surface**

---

Finally, we will drape the TIFF image on the terrain surface to illustrate the spatial relationship between the computed water table surface and the ground surface. To do this, we will first unhide the top TIN.

1. Check the box next to the **terrain** TIN  to show it.
2. Expand the **terrain** TIN  in the *Project Explorer* and select the **z\_idw\_const** Data Set .

Next we will set the option to map the TIFF image to the TIN when shaded.

3. Select the *Display Options* button .
4. Turn on the *Texture map image to active tin* option.
5. Select *OK*.
6. Select the *Rotate* tool .
7. Click on the screen and drag horizontally to the left side of the screen. Repeat as necessary until the other side of the model is visible.

Finally, we will try the smooth shade option.

8. Select the *Display Options* button .
9. Select the *Lighting Options* item  in the list box.
10. Select the *Smooth features* option on the *Lighting Options* tab.
11. Select the *OK* button.

## 13 Conclusion

---

This concludes the tutorial. Here are the things that you should have learned in this tutorial:

- FEMWATER is a 3D finite element model that is more complex than MODFLOW (which is a 3D finite difference model).
- How to create a FEMWATER conceptual model
- How to use a conceptual model to create a 3D finite element mesh, using a 2D mesh, TINs and scatter points.
- How to set up FEMWATER initial conditions.