

GMS TUTORIALS

FEMWATER – Flow Model

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.

1.1 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.

1.2 Required Modules/Interfaces

You will need the following components enabled to complete this tutorial:

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

2 Description of Problem

The site to be modeled in this tutorial is shown in Figure 2-1. The site is a small coastal aquifer with three production wells, each pumping at a rate of 2,830 m³/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.

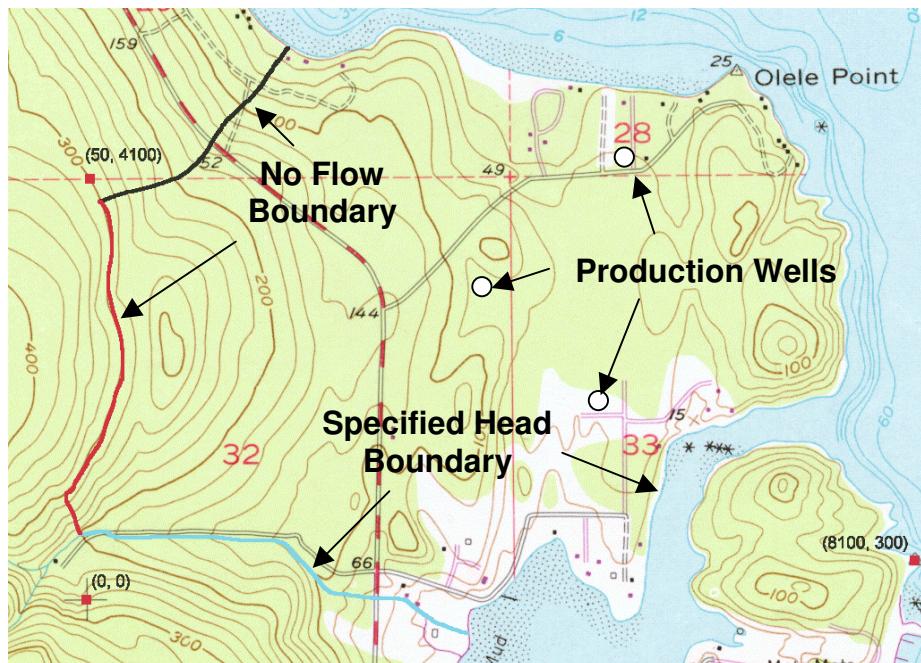


Figure 2-1 Site to be Modeled with FEMWATER Flow Model.

3 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.

4 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.

4.1 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: **tutfiles\FEMWATER\femwater**
3. Select the file entitled **start.gpr** and select the *Open* button.

4.2 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.

4.3 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. Change the name to “**femmod**”.
3. Change the model 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.

4.4 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.

4.5 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.

4.6 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.

1. Select the *Select Points/Nodes* tool .
2. Double click on the node at the top (left) end of the stream arc.
3. Enter a value of **60** for the head.
4. Select the *OK* button.

4.7 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.

4.8 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.

4.9 Creating the Wells

The final step in defining the conceptual model is to create the wells. 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.

1. Change the *Type* to **well**.
2. Enter **-44** for the *Top scr.*
3. Enter **-55** for the *Bot. scr.*
4. Enter **-2830** for the *Flow rate*.

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

5. Turn on the *Refine* option.
6. Enter **45** for the *Elem. size* (this controls the element size at the well).
7. Select *OK* to exit the dialog.

Repeat this entire process to create two more wells with the following properties:

Well #	1	2
X	1175	1532
Y	925	571
Z	23	14
Element size	45	45
Top of screen	-50	-75
Bottom of screen	-62	-90
Flow rate	-2830	-2830

5 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.

5.1 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**. To change the name click on the material name in the text window.
3. Change the *Material color* to green.
4. Select the *New* button to create another material.
5. Change the name of the new material to **Lower Aquifer**.
6. Change the *Material color* to red.
7. Select the *OK* button.

5.2 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.

5.3 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*.
7. Select *No* at the prompt. We do not want to delete the 2D mesh.

To create the second TIN:

8. Select the *Mesh | Mesh → TIN* command.
9. Enter **bottom upper aquifer** for the name of the TIN.
10. In the *Tin material* list, select *Lower Aquifer*.
11. Enter **1** for the *Horizon id*.
12. Select *OK*.
13. Select *No* at the prompt.

To create the third TIN:

14. From the *Mesh | Mesh → TIN* command.
15. Enter **bottom lower aquifer** for the name of the TIN.

16. In the *Tin material* list, select *Lower Aquifer* (this value is ignored for the bottom TIN).
17. Enter **0** for the *Horizon id*.
18. Select *OK*.
19. Select *No* at the prompt.

5.4 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
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.

5.5 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.

5.6 Building the 3D Mesh

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

1. Select the *TINs | Horizons -> 3D Mesh* command.

2. We will use the defaults on the first page of the wizard. Select *Next >*.
3. In the *Top elevation* section select *Tin elevations* and select the **terrain** TIN from the *TIN Data* list.
4. Also, in the *Bottom elevation* section select *Tin Elevations* and select the **bottom lower aquifer** TIN from the *TIN Data* list.
5. Select *Next >*.
6. Turn on the *Refine elements* option and select *refine all elements*.
7. Turn on the *Subdivide material layers* option and make sure *Target layer thickness* is selected
8. Change the *Upper Aquifer* maximum layer thickness to **10** and the *Lower Aquifer* thickness to **20**.
9. Select the *Finish* button.

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

6 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*.

7 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. Select the *FEMWATER | New Simulation* command. (this initializes the data structures for the FEMWATER simulation).
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.

8 Selecting the Analysis Options

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

8.1 Entering the Run Options

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

1. Select the *3D Mesh Data Folder*  in the *Project Explorer*.
2. Select the *FEMWATER | Run Options* command.
3. For the *Type of simulation*, ensure the *Flow only* option is selected.
4. 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.

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

8.2 Setting the Iteration Parameters

Next, we will adjust the iteration parameters.

1. Select the *FEMWATER | Iteration Parameters* command.
2. 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.

8.3 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.

8.4 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.

9 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.

9.1 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 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 9-1. 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*.

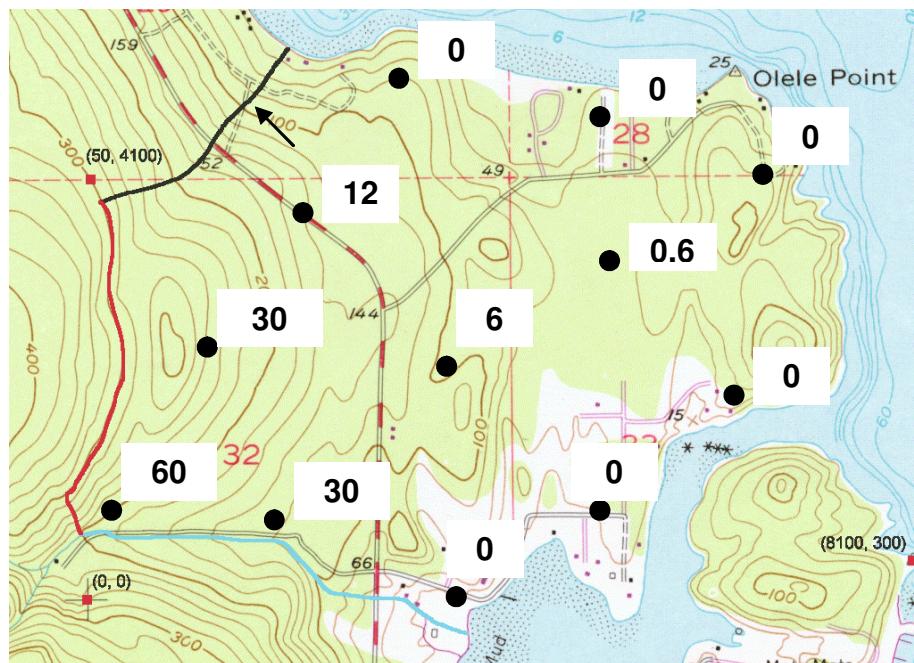


Figure 9-1 Locations and Elevations for New Points.

9.2 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.

10 Defining the Material Properties

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

9. Select the FEMWATER | Material Properties dialog.

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:

10. Select the *Upper aquifer* material from the list of materials.
11. Select the *FEMWATER Flow* tab.
12. Enter a value of **3.0** in the upper left edit field in the *Conductivity* section and select the *Isotropic* button. This should fill in all of the diagonal terms with a value of **3.0**.
13. Select the *Generate Curves* button.
14. Set the *Curve type* to **van Genuchten equations**.
15. Enter **1.8** for the Max. height above water table.
16. Select the *Preset parameter values* section and select *Silt*.
17. Select the *Compute Curves* button.
18. Select the *OK* button.

To define the material properties for the lower aquifer:

19. Select the *Lower aquifer* item in the list at the top of the dialog.

20. Enter a value of **9.0** in the upper left edit field in the *Conductivity* section and select the *Isotropic* button
21. Select the *Generate Curves* button.
22. Select the *Compute Curves* button.
23. Select the *OK* button.
24. Select the *OK* button to exit the *Material Properties* dialog.

11 Saving and Running the Model

We are now ready to save and run the model.

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

To run FEMWATER:

4. 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.

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

11.1 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 *Line options* section of the dialog change the color of the line to blue.
5. Select *OK* to exit the dialog.
6. Select the *FEMWATER* tab.
7. Turn off the *Head* option.
8. Select the *OK* button.

11.2 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 .

11.3 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.

12 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.