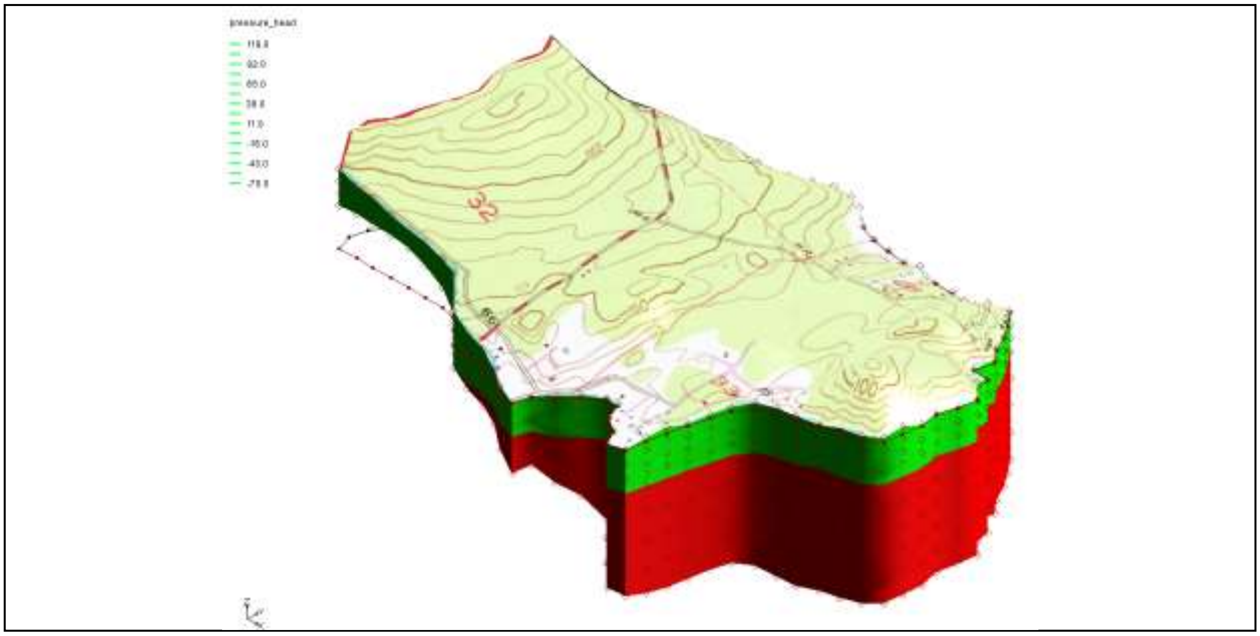
GMS

v. 10.0

GMS 10.0 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

- Feature Objects
- Geostatistics 2D
- Stratigraphy Modeling – TINs

Required Components

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

Time

- 45-65 minutes

AQUAVEO™

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 FEMWATER model to simulate flow only.

1.1 Outline

Here are the steps to the tutorial:

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.

2 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^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. The objective of this tutorial is to create a steady state flow model of the site.

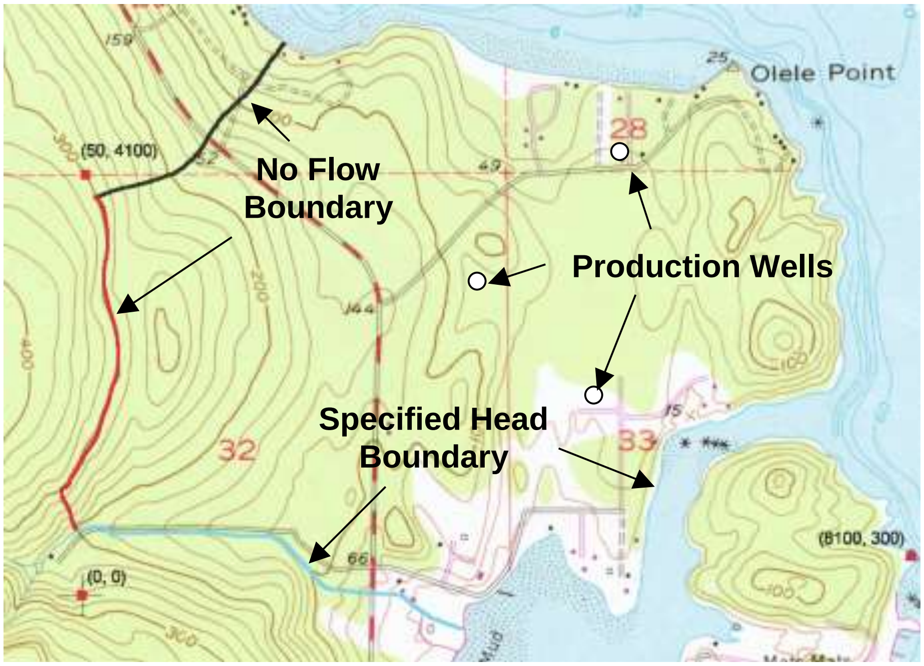


Figure 1 Site to be modeled with FEMWATER

3 Getting Started

Do the following to get started:

1. If GMS is not running, launch GMS.
2. If GMS is already running, 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. The conceptual model approach will be used for this tutorial.

4.1 Importing the Background Image

Before creating the feature objects, the user 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. Do the following 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."
4. Select the **Open** button.

4.2 Saving With a New Name

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.

Save the work periodically throughout the tutorial by clicking the **Save** button.

4.3 Defining the Units

The user will now define the units. GMS uses the units that the user selects to plot helpful labels next to input edit fields.

1. Select the *Edit* | **Units** command to open the *Units* dialog.
2. Check to ensure that the *Length* unit is "m" (meters) for horizontal and vertical units by clicking the "**…**" button next to the Length field.

3. Change both the horizontal and vertical units to “meters.”
4. Click **OK**.
5. Ensure that the *Time* unit is set to the default: “d” (days).
6. Change the *Mass* unit to “kg” (kilograms). The remaining units are for a transport simulation and can be ignored.
7. Select the **OK** button to exit the *Units* dialog.

4.4 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.
2. From the pop-up menu, select the *New* | **Conceptual Model** command to open the *Conceptual Model Properties* dialog.
3. Change the name to “femmod.”
4. Change the *Type* to “FEMWATER.”
5. Click **OK**.
6. Right-click on the “femmod” conceptual model.
7. Select the **New Coverage** command from the pop-up menu to open the *Coverage Setup* dialog.
8. Change the *Coverage Name* to “femwater.”
9. Turn on the following properties:
   - *Flow BC*
   - *Wells*
   - *Refinement*
   - *Meshing options*
10. Select the **OK** button.

4.5 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.
2. Select the **Create Arc** tool.
3. 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.

4. Create an arc for the coastline boundary. Once again, be sure the arc is connected to the other boundary arcs.

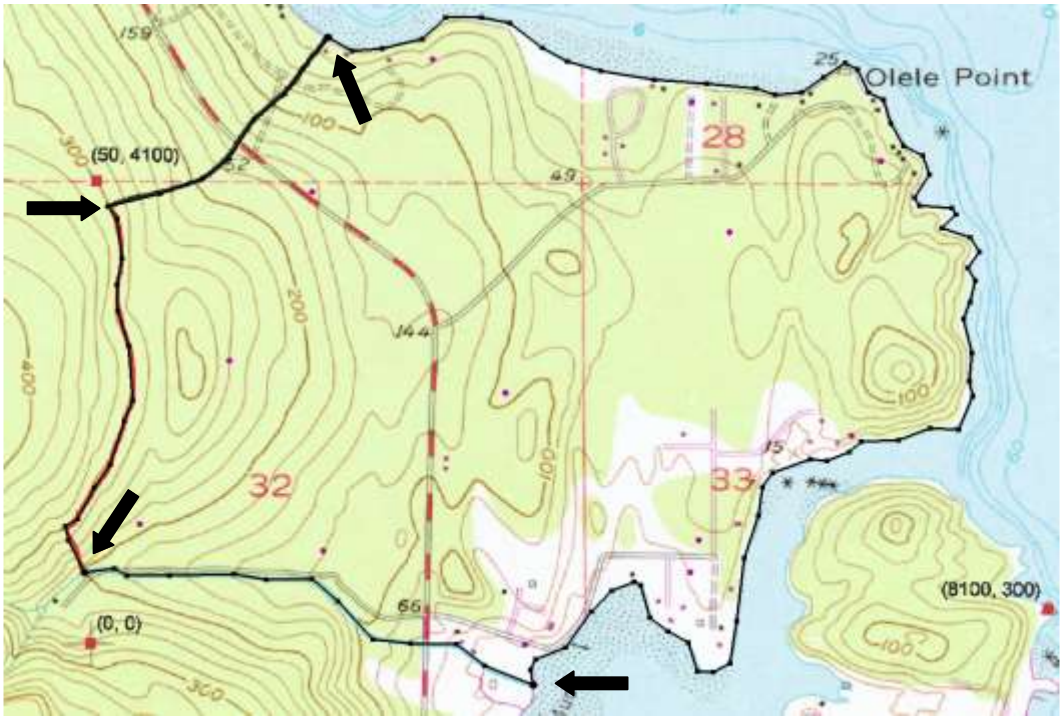


Figure 2 Boundary arcs. Arrows show endpoints of arcs

4.6 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, the user needs 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 to open the *Redistribute Vertices* dialog.
4. Change the *Average spacing* to "90."
5. Select the **OK** button.

4.7 Defining the Boundary Conditions

Now that the arcs are defined, the user 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, the user needs 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 to open the *Attribute Table* dialog.
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, the user does 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.

4.8 Building the Polygon

Now that the arcs are defined, the user is 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, the user will use one polygon since the model has a single recharge value.

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

4.9 Assigning the Recharge

Next, the user 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, the user 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.

4.10 Creating the Wells

The final step in defining the conceptual model is to create the wells. It is necessary 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 that was 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 to open the *Attribute Table* dialog.

First, the user will mark the point as a well point. For a well, the user defines 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 this case, there will only be one node.

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

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

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

5 Building the 3D Mesh

At this point, the conceptual model is complete, and the user is 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, the user will first create a 2D "projection" mesh using the feature objects in the conceptual model. The user 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. The user will then create the 3D elements by using the **Horizons → 3D Mesh** command.

5.1 Defining the Materials

Before building the mesh, the user 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.

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, the user will 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. The user 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* | *Convert to* **| TIN** command to open the *Properties* dialog.
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* | *Convert to* **| TIN** command to open the *Properties* dialog.
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:

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

2. Enter "bottom lower aquifer" for the name of the TIN.
3. In the *Tin material* list, select "Lower Aquifer" (this value is ignored for the bottom TIN).
4. Enter "0" for the *Horizon id*.
5. Select **OK**.

5.4 Interpolating the Terrain Data

Next, the user 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, the user will need to make sure the top TIN is the active TIN.

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

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

2. Expand the "2D Scatter Data" folder.
3. Select the scatter point set entitled "terrain" in the Project Explorer to make it the active scatter set.
4. Select the *Interpolation* | **Interpolation Options** command to open the *2D Interpolation Options* dialog.
5. Select the **Options** button next to the *Inverse distance weighted* item. This will open the *2D IDW Interpolation Options* dialog.
6. In the *Nodal function* section, make sure the *Constant (Shepard's method)* option is selected.
7. Select **OK** to exit the *2D IDW Interpolation Options* dialog.
8. Select **OK** to exit the *2D Interpolation Options* dialog.

To interpolate from the scatter points to the TIN.

9. In the Project Explorer, right-click on the *terrain* scatter set .
10. From the pop-up menu, select the *Interpolate To* | **Active TIN** command.
11. Select the **OK** button.

To view the interpolated elevations:

12. Select the **Oblique View** macro.
13. Select the **Display Options** macro.
14. Enter a *Z magnification* factor of "4.0."
15. Select the **OK** button.

5.5 Interpolating the Layer Elevation Data

Next, the user 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, the user needs 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 datasets: one set of elevations for the bottom of the upper aquifer and one set for the bottom of the lower aquifer. The user will first interpolate the elevations for the bottom of the upper aquifer.

3. Select the dataset entitled "elevs" in the Project Explorer to make it the active scatter point set.
4. If necessary, expand the "elevs" item so that the user can see the datasets associated with the scatter point set.
5. Select the "bot of layer 1**"** dataset 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.
7. From the pop-up menu, select the *Interpolate To* | **Active TIN** command.
8. Select the **OK** button.

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

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

5.6 Building the 3D Mesh

The user is 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. Use the defaults on the first page of the wizard by selecting **Next**.
4. In the *Top elevation* section, select *Tin elevations.*
5. Then select the "terrain" TIN from the *TIN Data* list.
6. Also, in the *Bottom elevation* section, select *Tin Elevations.*
7. Then select the "bottom lower aquifer" TIN from the *TIN Data* list.
8. Select **Next**.
9. Turn on the *Refine elements* option.

10. Select *refine all elements*.
11. Turn on the *Subdivide material layers* option.
12. Make sure *Target layer thickness* is selected.
13. Change the *Upper Aquifer* maximum layer thickness to “10.”
14. Change the *Lower Aquifer* maximum layer thickness to “20.”
15. Select the **Finish** button.

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

6 Hiding Objects

Before continuing, the user will unclutter the display by hiding the objects that the user is finished with. The user will hide everything but the feature objects and the 3D mesh.

To hide the TINs:

1. Uncheck all three TINs in the Project Explorer.

The user can also hide the scatter points:

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

7 Converting the Conceptual Model

Now the user is 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” item.
2. Select the **New FEMWATER** command from the pop-up menu.
3. Select the **OK** button.
4. In the Project Explorer, right-click on the “femmod” conceptual model in the “Map Data” folder.
5. Select the **Map to → FEMWATER** command from the pop-up menu.
6. Select **OK** at the prompt (since the model has only 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, the user will switch to the *3D Mesh* module and select the analysis options.

8.1 Entering the Run Options

First, the user will indicate that the user wishes to run a steady state flow simulation:

1. Select the *FEMWATER* | **Run Options** command to open the *FEMWATER Run Options* dialog.
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 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. In the *Quadrature selection* section, select the "Nodal/Nodal" option.
5. Select the **OK** button.

8.2 Setting the Iteration Parameters

Next, the user 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.

8.3 Selecting Output Control

Next, the user will select the output options. The user will choose to output a pressure head file only.

1. Select the *FEMWATER* | **Output Control** command to open the *FEMWATER Output Control* dialog.
2. Make sure that the following options are turned off:
   - *Save nodal moisture content file*
   - *Save velocity file*
   - *Save flux file*
3. Select the **OK** button.

8.4 Defining the Fluid Properties

Finally, the user will define the fluid properties.

1. Select the *FEMWATER* | **Fluid Properties** command to open the *FEMWATER Fluid Properties* dialog.

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

2. Ensure that “1000” is entered 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 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.

9.1 The Scatter Point Set

To define the initial condition, the user 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 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, the user will turn off the flux boundary condition display options.

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

To view the scatter point set.

4. Switch to the **Plan View** button.
5. In the Project Explorer, uncheck the “2D Mesh Data” and “3D Mesh Data” folders.
6. Turn on the scatter point set named “startheads” .

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

7. When finished viewing the scatter points, turn the “3D Mesh Data” folder back on by checking it in the Project Explorer.

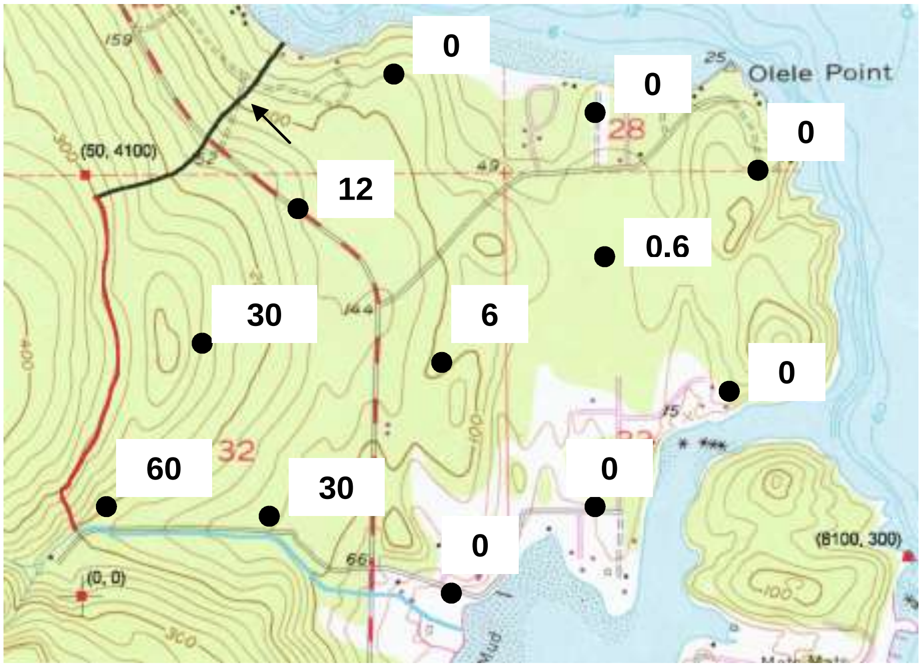


Figure 3. Locations and elevations for new points.

9.2 Creating the Dataset

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

1. Select the *FEMWATER* | **Initial Conditions** command to open the *FEMWATER Initial Conditions* dialog.
2. In the *Pressure head (ICH)* section on the upper left side of the dialog, select the *Spatially variable. Read from dataset file* option.
3. Select the *Generate IC* button.
4. Select "startheads" 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."
8. Select the **Save** button.
9. Select the **OK** button to exit the *FEMWATER Initial Conditions* dialog.

10 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 to open the *Materials* dialog.

The user needs to enter a hydraulic conductivity and a set of unsaturated zone curves for each aquifer.

Do the following to define the material properties for the upper aquifer:

2. For the *Upper aquifer* material enter a value of "3.0" for *Kxx*, *Kyy* and *Kzz*.
3. With one of the cells in the "Upper Aquifer" material row selected, select the **Generate Unsat Curves** button to open the *van Genuchten Curve Generator* dialog.
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.

Do the following to define the material properties for the lower aquifer:

9. For the *Lower aquifer* enter a value of **9.0** for *Kxx*, *Kyy* and *Kzz*.
10. With one of the cells in the "Lower Aquifer" material row selected, select the *Generate Unset Curves…* button.
11. Select the **Compute Curves** button.
12. Select the **OK** button.
13. Select the **OK** button to exit the *Materials* dialog.

11 Saving and Running the Model

Save and run the model.

1. Select the **Save** button.
2. Select the *FEMWATER* | **Run FEMWATER** command.

The FEMWATER window should appear and the user 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.

11.1 Viewing Head Contours

First, the user will view a color fringe plot.

1. Select the **Display Options** button to open the *Display Options* dialog.
2. Verify that the *Contours* option is turned on.
3. Select the **Options** button next to the *Contours* item to open the *Dataset Contour Options – 3D Mesh – pressure head* dialog.
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.

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 the user caps the iso-surface on the side greater than zero, the user 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 the “pressure_head” dataset to make it the active dataset.
3. Select the **Display Options** button to open the *Display Options* dialog.
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 to open the *Iso-surface Options* dialog.
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 **OK**.
14. Select **OK** to exit the *Display Options* dialog.
15. Select the **Oblique View** macro.

11.3 Draping the TIFF Image on the Ground Surface

Finally, the user 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, the user 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.
3. Select the “z_idw_const” dataset.

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

4. Select the **Display Options** button to open the *Display Options* dialog.
5. Turn on the *Texture map image to active tin* option.

6. Select **OK**.
7. Select the **Rotate** tool.
8. 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, the user will try the smooth shade option.

9. Select the **Display Options** button to open the *Display Options* dialog.
10. Select the **Lighting Options** item in the list box.
11. Select the *Smooth Edges* option on the *Lighting Options* tab.
12. Select the **OK** button.

12 Conclusion

This concludes the tutorial. Here are the key concepts 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.