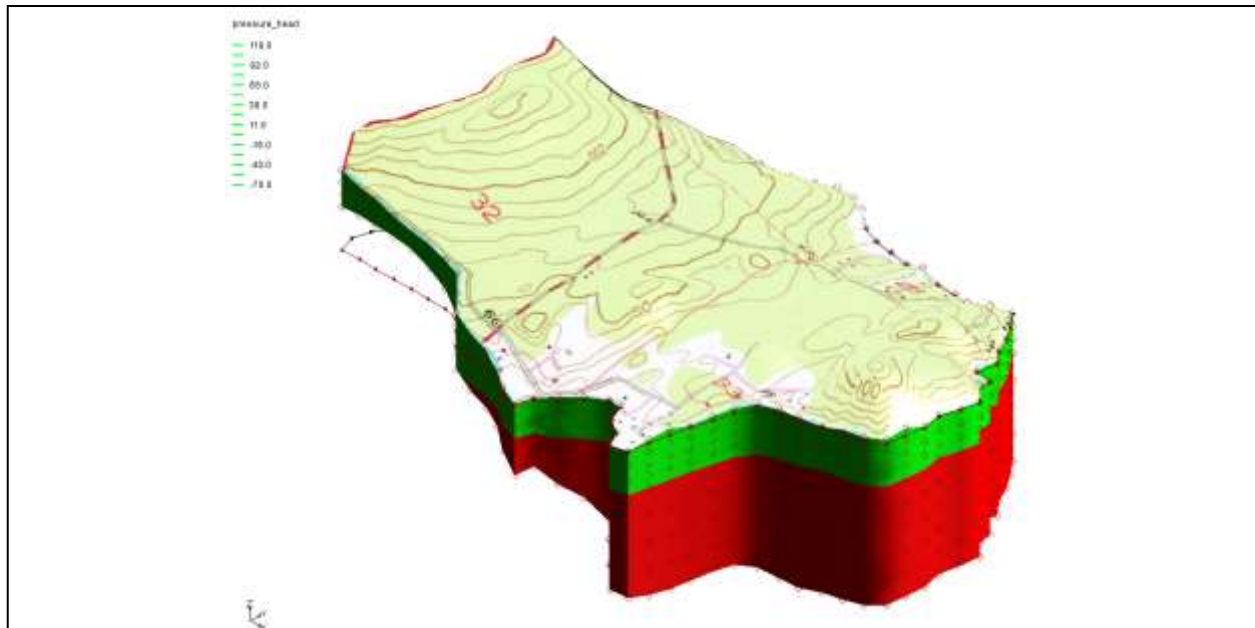*GMS 10.5 Tutorial*

FEMWATER – Flow Model

Build a FEMWATER model to simulate flow

Objectives

This tutorial shows how to build a 3D mesh and a FEMWATER flow model using the conceptual model approach. It will also review running the model and examining the results.

Prerequisite Tutorials	Required Components	Time
• Feature Objects • Geostatistics 2D • Stratigraphy Modeling – TINs	• FEMWATER • Geostatistics • Map Module • Mesh Module • Sub-surface characterization	• 45–65 minutes

1   Introduction

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

The site being modeled in this tutorial is a small coastal aquifer with three production wells, each pumping at a rate of 2,830 m^3/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.

This tutorial demonstrates and discusses importing a background image, defining coverages and mapping them to a 2D mesh, creating TINs from the mesh, interpolating elevations from scatter points to TINs, building a 3D mesh from the TIN horizons, mapping the conceptual model to a FEMWATER simulation, defining additional conditions and running FEMWATER, viewing the water table as an isosurface, and draping the TIFF image on the ground surface.

1.1   Getting Started

Do the following to get started:

1. If GMS is not running, launch GMS.

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

2   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. The conceptual model approach will be used for this tutorial.

2.1   Importing the Background Image

Before creating the feature objects, 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 then saved to a GMS project file.

Do the following to import the image:

1. Click **Open** 📁 to bring up the *Open* dialog.

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

3. Browse to the *femwater* directory for this tutorial and select "start.gpr".

4. Click **Open** to import the project file and close the *Open* dialog.

2.2   Saving with a New Name

Create a new project by saving it with a new name:

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

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

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

4. Click **Save** to create the new project file and close the *Save As* dialog.

It is recommended to **Save** ![floppy disk icon] periodically throughout the tutorial.

2.3   Defining the Units

Now define the units. GMS uses the selected units to plot helpful labels next to input edit fields.

1. Select *Edit | **Units…*** to bring up the *Units* dialog.

2. Select "d" (days) from the *Time* drop-down.

3. Select "kg" (kilograms) from the *Mass* drop-down.

The remaining units are for a transport simulation and can be ignored.

4. Click **OK** to exit the *Units* dialog.

2.4   Initializing the FEMWATER Coverage

Before creating the feature objects, first create a FEMWATER coverage.

1. In the Project Explorer, right-click on the empty space and select *New | **Conceptual Model…*** to open the *Conceptual Model Properties* dialog.

2. Enter "femmod" as the *Name*.

3. Select "FEMWATER" from the *Type* drop-down.

4. Click **OK** to close the *Conceptual Model Properties* dialog.

5. Right-click on the new "![icon] femmod" conceptual model and select **New Coverage…** to open the *Coverage Setup* dialog.

6. Enter "femwater" as the *Coverage Name*.

7. In the *Sources/Sinks/BCs* column, turn on *Flow BC*, *Wells*, and *Refinement*.

8. In the *Areal Properties* column, turn on *Meshing options*.

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

2.5   Creating the Boundary Arcs

It is now possible 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 "femwater" coverage to make it active.

2. Using the **Create Arc** ![arc icon] tool, click out a series of vertices to create an arc for the black line, double-clicking to end the arc (black line in Figure 1).

3. Click out a series of vertices to create an arc for the red line, pressing *Enter* to end the arc (red line in Figure 1). Make sure the arcs are connected by starting precisely at the ending point of the previous arc.

4. Click out a series of vertices to create an arc for the blue river line (blue line in Figure 1). Again, make sure the arcs are connected

Figure 1   First three boundary arcs

5. Create an arc for the coastline boundary (between the arrows in Figure 2). Once again, be sure the arc is connected to the other boundary arcs.

Figure 2   Coastline boundary arc

2.6   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. Redistributing the vertices along each arc ensures that they are evenly spaced and of the correct length.

1. Using the **Select Arcs** ⟋ tool, select all four arcs by dragging a box that encloses all of the arcs.

2. Select *Feature Objects* / **Redistribute Vertices…** to open the *Redistribute Vertices* dialog.

3. Enter "90" for the *Average spacing.*

4. Click **OK** to close the *Redistribute Vertices* dialog and redistribute the vertices.

2.7   Defining the Boundary Conditions

Now that the arcs are defined, 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, it's necessary to mark the stream arc and the coastline arc as constant-head arcs.

1. Using the **Select Arcs** ⟋ tool, hold down the *Shift* key and select the coastline arc and the stream arc (the blue one in Figure 1, above).

2. Click **Properties** 🖼 to open the *Attribute Table* dialog.

3. In the *All* row in the spreadsheet, select "spec. head" from the *Flow bc* column drop-down to assign this to both arcs.

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

No head values were assigned to the arcs because they 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, it is necessary 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 inland end (or top) to zero at the coast.

5. Using the **Select Points\Nodes** ⟋ tool, double-click on the inland node of the stream arc to bring up the *Attribute Table* dialog.

6. On the "spec. head" row, enter "60.0" in the *Head (m)* column.

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

2.8   Building the Polygon

Now that the arcs are defined, build a polygon defining the model domain. The polygon is necessary because it is used to define the model domain when the mesh is generated, and 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, use only one polygon since the model has a single recharge value.

1. Select *Feature Objects* / **Build Polygons**.

2. To view the polygon area, use the **Select Polygons** 🔲 tool to click on the center of the new polygon.

The Graphics Window should appear similar to Figure 3.

Figure 3   The polygon defining the model domain

2.9   Assigning the Recharge

Next, 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, use the simpler specified flux approach.

1. Using the **Select Polygons** 🔲 tool, double-click anywhere in the interior of the model domain to bring up the *Attribute Table* dialog.

2. Select "spec. flux" from the *Flow bc* drop-down.

3. Enter "0.0009" in the *Flux rate* column. This value is in m/d and corresponds to about 0.34 m/yr.

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

5. Click anywhere outside the polygon to deselect it.

2.10   Creating the Wells

The final step in defining the conceptual model is to create the wells. It is necessary to create three wells in this case.

To create the first well, do the following:

1. Using the **Create Point** ⣀ tool, create a point anywhere in the upper right corner of the model.

2. Using the edit fields at the top of the Graphic Window, change the *XYZ* coordinates of the point that was just created to "1612," "1282," "14". Hit the *Tab* or *Enter* key after entering each value.

3. With the point still selected, click **Properties** to open the *Attribute Table* dialog.

A well point needs to be of the type "well". The pumping rate and elevation of the screened interval need to be defined. The screened interval is used to determine which 3D mesh nodes need to have the pumping rate assigned. The pumping rate is factored among all nodes within the screened interval. In this case, there is only one node.

4. Scroll to the right if necessary, to see the additional columns in the spreadsheet.

5. In the spreadsheet, select "well" from the drop-down in the *Type* column.

6. Enter "-44" in the *Top scr* column.

7. Enter "-55" in the *Bot. scr* column.

8. Enter "-2830" in the *Flow rate* column.

9. Turn on *Refine*.

This refines the mesh around the well.

10. Enter "45" in the *Elem. size* column.

This controls the element size at the well.

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

12. Repeat steps 1–11 to create the second and third wells with the properties given in the table below.

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

3   Building the 3D Mesh

At this point, the conceptual model is complete and ready to be used 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, a 2D "projection" mesh will first be created by using the feature objects in the conceptual model. Three triangulated irregular networks (TINs) will then be created: one for the top (terrain) surface, one for the bottom of the upper aquifer, and one for the bottom of the lower aquifer. Finally, the 3D elements will be created by using the **Horizons → 3D Mesh** command.

3.1   Defining the Materials

Before building the mesh, 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 *Edit* | **Materials…** to bring up the *Materials* dialog.

2. In row *1*, enter "Upper Aquifer" in the *Name* column and press the *Tab* key.

3. Select *Green* from the drop-down in the *Color/Pattern* column.

4. In row *2* (marked with * until this step is completed), create a new material by entering "Lower Aquifer" in the *Name* column, then press the *Tab* key.

5. Select *Red* from drop-down in the *Color/Pattern* column.

6. Click **OK** close the *Materials* dialog.

3.2   Building the 2D Projection Mesh

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

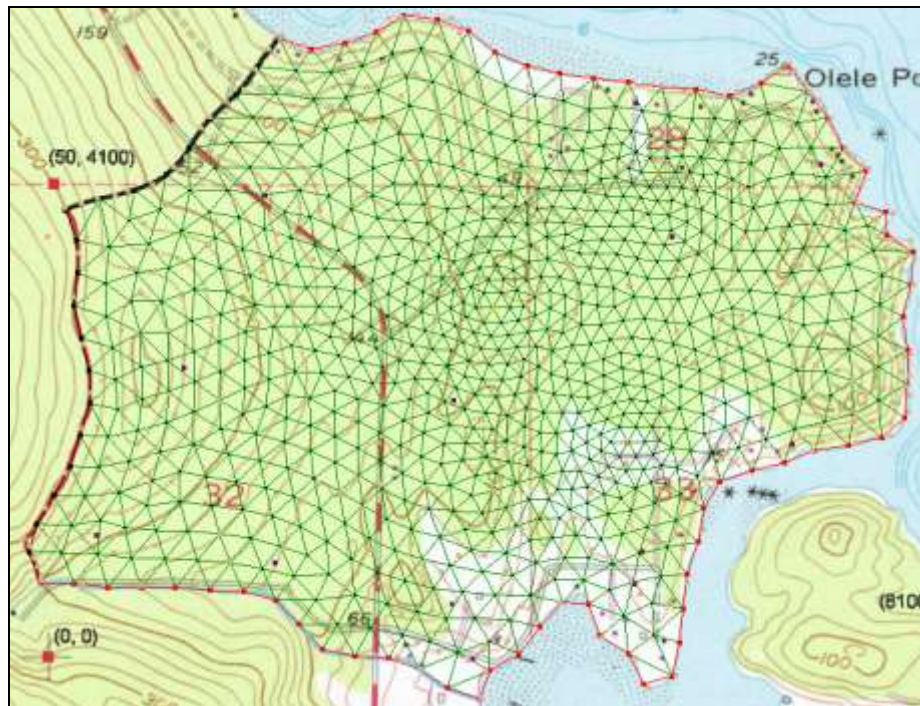1. Select *Feature Objects* | **Map → 2D Mesh**.

After a few seconds, the mesh should appear (Figure 4).

Figure 4   The 2D mesh

3.3   Building the TINs

To build the TINs defining the stratigraphic horizons, make three TINs where each TIN is a copy of the 2D mesh. At first, these three TINs will have the same elevations (zero) as the 2D mesh. After creating the TINs, 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 to make it active.

2. Select *Mesh | Convert to | **TIN*** to open the *Properties* dialog.

3. Enter "terrain" for the *TIN name*.

4. Select "Upper Aquifer" from the *TIN material* drop-down. This defines the material below the TIN.

5.   Enter "2" for the *Horizon ID*.

6.   Click **OK** to close the *Properties* dialog.

To create the second TIN:

1.   Select *Mesh | Convert to | **TIN** to open the *Properties* dialog.

2.   Enter "bottom upper aquifer" for the *TIN name*.

3.   Select "Lower Aquifer" from the *TIN material* drop-down.

4.   Enter "1" for the *Horizon ID*.

5.   Click **OK** to close the *Properties* dialog.

To create the third TIN:

1.   Select *Mesh | Convert to | **TIN** to open the *Properties* dialog.

2.   Enter "bottom lower aquifer" for the *TIN name*.

3.   Select "Lower Aquifer" from the *TIN material* drop-down. This defines the material below the TIN.

4.   Enter "0" for the *Horizon ID*.

5.   Click **OK** to close the *Properties* dialog.

3.4   Interpolating the Terrain Data

Next, 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, make sure the top TIN is active, and make some adjustments to the interpolation options (based on the experience with these scatter point data).

1.   From the Project Explorer, select the " terrain" TIN.

2.   Expand the " 2D Scatter Data" folder.

3.   Select the " terrain" scatter point set to make it active.

4.   Select *Interpolation* / **Interpolation Options…** to open the *2D Interpolation Options* dialog.

5.   In the *Interpolation methods* section, click **Options…** next to *Inverse distance weighted* to open the *2D IDW Interpolation Options* dialog.

6.   In the *Nodal function* section, select *Constant (Shepard's method)*.

7.   Click **OK** to exit the *2D IDW Interpolation Options* dialog.

8.   Click **OK** to exit the *2D Interpolation Options* dialog.

To interpolate from the scatter points to the TIN, do the following:

1.   In the Project Explorer, right-click on the " terrain" scatter point set and select *Interpolate To* / **Active TIN** to bring up the *Interpolate → Object* dialog.

2.   Click **OK** to accept the defaults and close the *Interpolate → Object* dialog.

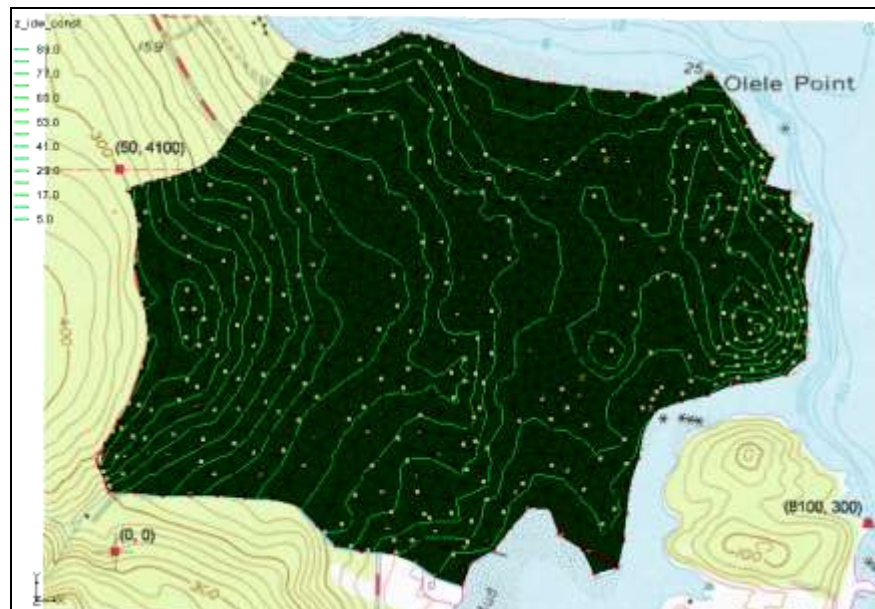The Graphics Window will appear as in Figure 5.

Figure 5   Terrain scatter data has been interpolated

To view the interpolated elevations:

1. Switch to **Oblique View** ◈.

2. Click **Display Options** 🖵 to bring up the *Display Options* dialog.

3. In the section below the list on the left, enter "4.0" for the *Z magnification.*

4. Click **OK** to close the *Display Options* dialog.

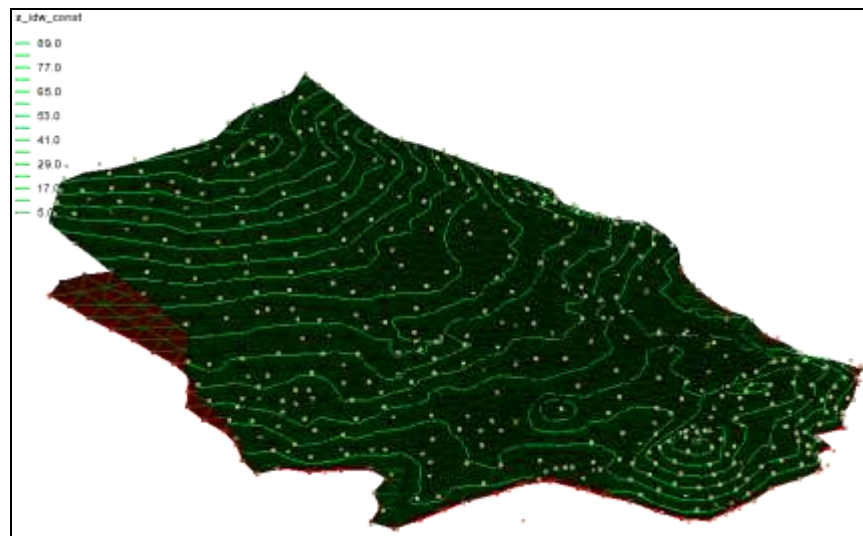The Graphics Window should appear similar to Figure 6.

Figure 6   Oblique view showing elevations on the terrain TIN

3.5   Interpolating the Layer Elevation Data

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

1.   Select the "⬡ bottom upper aquifer" TIN in the Project Explorer to make it active.

2.   Expand the "⬚ elevs" dataset under "⬚ 2D Scatter Data Folder" in the Project Explorer.

This dataset contains two sets of data: a set of elevations for the bottom of the upper aquifer, and a set for the bottom of the lower aquifer. First interpolate the elevations for the bottom of the upper aquifer.

3.   Select the "⬚ bot of layer 1" dataset to make it active.

To interpolate from the scatter points to the active TIN, do the following:

4.   Right-click on the "elevs" ⬚ scatter set and select *Interpolate To* | **Active TIN** to bring up the *Interpolate → Object* dialog.

5.   Click **OK** to accept the defaults and close the *Interpolate → Object* dialog.

Finally, interpolate the elevations for the bottom TIN:

6.   Select the "⬡ bottom lower aquifer" TIN in the Project Explorer to make it active.

7.   Select the "⬚ bot of layer 2" dataset to make it active.

8.   Right-click on the "elevs" ⬚ scatter set and select *Interpolate To* | **Active TIN** to bring up the *Interpolate → Object* dialog.

9.   Click **OK** to accept the defaults and close the *Interpolate → Object* dialog.

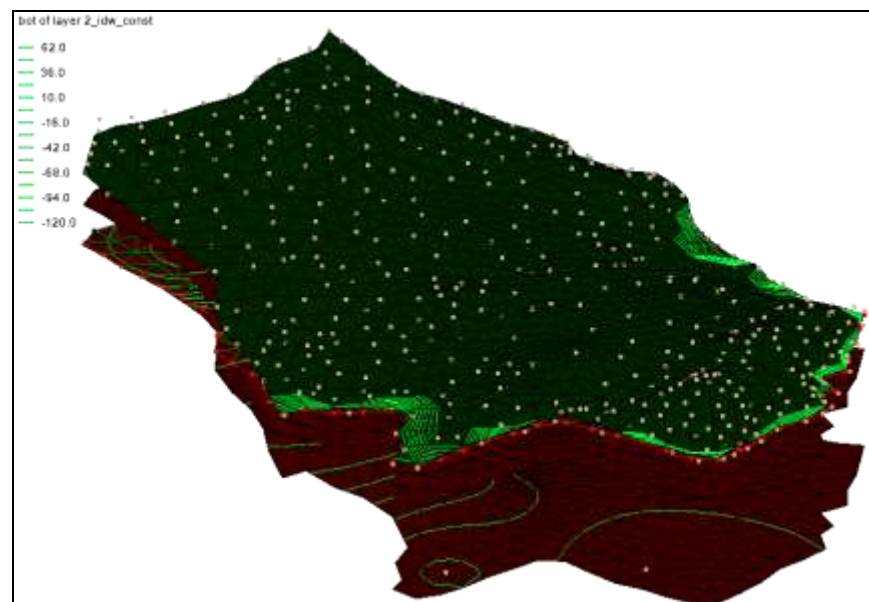At this point, the correct elevations on all three TINs should be visible (Figure 7).

Figure 7   All three TINs with elevations interpolated

3.6   Building the 3D Mesh

Now construct the 3D mesh using the horizon method by doing the following:

1. Click on the "⊞ TIN Data" folder in the Project Explorer.

2. Select *TINs* | **Horizons → 3D Mesh…** to bring up the *Horizon Elevations* page of the *Horizons to Mesh* dialog.

3. Click **Next** to accept the defaults and go to the *Top and Bottom Elevations* page of the *Horizons to Mesh* dialog.

4. In the *Top elevation* section, select *Tin elevations* and then "terrain" from the TIN Data tree below that.

5. In the *Bottom elevation* section, select *Tin elevations* and then "bottom lower aquifer" from the TIN Data tree below that.

6. Click **Next** to go to the *Build Mesh* page of the *Horizons to Mesh* dialog.

7. In the *Meshing options* section, turn on *Refine elements* and *Subdivide material* layers.

8. Select *Refine all elements* below *Refine elements*.

9. Turn on *Subdivide material layers*, then select *Target layer thickness* below that.

10. Enter "10.0" in the *Max layer thickness* column of the *Upper Aquifer* row.

11. Enter "20.0" in the *Max layer thickness* column of the *Lower Aquifer* row.

12. Click **Finish** to close the *Horizons to Mesh* dialog.

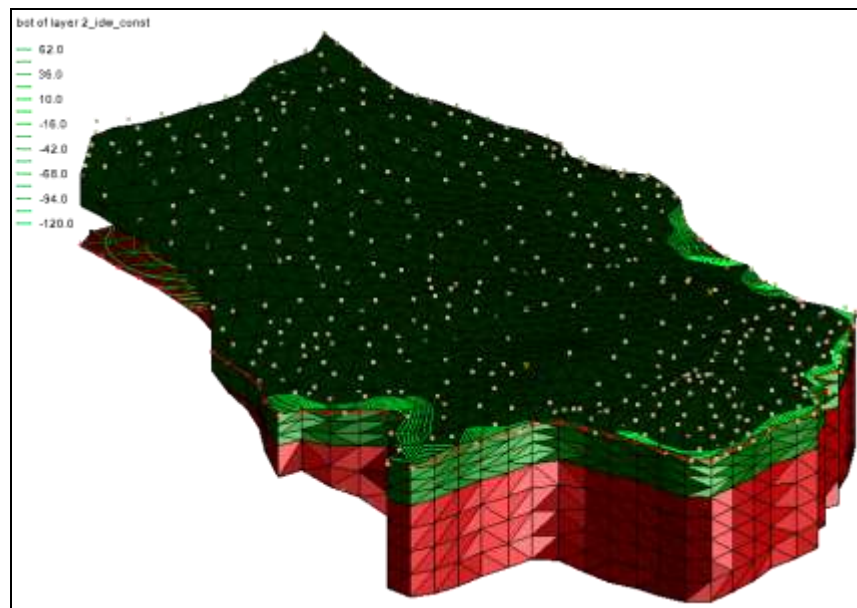A 3D mesh will now be constructed between the TINS from the options that were selected (Figure 8).

Figure 8   The 3D mesh between the TINs

4   Converting the Conceptual Model

Before continuing, unclutter the display by hiding the objects that are no longer needed. Hide everything in the Project Explorer except for the feature objects and the 3D mesh by doing the following:.

1. Turn off " TIN Data", " 2D Mesh", " 2D Scatter Data", and " GIS Layers".

Now 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.

2. Right-click " mesh (1)" under " 3D Mesh Data" and select **New FEMWATER** to bring up the *FEMWATER Run Options* dialog.

3. Click **OK** to accept the defaults and close the *FEMWATER Run Options* dialog.

4. Right-click on " femmod" in the Project Explorer and select **Map to →
    FEMWATER** to bring up the *Map → Model* dialog.

5. Click **OK** to accept the defaults and close the *Map → Model* dialog. Since the model has only one coverage, the choice here does not make a difference.

A set of symbols should appear indicating that the boundary conditions have been assigned (Figure 9).
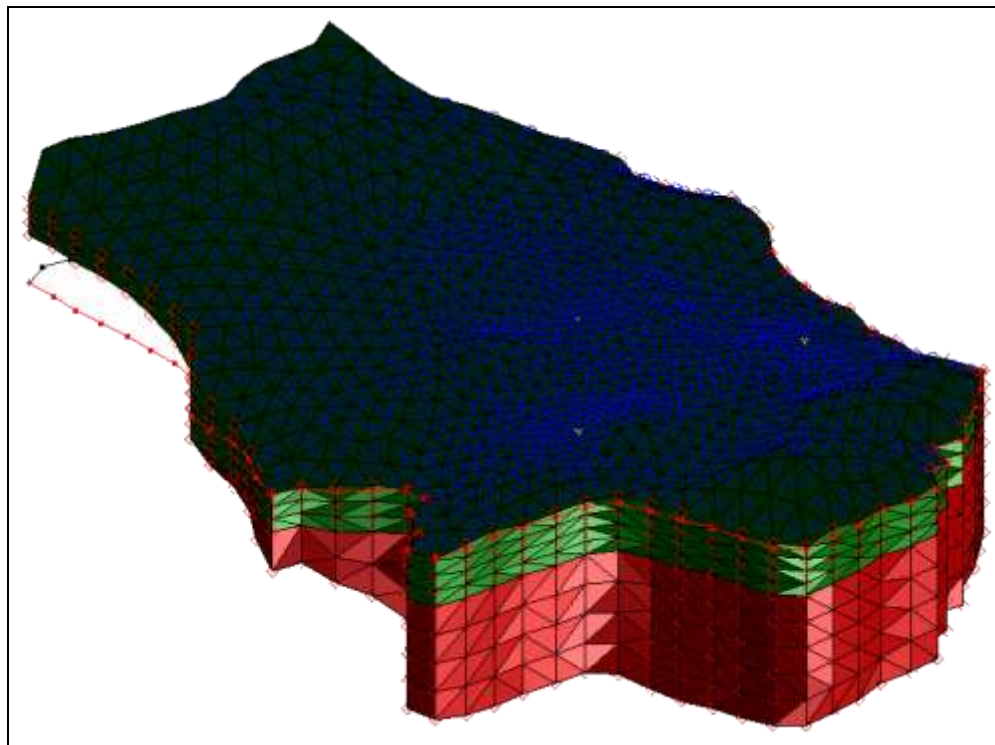

Figure 9   Conceptual model converted to 3D mesh FEMWATER model

5   Selecting the Analysis Options

The next is to switch to the 3D Mesh module and select which analysis options to use.

5.1   Entering the Run Options

To set up a steady state flow simulation, do the following:

1. Select *FEMWATER* / **Run Options…** to open the *FEMWATER Run Options* dialog.

2. Select "Flow only (10)" from the *Type of simulation* drop-down.

3. In the *Solver options (OP2)* section, select "Steady state (0)" from the *Steady-state vs. transient (KSSF)* drop-down.

The problem to be solved 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. Select "Nodal/Nodal (11)" from the *Quadrature (IQUAR)* drop-down.

5. Click **OK** to close the *FEMWATER Run Options* dialog.

5.2   Setting the Iteration Parameters

Adjust the iteration parameters next by doing the following:

1. Select *FEMWATER* / **Iteration Parameters…** to bring up the *FEMWATER Iteration Parameters* dialog.

2. In the *Flow simulation* section, enter "100" for the *Max iterations for non-linear equation*.

3. Enter "1000" for the *Max iterations for linear equation*.

4. Enter "0.003" for the *Steady-state convergence criterion*.

5. Enter "0.003" for the *Transient convergence criterion*.

6. Click **OK** to close the *FEMWATER Iteration Parameters* dialog.

5.3   Selecting Output Control

The next step is to select the output options. Choose to output a pressure head file only.

1. Select *FEMWATER* / **Output Control…** to open the *FEMWATER Output Control* dialog.

2. In the *Save options* section, turn off *Save nodal moisture content file (3), Save velocity file (4),* and *Save flux file (2).*

3. Turn on *Save pressure head (.phd) (1)* file.

4. Click **OK** to close the *FEMWATER Output Control* dialog.

5.4   Defining the Fluid Properties

Finally, define the fluid properties.

1. Select *FEMWATER* / **Fluid Properties…** to open the *FEMWATER Fluid Properties* dialog.

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

2. Enter "1000.0" for the *Density of water*.

3. Click **OK** to close the *FEMWATER Fluid Properties* dialog.

6   Defining Initial Conditions

Because FEMWATER uses non-linear equations to model flow in the unsaturated zone, it 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 dataset is generated by interpolating head values from the scatter points to the nodes of the 3D mesh. Finally, the pressure head dataset is created by subtracting the node elevations from the total head values.

6.1   Digitizing the Scatter Point Set

To define the initial condition, a small set of points is used at the expected elevation of the computed water table surface. In the interest of time, these points have already been created and are already in the project. The "Geostatistics 2D" tutorial discusses the creation of scatter points in more detail.

Before digitizing the points, turn off the flux boundary condition display options and turn on and modify the labels for each point by doing the following:

1. Select *FEMWATER* / **BC Display Options** to open the *Display Options* dialog. Notice that "3D Mesh Data" is already selected in the list on the left.

2. On the *FEMWATER* tab, turn off *Flux fluid (CB1)*.

3. Select "2D Scatter Data" from the list on the left.

4. On the *2D Scatter Point Set* tab in the table below the note, click on the large button (not the down arrow button) to the right of "startheads" to bring up the *Symbol Attributes* dialog.

5. Enter "10" for the *Size*.

6. Click **OK** to close the *Symbol Attributes* dialog.

7. Turn on *Scatter point scaler values* and click the **123** button to the right to bring up the *Font* dialog.

8. Select "Arial" from the *Font* list.

9. Select "Regular" from the *Font style* list.

10. Select "11" from the *Size* list.

11. Turn on *Fill behind text* and select "Specified" from the *Color* drop-down.

12. Click the down arrow to the right of *Fill behind text* and select "White" from the list of colors.

13. Click **OK** to close the *Font* dialog.

14. Click **OK** to close the *Display Options* dialog. Notice that all the blue circles disappeared from the top of the 3D Mesh.

To view the scatter point set.

15. Switch to **Plan View** 📦.

16. In the Project Explorer, turn off the " 🏠 3D Mesh Data" folder.

17. Turn on the " ⁘ startheads" scatter point set and the " 🌐 GIS Layers" folder.

The scatter points should be located as shown in Figure 10. The value displayed to the right of each point shows its elevation.

Figure 10   Point locations and elevations

18. When finished viewing the scatter points, turn on the " 🏠 3D Mesh Data" folder in the Project Explorer.

19. Switch to **Oblique View** 🔷.

20. Click **Display Options** 🖥 to bring up the *Display Options* dialog.

21. Select "2D Scatter Data" from the list on the left.

22. On the *2D Scatter Point Set* tab, turn off *Scatter point scalar values*.

23. Click **OK** to close the *Display Options* dialog.

6.2   Creating the Dataset

Now a pressure head dataset must be created and saved in order to define the initial conditions. The path to this file will be provided by GMS to FEMWATER when the simulation is launched.

1. Select *FEMWATER* / **Initial Conditions…** to open the *FEMWATER Initial Conditions* dialog.

2. In the *Pressure head (ICH)* section, select *Spatially variable. Read from dataset file (1)*, then click **Generate IC…** to bring up the *Generate Pressure Head Initial Condition* dialog.

3. Select "startheads" from the *Active 2D scatter point set* drop-down.

4. Enter "-60.0" for the *Minimum pressure head*.

5. Click **OK** to close the *Generate Pressure Head Initial Condition* dialog and bring up the *Select name and path of pressure head file* dialog.

6. Enter "starthd.phd" for the *File name*.

7. Select "FEMWATER Pressure Head Files (*.phd)" from the *Save as type* drop-down.

8. Click **Save** to save the new pressure head file and close the *Select name and path of pressure head file* dialog.

9. Click **OK** to exit the *FEMWATER Initial Conditions* dialog.

7   Defining the Material Properties

The final step in setting up the model is to define the material properties. A hydraulic conductivity value and a set of unsaturated zone curves must be set for each aquifer. Do the following to define the material properties for the aquifers:

1. Select *Edit* / **Materials…** to open the *Materials* dialog.

2. On the *Upper Aquifer* row, enter "3.0" in the *Kxx*, *Kyy* and *Kzz* columns.

3. With any cell in the *Upper Aquifer* row selected, click **Generate Unsat Curves…** to open the *van Genuchten Curve Generator* dialog.

4. Select "van Genuchten equations" from the *Curve type* drop-down.

5. Enter "1.8" for *Max. height above water table*.

6. Select *Preset parameter values* and select "Silt" from the drop-down.

7. Click **Compute Curves**. Notice that there are three curves below that button now.

8. Click **OK** to close the *van Genuchten Curve Generator* dialog.

9. On the *Lower Aquifer* row, enter "9.0" in the *Kxx*, *Kyy* and *Kzz* columns.

10. With any cell in the *Lower Aquifer* row selected, click **Generate Unsat Curves…** to open the *van Genuchten Curve Generator* dialog.

11. Accept the defaults and click **Compute Curves** to generate the three curves.

12. Click **OK** to close the *van Genuchten Curve Generator* dialog.

13. Click **OK** to exit the *Materials* dialog.

8   Saving and Running the Model

Save and run the model.

1. **Save** ![save icon] the project.

2. Select *FEMWATER* / **Run FEMWATER…** to bring up the *FEMWATER* model wrapper dialog.

The FEMWATER window will display information about the progress of the model convergence. The model should converge within a few minutes.

3. When the model converges, turn on *Read solution on exit* and click **Close** to exit the *FEMWATER* model wrapper dialog.

9   Viewing the Solution

9.1   Viewing Head Contours

First, view a color fringe plot.

1. Click **Display Options** ![display options icon] to open the *Display Options* dialog.

2. Select "3D Mesh Data" from the list on the left.

3. On the *3D Mesh* tab in the lower section, turn on *Contours*.

4. Click the **Options…** button next to *Contours* to open the *Dataset Contour Options – 3D Mesh – pressure_head* dialog.

5. In the *Contour method* section, 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.

6. Click **OK** to exit the *Dataset Contour Options – 3D Mesh – pressure_head* dialog.

7. On the *FEMWATER* tab, turn off *Specified Head (DB1)*.

8. Click **OK** to close the *Display Options* dialog.

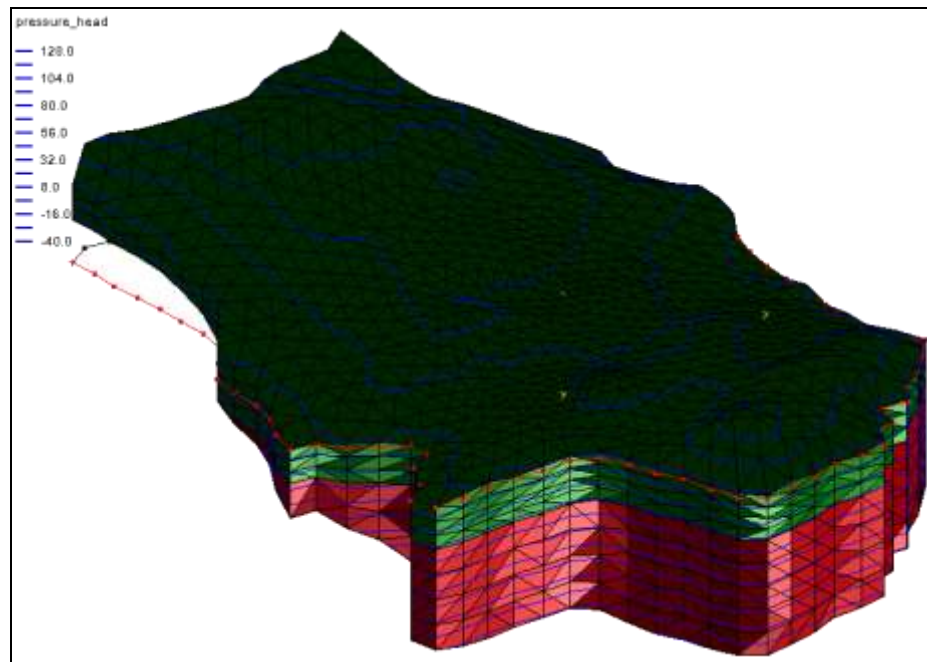The Graphics Window should appear similar to Figure 11.

Figure 11   Head contours marked in blue

9.2   Viewing a Water Table Isosurface

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

1. In the Project Explorer, expand the " femmod (FEMWATER)" solution and select the " pressure_head" dataset to make it active.

2. Click **Display Options** to open the *Display Options* dialog.

3. Select "3D Mesh Data" from the list on the left.

4. On the *3D Mesh* tab in the upper section, turn off *Element edges*.

5. In the lower section, turn off *Contours* and turn on *Isosurfaces*.

6. Click the **Options…** button to open the *Isosurface Options* dialog.

7. Below the first section, enter "1" for the *Number of isosurfaces*.

8. In row *1* of the spreadsheet, enter "0.0" in the *Upper Value* column.

9. On row 2, turn on *Fill between* in order to fill between "0.0" and the maximum value.

10. In the section on the right, turn on *Isosurface faces* and *Contour specified range*.

11. Enter "0.0" for the *Minimum* and "140.0" for the *Maximum*.

12. Click **OK** to close the *Isosurface Options* dialog.

13. Click **OK** to exit the *Display Options* dialog.
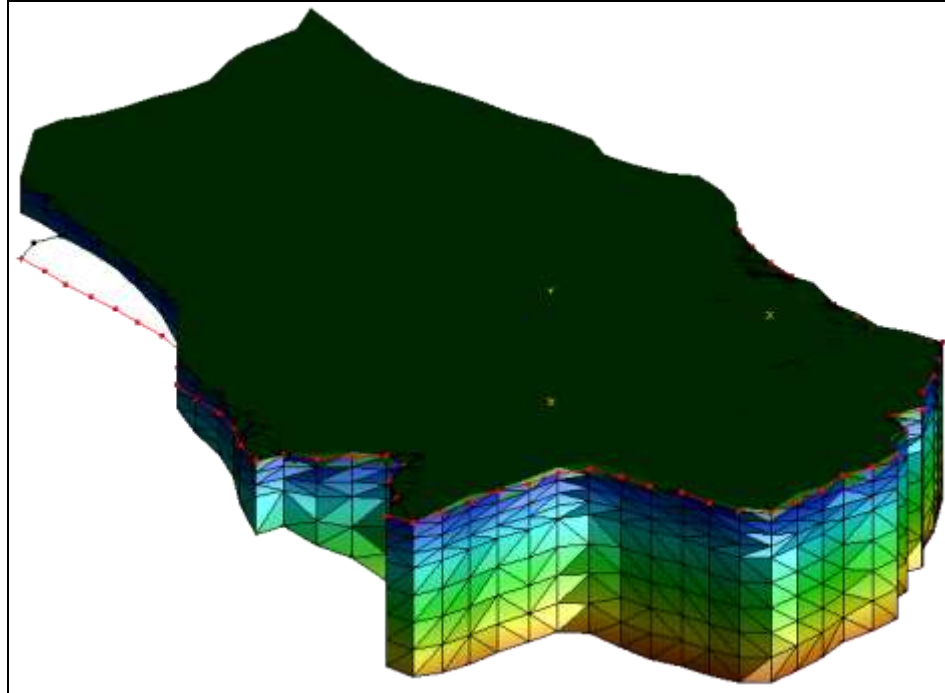
The Graphics Window should appear similar to Figure 12.

Figure 12   Water table isosurface

9.3   Draping the TIFF Image on the Ground Surface

The TIFF image can be draped on the terrain surface to illustrate the spatial relationship between the computed water table surface and the ground surface. To do this, first unhide the top TIN.

1. Turn on and expand the "⬡ terrain" TIN in the Project Explorer.

2. Select the "⊞ z_idw_const" dataset to make it active.

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

3. Click **Display Options** 🖥 to open the *Display Options* dialog.

4. Select "TIN Data" from the list on the left.

5. On the *TIN* tab in the top section, turn on *Texture map image to active tin*.

6. Click **OK** to close the *Display Options* dialog.

7. Use the **Rotate** 🔄 tool to click and drag horizontally to the left side of the screen until the other side of the model is visible (Figure 13).
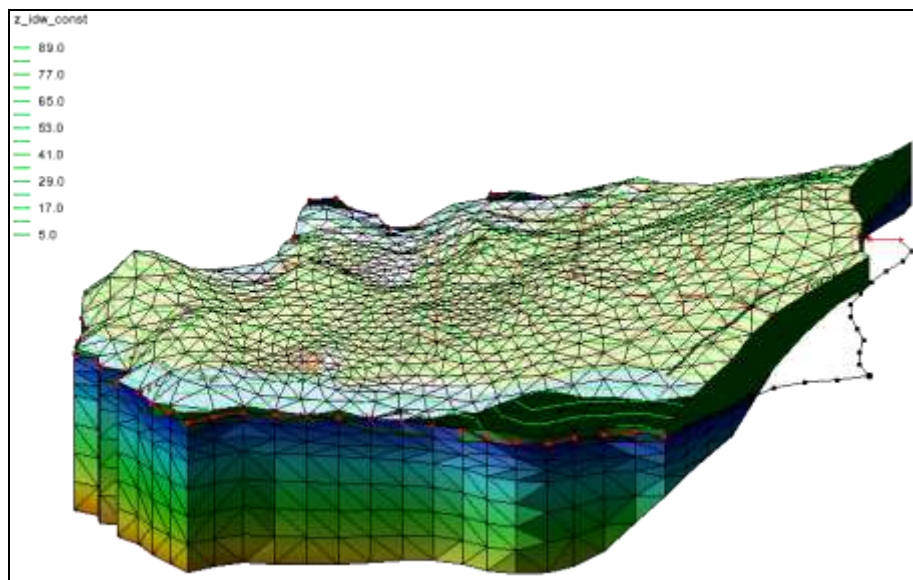
Figure 13   TIFF image draped and the 3D mesh rotated

Finally, try the smooth shade option.

8. Click **Display Options** 🖥 to open the *Display Options* dialog.

9. Select "Lighting Options" from the list on the left.

10. On the *Lighting* tab in the *Surface attributes for all lights* section, turn on *Smooth Edges*.

11. Click **OK** to close the *Display Options* dialog.

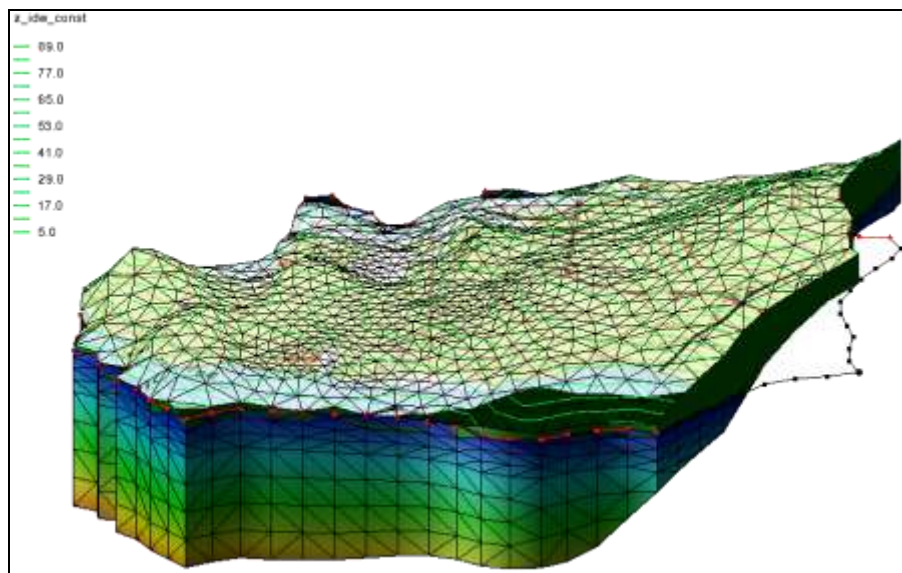Notice that the water table isosurface contour is smoother now (Figure 14).

Figure 14   TIFF image draped and the 3D mesh rotated, with smoothing

10   Conclusion

This concludes the "FEMWATER – Flow Model" tutorial. The following concepts were discussed and demonstrated:

- 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.