Visual MODFLOW 2.8.2

Student Version Tutorial Guide:

Introduction

The following laboratory exercise consists of a number of instruction modules that correspond to some of the lecture material that will be covered during the course. This exercise includes information on:

Building a Groundwater Flow Model - Modules 1 - 5

Calibrating a Groundwater Flow Model - Modules 6 & 7

The purpose of this exercise is to introduce you to the basic steps necessary to build and calibrate a groundwater flow model using Visual MODFLOW. The model you will build for this exercise is a simplified example of an actual project site that will allow you to easily examine the various features and capabilities of using Visual MODFLOW to assign input parameters, run the simulations, calibrate the model, and visualize the results.

The site that you will be modeling is the Sunrise Fuel Supplies Site. A site map of the Sunrise Fuel Supplies Co. and surrounding area is shown in Figure 1.

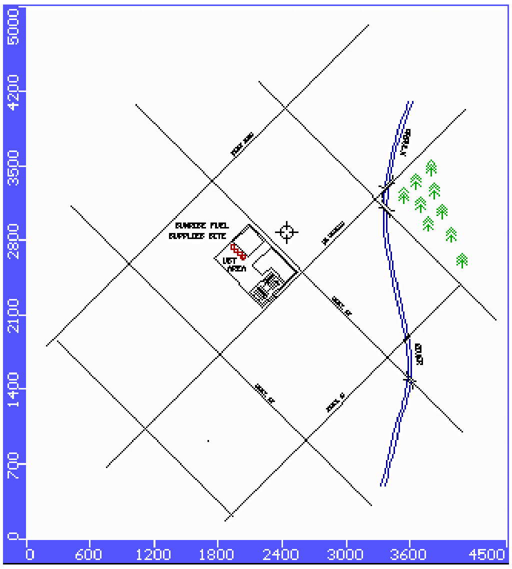


Figure 1: Map of Sunrise Fuel Supplies Site and Surrounding Area

Problem Description

Approximately 2 years ago the company inventory manager discovered that fuel quantities sold did not correspond to the volume of fuel purchased for re-sale. Initially, it was suspected that some employees were stealing from the company. However, it was recently determined that leaks in the fuel storage tanks were responsible for the fuel volume discrepancies. This problem was promptly reported to state EPA officials and a detailed site investigation was ordered to determine the full vertical and horizontal extents of contamination.

The field investigation activities included the following:

- drilling several boreholes to characterize the underlying geology
- performing slug tests at each borehole to characterize the soil hydraulic properties
- performing a well pumping test to characterize the soil hydraulic properties
- monitoring the water levels and chemistry for six months

Site Description

The site is relatively flat with an average elevation of 200 feet (60 m) above mean sea level (AMSL). The local topography slopes from northwest to southeast towards the Proulx River, which is located about 2000 feet (608 m) east of the site. The Proulx River flows from north to south with a surface water elevation that ranges from 174 - 176 ft (52 to 53 m) AMSL in the north, to 169 to 171 ft (51 to 52m) AMSL in the south (depending on the time of year). The average depth of the Proulx River is approximately 5 ft (1.5 m).

The borehole drilling program revealed that the site is underlain by a relatively homogeneous silty sand and gravel aquifer extending to a depth of about 50 feet (15 m) below ground surface. The results of slug test and pumping test analyses indicate a soil hydraulic conductivity ranging from 0.5 - 15 ft/day (0.1 to 4.5 m/d). Water level monitoring data at the site indicates a flat water table, which generally follows the surface topography in the area, with a gentle slope towards the Proulx River throughout most of the year.

The results of the water quality analysis data showed high levels of BTEX contaminants in the groundwater beneath the site. Based on the results of the field investigation, a groundwater modeling study was recommended to evaluate the future extent of contamination on the site and to determine the feasibility of a pump and treat remediation system to capture and remove the plume.

Learning Objectives

[1] To conceptualize a model and solution

[2] To dimension and build a model grid

[3] To input model properties and boundary conditions

[4] To assign particle tracking locations

[5] To run MODFLOW and MODPATH

[6] To visualize the model results

[7] To assign a pumping well

[8] To refine the model grid

[9] To calibrate a model to steady-state, non-pumping conditions

[10] To calibrate a model to steady-state, pumping conditions

Model Conceptualization

The first step in this laboratory exercise will be to develop a model conceptualization and identify model boundaries. From the cross-section in Figure 2, we can see that groundwater flow in the silty sand and gravel aquifer discharges to the Proulx River. Along the west side of the site the groundwater table has fluctuated between an elevation of 193 to 198 ft (58 to 60 m) AMSL throughout the year, with an average elevation of 196.5 ft (59.7 m) AMSL. The groundwater flow direction across the site is predominantly west to east.

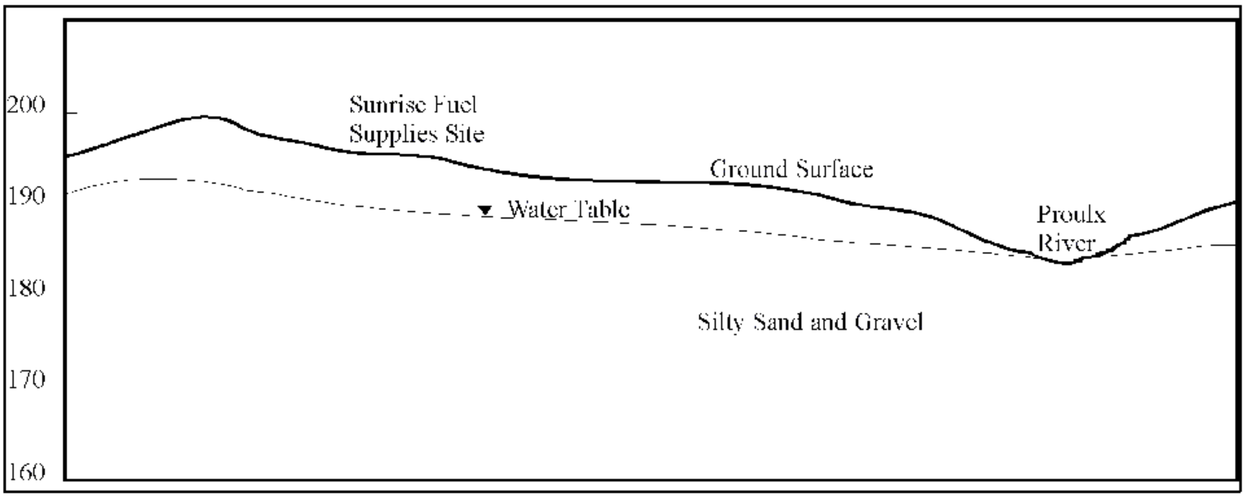


Figure 2: Cross-section through the Sunrise Fuel Supplies Site

From this information, the model boundaries and boundary conditions were determined. The selected model boundaries are illustrated in Figure 3.

A constant head (196.5 ft, or 59.7 m, AMSL) boundary condition was selected to represent the western boundary of the model. This boundary has been selected a significant distance away from area of interest such that it would not likely be influenced by any local stresses to the groundwater flow system initiated within the site property (such as a remediation pumping well).

The eastern boundary was selected along the Proulx River since the river acts as a local groundwater flow divide whereby the groundwater in the aquifer discharges to the river. A river boundary condition will be used to simulate the river.

The north and south boundaries were selected as no-flow "streamline boundaries" since the majority of the groundwater flow is parallel to these boundaries. These hydraulic boundaries were selected a significant distance from the area of interest so that they would not be influenced by any local stresses to the groundwater flow system within the site property

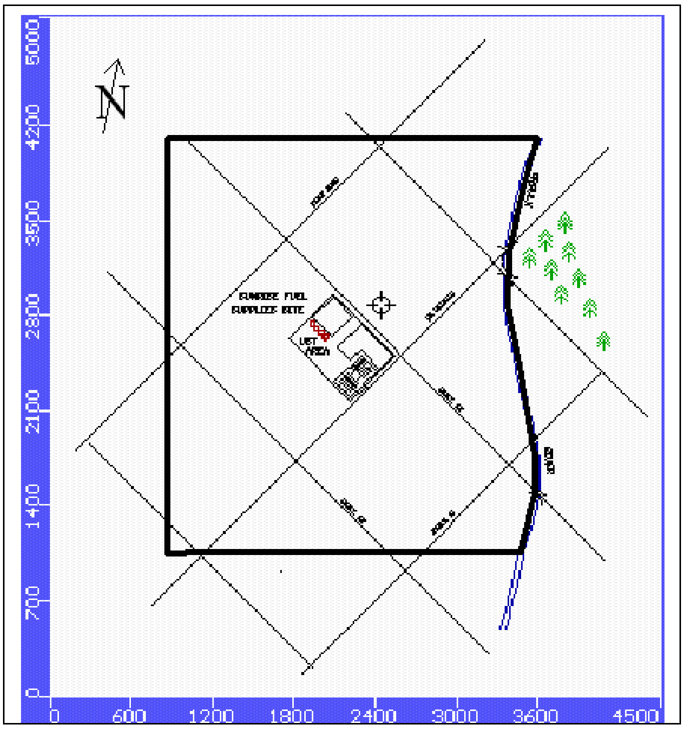


Figure 3: Model Domain Boundary for Sunrise Fuel Supplies Site

.

Terms and Notations

For the purposes of this tutorial, the following terms and notations will be used:

type: - to type in the given word or value

select - to click the left mouse button where indicated

⇔ - to press the <Tab> key

↵ - to press the <Enter> key

☞ - to click the left mouse button where indicated

☞☞ - to double-click the left mouse button where indicated

Starting Visual MODFLOW

On your Windows 95/98/NT desktop, you will see an icon for Visual MODFLOW.

☞☞ 

You will then be presented with the Visual MODFLOW opening screen and taken to the program's Main Menu area.

Module 1: Model Design and Input

Section 1: Dimension and Build a Model Grid

To start a new model,

☞	**File**	(from the top menu bar)
☞	**New**	(from the drop-down menu)

A **Create New Model** dialogue box will be displayed prompting you to enter the filename of the new Visual MODFLOW project.

type:	**Sunrise**	(in the box labeled File name)
☞	**[Save]**	

When the dialog box appears, you will use an AutoCAD **.DXF** as a background map for your model.

☞ ☑ **Create Model Using Base Map** (to select this option)

Next, you must specify the location and file name of the **.DXF** background map.

☞ **[Browse]**

Navigate to the **C:\VMODNT** directory and select the following file:

☞ **Sunrise.dxf**

☞ **[Open]**

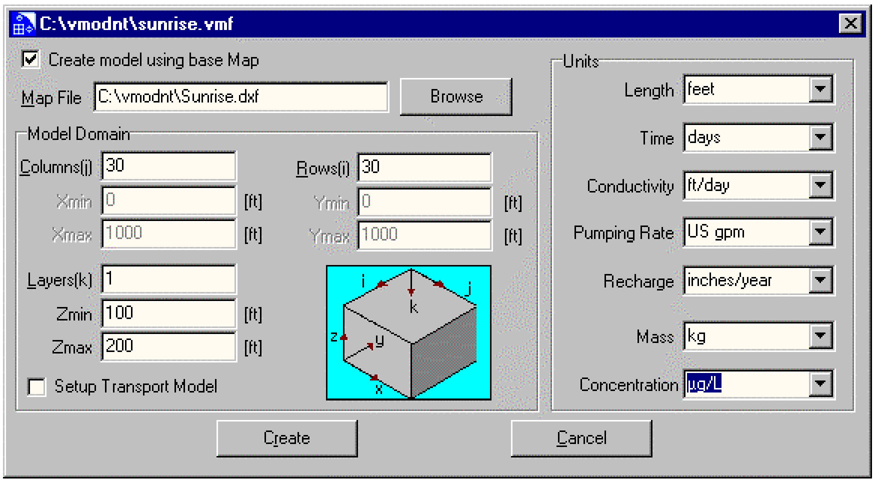
Next, under **Model Domain**, enter the model co-ordinate, layer and depth information in the appropriate boxes.

Columns (j):	**30 ⇔**
Rows (i):	**30 ⇔**
Layers (k):	**1 ⇔**
Zmin (ft):	**100 ⇔**
Zmax (ft):	**200**

Under the **Units** section, specify the following information using the mouse pointer and the individual drop-down lists.

Length:	**feet**
Time:	**days**
Conductivity:	**ft/day**
Pumping Rate:	**US gpm**
Recharge:	**inches/year**
Mass:	**kg**
Concentration:	**μg/l**

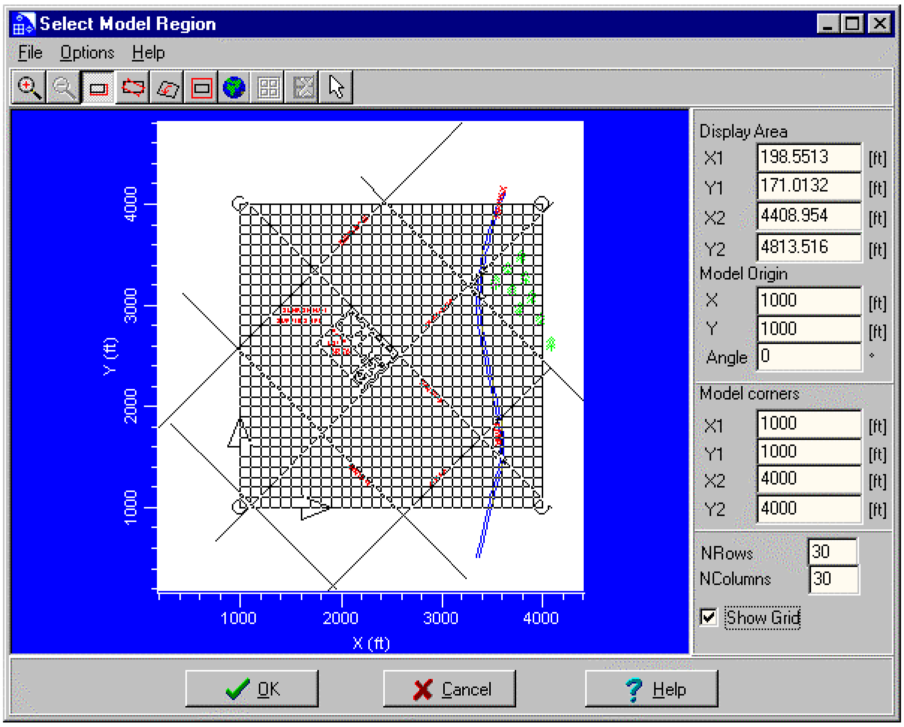
Your dialog box should now appear identical to the one shown below.



☞ **[Create]**

The **Select Model Region** dialogue box will be displayed.

Using the mouse pointer, left-click and hold one of the corners of the resizable rectangle overlying the **.DXF** background map and drag it open until it encloses the study area. Be sure to include the river on the right and the wooded area. Release the mouse button.



In the Model Origin and Model Corners boxes on the right, ensure that the values shown match the ones listed below. If they do not match, change them where appropriate.

Model Origin:

X: **1000 ⇔**

Y: **1000 ⇔**

Angle: **0 ⇔**

Model Corners:

X1: **1000 ⇔**

Y1: **1000 ⇔**

X2: **4000 ⇔**

Y2: **4000**

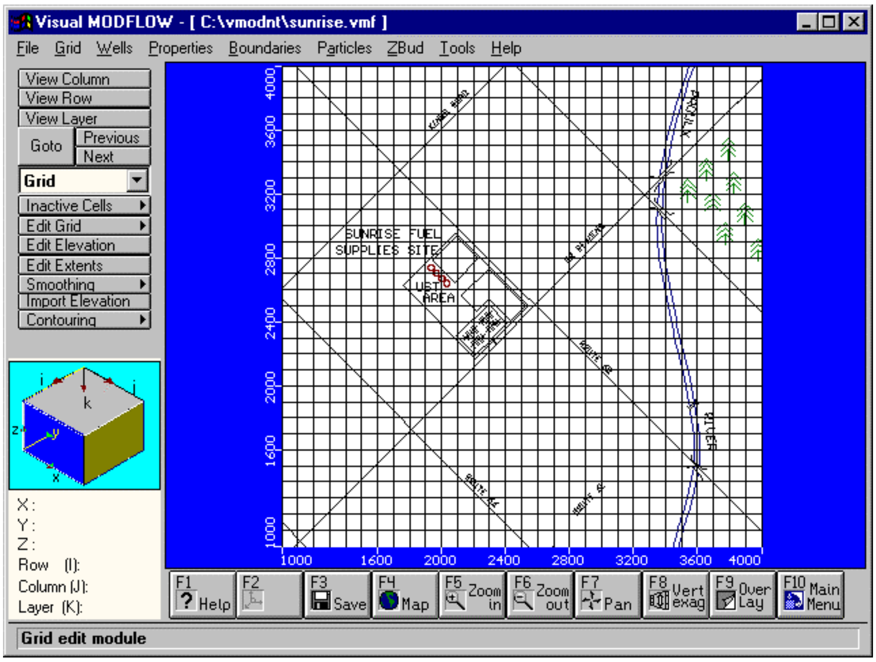
To display the model grid on the **.DXF** background map, in the lower right of the **Select Model Region** dialogue box,

☞ ☑ **Show Grid** (click to select)

Your display should now look similar to the figure at the top of the page.

☞ **[OK]** (to accept these values)

Visual MODFLOW will then construct a 30 X 30 X 1 finite difference grid with uniform grid spacing in both the X and Y directions and automatically opens the **Input Module** as shown in the figure below.



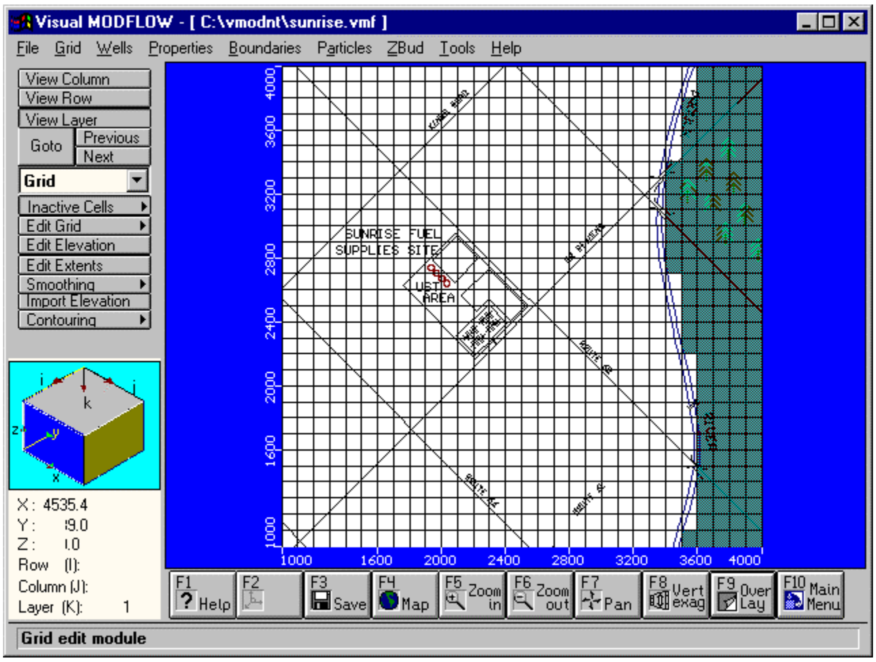
Once the model grid has been constructed, the next step is to begin assigning boundaries, model properties and boundary conditions.

The default input selection is always the Grid Input screen, which allows you to modify the model grid by adding or deleting rows, columns and layers. The Grid Input screen also allows you to designate grid cells as inactive, which means that groundwater flow in these cells does not contribute to groundwater flow in the model domain. In this example, we are assuming the Proulx River acts as a groundwater discharge zone, such that the groundwater flow directly underneath the river is vertically upwards and discharges to the river. In this case, we can assume that groundwater flow in the area east of the river does not contribute to groundwater flow in the area west of the river. Therefore, the model grid cells located east of the Proulx River can be set as inactive.

☞ **[Inactive Cells]** (from the left-hand menu bar)

☞ **[Mark Poly. Inactive]** (from the pop-up menu buttons)

This graphical tool will allow you to digitize a polygon around an area east of the Proulx River. Move the mouse pointer to the top right corner of the model grid and click the left mouse button once to anchor the polygon. Then move the mouse pointer down to the lower right-hand corner of the model grid and click again. Now move the mouse to the grid cell corresponding to the location just east of the Proulx River and click again. Continue this procedure until you have digitized a polygon around the area east of the Proulx River. To close the polygon, click the right mouse button in the top right-hand corner of the model grid (i.e. near the starting point of the polygon). A shaded polygon will appear as shown in the figure on the following page, indicating cells that have been designated inactive.



If the polygon you assigned has missed some cells that should also be inactive, you can assign single inactive cells as follows:

☞ **[Inactive Cells]** (from the left-hand menu)

☞ **[Mark Single]** (from the pop-up menu buttons)

Move the mouse to the cell that you want to assign inactive and click the left mouse button. To paint several cells inactive, click and hold the left mouse button while you drag the mouse across the cells. Alternatively, if you would like to re-activate cells that have been accidentally set as inactive, click the right mouse button and drag it across the de-activated cells.

Section 2: Input Model Properties and Boundary Conditions

The next step to building a groundwater flow model with Visual MODFLOW is to begin graphically assigning the model input parameters.

☞ **Properties** (from the top menu bar)

☞ **Conductivity** (from the drop-down menu)

☞ **[Yes]** (to save grid data before exiting)

A **Default Property Values** window will appear, prompting you to enter initial values for each of the soil properties required by the model. These values will be assigned to each grid cell in the model domain as initial default values. This will ensure that no grid cells are neglected when you are assigning subsequent model properties using Visual MODFLOW's graphical tools.

Enter the following default property values:

Kx (ft/d): **10** ⇔

Ky (ft/d): **10** (Ky is automatically set equal to Kx)

Kz (ft/d): **1.0** ⇔

Ss (1/ft): **0.005** ⇔

Sy: **0.20** ⇔

Eff. Por.: **0.20** ⇔

Tot. Por.: **0.23**

☞ **[OK]**

You will then be transferred to the Conductivity Input screen which allows you to graphically assign hydraulic conductivity values to model grid cells. This can be completed by painting individual cells, drawing polygons around multiple cells, or by stretching a window around multiple cells. In this particular example, a single layer, homogeneous model represents the aquifer. You therefore do not need to assign any heterogeneity to the system.

The next step is to begin assigning model boundary conditions.

☞ **Boundaries** (from the top menu bar)

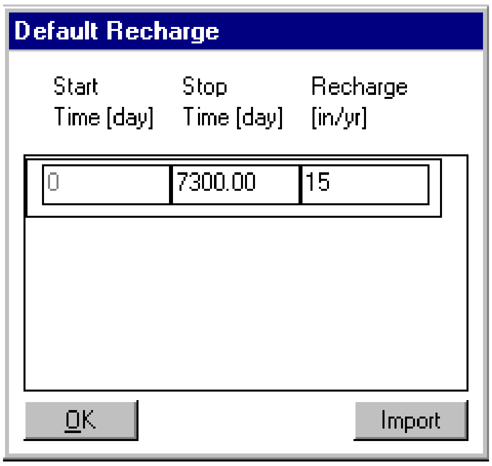
A drop-down menu will appear listing all of the available MODFLOW boundary conditions you can assign to your model.

☞ **Recharge** (from the drop-down menu)

A **Default Recharge** window will appear prompting you to enter an initial recharge value to be assigned to each grid cell in the top layer of the model (the only layer in this model). Enter the following default property values, as seen in the figure below:

Stop Time (day): **7300** ⇔

Recharge (in/yr): **15** (381 mm/yr.)



☞ **[OK]**

You will be transferred to the Recharge Input screen where you can graphically assign spatially and temporally variable recharge rates. In this example, you will assume a steady-state recharge rate of 15 in/year represents a good approximation of the annual average recharge to the aquifer.

To assign the constant head boundary condition along the western boundary,

☞ **Boundaries** (from the top menu bar)

☞ **Constant Head** (from the drop-down menu)

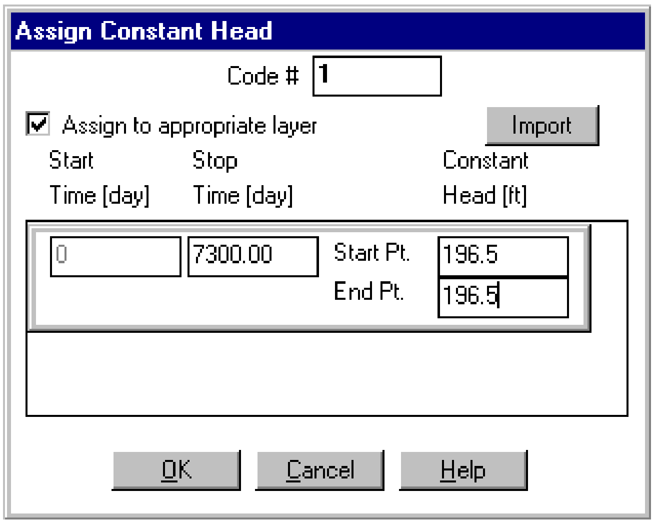
☞ **[Yes]** (to save property data before exiting)

You will then be transferred to the Constant Head Input screen that allows you to graphically assign constant head boundary conditions as single cells, lines, polygons or windows. To assign a constant head boundary along the entire western boundary of the model domain,

☞ **[Assign]** (from the left-hand menu bar)

☞ **[Line]**

Move the mouse to the top left-hand corner of the model grid (row 1, column 1, layer 1) and click once to anchor the line. Then move the mouse to the bottom left-hand corner of the model domain (Row 30, Column 1, Layer 1) and click the **RIGHT MOUSE BUTTON** to close the line. The cells corresponding to the line will be shaded pink, indicating they will be assigned a constant head boundary condition. An **Assign Constant Head** window will appear prompting you to enter the required constant head information, as seen in the figure below:



The **Code #** is used to group the selected cells for copying the specified constant head to the other layers. The ☑ **Assign to appropriate layer** option is used to assign the constant head boundary condition to the cell in the layer corresponding to the elevation of the specified constant head value. The **Start Time** is the time when you would like to begin applying the specified constant head, while the **Stop Time** is the time when you would like to stop applying the specified constant head. Finally, the **Start Pt.** is the constant head value specified at the beginning of the line, while the **End Pt.** is the constant head value specified at the end of the line.

Enter the following constant head information in the dialogue window:

Code #: **1**

☑ **Assign to appropriate layer**

Start Time (day): **0.000 (default)**

Stop Time (day): **7300** ⇔

Constant Head Start Pt. (ft): **196.5** ⇔

Constant Head End Pt. (ft): **196.5**

☞ **[OK]**

The pink line will turn red indicating that constant head values have been assigned to these cells.

Next, you will assign a line of river boundary conditions in the grid cells along the Proulx River to simulate the average water level in the vicinity of the river.

☞ **Boundaries** (from the top menu bar)

☞ **Rivers** (from the drop-down menu)

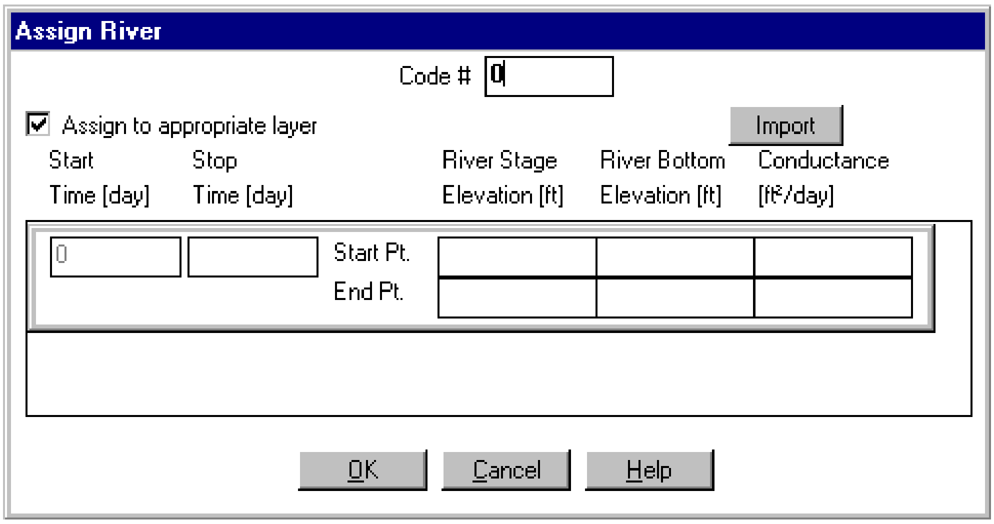
You will then be transferred to the River Boundary input screen.

☞ **[Assign]** (from the left-hand menu bar)

☞ **[Line]**

Move the mouse to the grid cell corresponding to northernmost portion of the Proulx River and click once to anchor the line. Then use the mouse to digitize a line along the Proulx River in the cells adjacent to the inactive zone.

Click the right mouse button to close the line at the southern most location of the Proulx River. The cells corresponding to the polyline will be shaded pink, indicating they will be assigned a river boundary condition. An **Assign River** window will then appear prompting you to enter the required information.



BEFORE you enter the required information, read the following definitions to gain a greater understanding of how MODFLOW defines river boundary parameters:

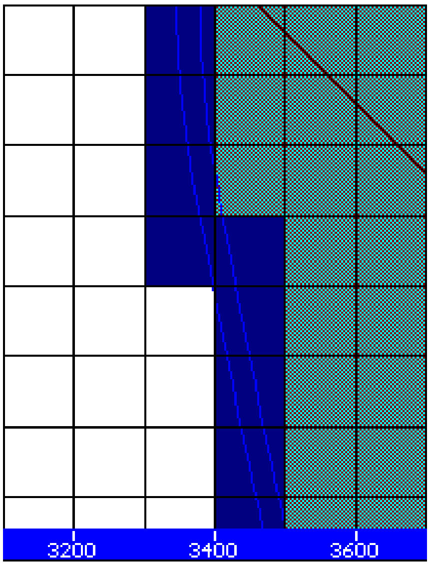
The **River Stage Elevation** describes the water surface elevation of the river.

The **River Bottom Elevation** describes the elevation of the bottom of the riverbed.

The **Conductance** is a MODFLOW specific term used to describe the ability of the riverbed to conduct flow from the river to the aquifer or vice versa. The equation to calculate Conductance for each cell is as follows:

$$C = K / B * (AREA)$$

Where, C is the Conductance
K is the vertical hydraulic conductivity of the riverbed
B is the thickness of the riverbed
AREA is the river length multiplied by the river width



Conductance values assigned to each model cell apply to the entire cell. For example, the figure above illustrates model river cells that are 100 x 100 feet in size. **However, in reality the river is only 50 feet wide, which is the value used in the calculation for the cell conductance value.**

For the purpose of this exercise, we will use average values to calculate the Conductance as follows:

$$C = \frac{(0.1\frac{ft}{day})(100\,ft)(50\,ft)}{(1\,ft)} = 500\frac{ft^2}{day}$$

Although conductance values can vary from cell to cell, we will assume the same value for all cells to simplify the calculations.

Enter the following values:

Code #: **2**

☑ **Assign to appropriate layer**

Start Time (day): **0.000 (default)**

Stop Time (day): **7300 ⇔**

Start Point:

River Stage Elevation (ft): **175 ⇔**

River Bottom Elevation (ft): **169 ⇔**

Conductance (ft^2/d): **500 ⇔**

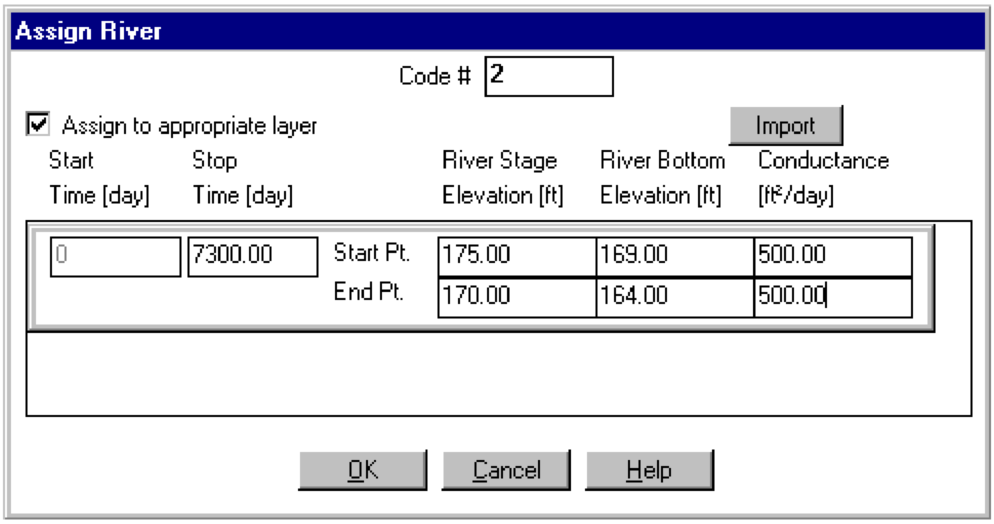
End Point:

River Stage Elevation (ft): **170 ⇔**

River Bottom Elevation (ft): **164 ⇔**

Conductance (ft^2/d): **500**

Your **Assign River** dialogue box should now appear as follows:



☞ **[OK]**

Section 3: Assigning Particle Tracking Locations

You have now completed all of the steps required to build a groundwater flow model. The finite-difference grid has been constructed, the model domain has been delineated, and the appropriate properties and boundary conditions have been assigned. This model could now be run and groundwater flow simulation results would be produced. However, before you run the model you will assign some forward tracking particles in the vicinity of the UST to determine the preferred migration pathways of the groundwater plume.

☞ **Particles** (from the top menu bar)

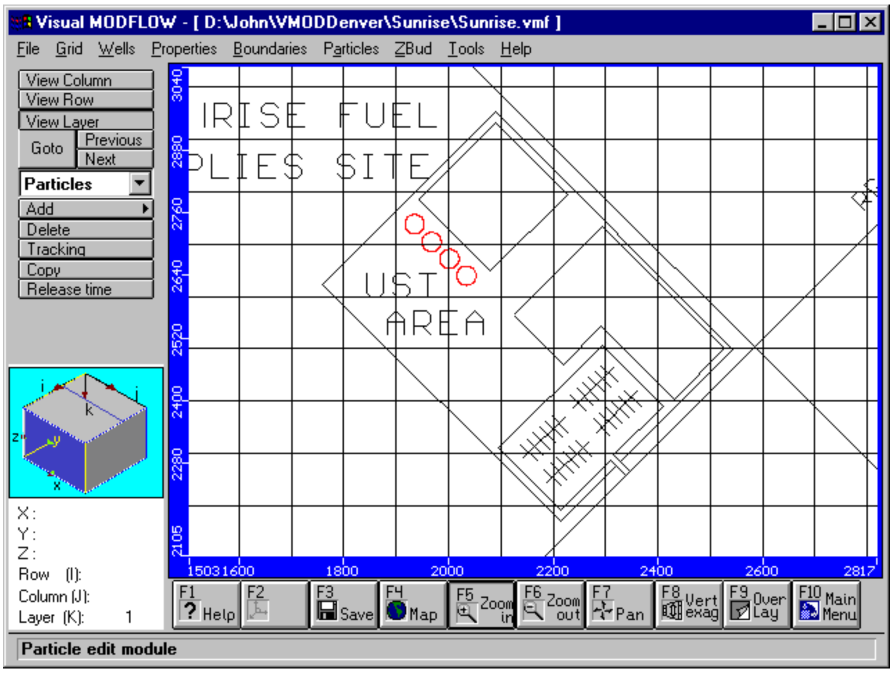
☞ **[Yes]** (to save the boundary data before exiting)

You will then be transferred to the Particles input screen (see the status bar along the bottom of the screen). Examine the buttons on the left-hand menu bar to see the options available for assigning particles within the model domain.

To begin assigning particles in the vicinity of the UST you should first zoom-in to the Sunrise Fuel Supplies Site area.

☞ **[F5 - Zoom-In]** (from the bottom menu bar)

Move the mouse cursor into the model domain and click on a location to the northwest of the site to anchor the zoom window. Then, stretch a box around the site and click again to close the zoom window. Your screen should appear similar to the figure below.

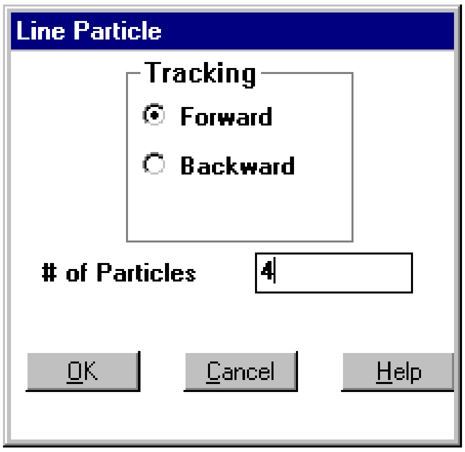


☞ **[Add]** (from the left-hand menu)

☞ **[Add Line]**

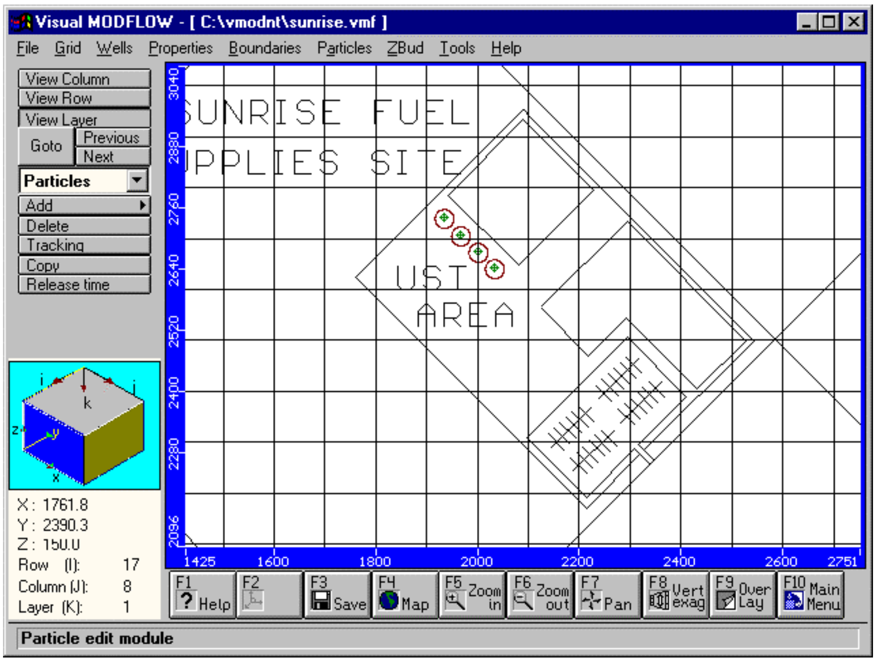
Move the mouse cursor to the area of the USTs and click once on the northern red circle to anchor the location of the particle line. Then stretch a line to the center of southern red circle and click again to close the line.

A Line Particle dialogue box will appear with default settings for the line of particles. Change the **# of Particles** from 10 to **4**.



☞ **[OK]** (to accept a line of 4 forward particles)

The particles should appear in the center of each UST as shown in the following figure.



You have just completed all the steps necessary to build a groundwater flow and pathline model.

Module 2: Running MODFLOW and MODPATH

You are now ready to run the computational simulation of this model.

☞ **[F10-Main Menu]**

☞ **[Yes]** (to save the particle information)

☞ **Run** (from the top menu bar)

You will then be transferred to the Run Options screen for Visual MODFLOW. This screen allows the user to customize some of the run-specific settings for running MODFLOW, MODPATH, MT3D/RT3D, and PEST.

A **Select Run Type** dialogue box will prompt you to specify whether you will be running either a Transient or Steady State simulation. The default setting is ⦿ **Steady State.**

☞ **[OK]** (to accept a steady-state simulation)

For this particular example, the remaining default run settings will be sufficient for running the model simulation that you have constructed. From the top menu bar:

☞ **Run**

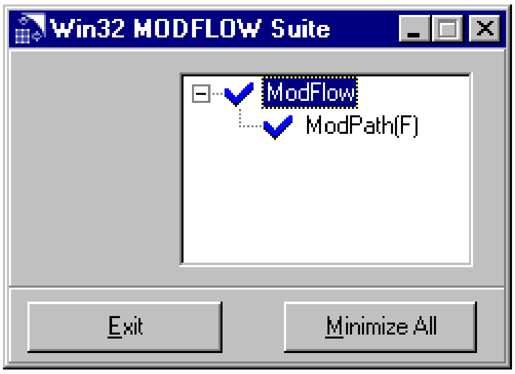
A **Translate/Run** dialogue box will appear,

☞ ☑ **MODFLOW** (so a ✓ appears in the box next to it)

☞ ☑ **Modpath** (so a ✓ appears in the box next to it)

☞ **[Translate & Run]**

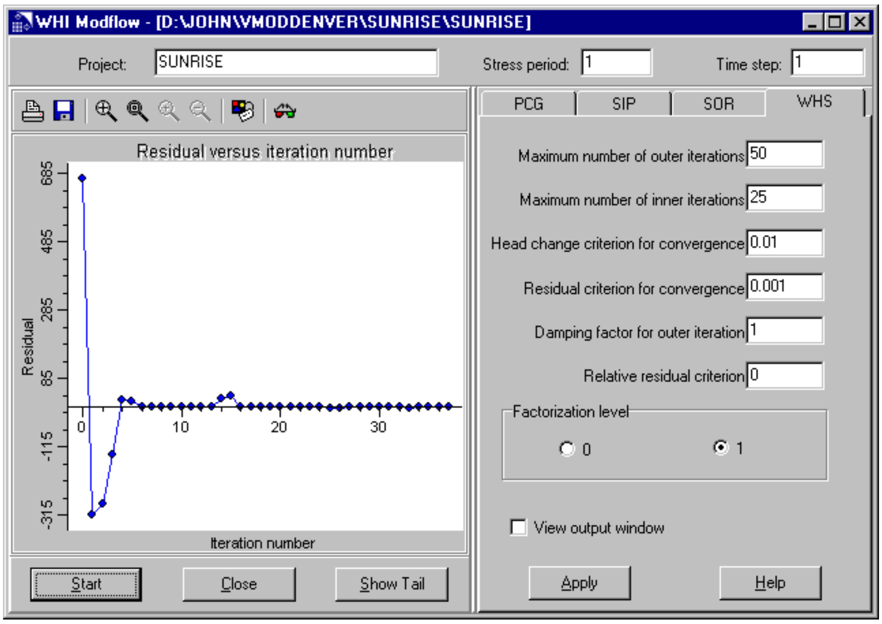
Visual MODFLOW will then create the necessary files and run the USGS MODFLOW program. Visual MODFLOW 2.8.2 contains MODFLOW, MODPATH, ZONE BUDGET, MT3D/RT3D, and PEST for Windows 95/NT applications. This unique modeling utility runs all of the available numeric engines and provides a graphical progress report for the MODFLOW solution convergence data and Zone Budget flow data. In addition, it allows on-the-fly modifications to the solver parameters while the solution is iterating.



When the Win^{32} MODFLOW Suite is executed from Visual MODFLOW, the Numeric Engine Control dialogue box will appear displaying the status of the models selected. A check mark indicates the numeric engine has completed running, a running horse indicates it is currently running, a red circle indicates it is waiting to be run, and a red cross indicated the engine has terminated and the model did not complete a solution.

Each engine will have an information window that displays simulation results and progress. These windows can be minimized by selecting **[Minimize All]** from the Numeric Engine Control dialogue box. Clicking on the specific model in the Numeric Engine Control dialogue box can open these dialogue boxes again.

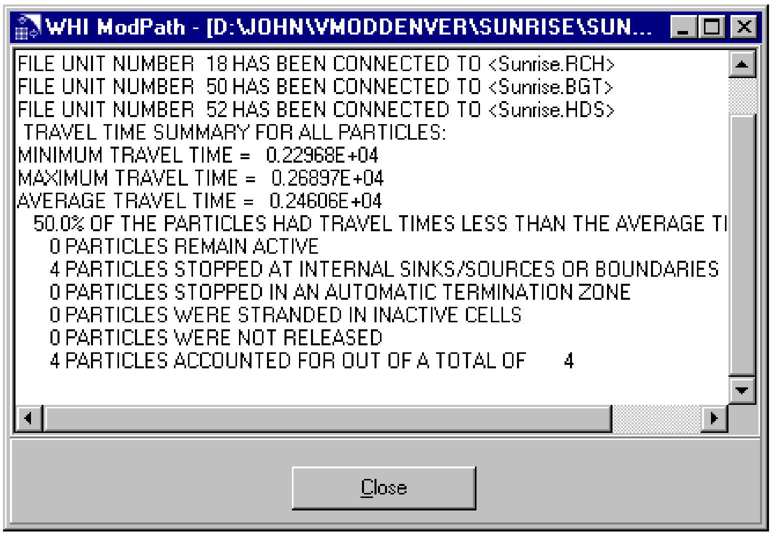
Clicking on the model in the Win^{32} MODFLOW Suite dialogue box can access the interactive MODFLOW screen below:



At the top of the screen is the project name of the model and the current stress period and time step MODFLOW is simulating. Beneath this information are the solver parameters and the graphical display of solution convergence data (maximum residual head vs. number of iterations).

Solution convergence data is graphically displayed on a plot of maximum residual head vs. number of iterations, which is updated after each iteration. Numerical output can also be displayed by selecting ☑ **View Output Window**.

The MODPATH dialogue box is shown in the following figure. This dialogue box displays the results and progress of the MODPATH calculations. It also provides a travel time summary for all particles and an explanation of where each particle became inactive or stopped in the simulation.



Once the MODFLOW and MODPATH calculations have been completed (as indicated by blue check marks),

☞ **[Exit]** (in the **Win^{32} MODFLOW Suite** dialogue box)

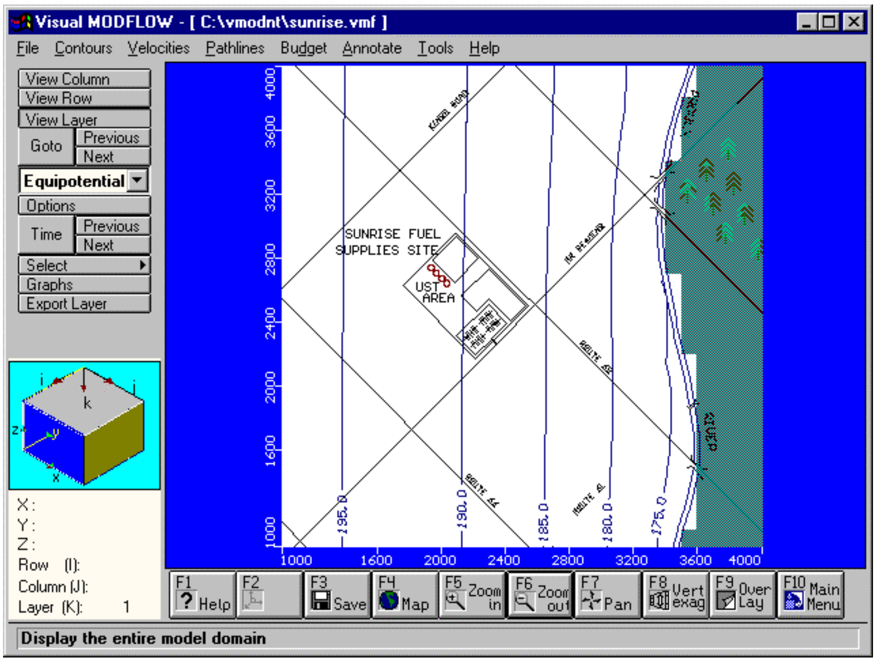
Module 3: Visualizing Model Results

Visual MODFLOW's powerful post-processing tools have been specifically designed for optimizing the display of groundwater flow and contaminant transport simulations. The post-processing of results includes steady-state or transient contouring of equipotentials, head differences between layers, head fluxes between layers, drawdown, water table elevation, and MT3D concentrations. The contouring options allow you to plot contour lines and/or color shading, select the contouring resolution/ speed, and customize the display of contour intervals and labels.

☞ **Output** (from the top menu bar)

You will be transferred to the Visual MODFLOW Output Menu which allows you to select and customize the display of results.

☞ **[F6 - Zoom-Out]** (from the bottom menu bar)



A plan view of the model domain will be displayed with equipotential contours displayed. These contours indicate a west to east groundwater flow direction towards the Proulx River. To see the preferred contaminant migration pathways from the UST Area,

☞ **Pathlines**

You will be transferred to the Pathlines Output screen and the steady-state pathlines will be displayed. Zoom in to the Sunrise Fuel Supplies Site area to examine these flow pathlines a little more closely.

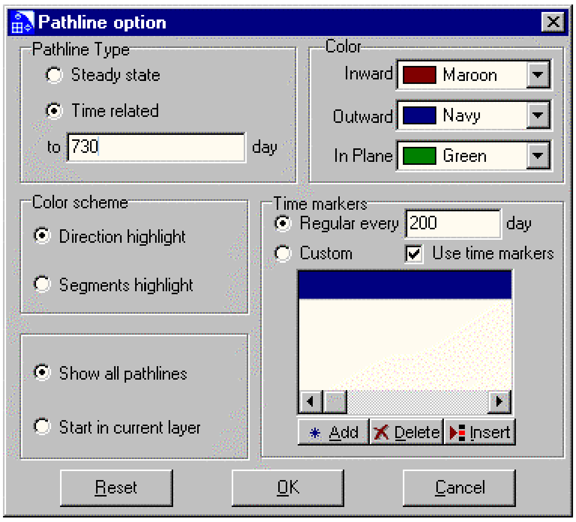
☞ **[F5 - Zoom In]** (from the bottom menu bar)

Using the mouse pointer, click once to anchor a corner of the zoom box and drag it to encompass the Fuel Supplies Site. Left-click your mouse again to close the box.

To estimate how far the groundwater plume may have migrated from the UST Area, a conservative approach would be to assume the groundwater plume travels at the same velocity as the groundwater flow. Therefore, the time markers on the flow pathlines will give an indication of the potential extent of contamination at the site.

☞ **[Options]** (from the left menu bar)

A **Pathline Option** dialogue box will appear showing the pathlines display options (see the following figure). The pathline type allows you to select either ⦿ **Steady state** or ⦿**Time related** pathlines. **Steady-state** pathlines will display flow pathlines for a steady state condition, while **Time-related** pathlines will display flow pathlines from time zero to a specific time. The time interval for the pathline time markers is displayed in the bottom right-hand section of the dialogue box. For this example, the default setting should say ⦿ **Regular every 200 days**.



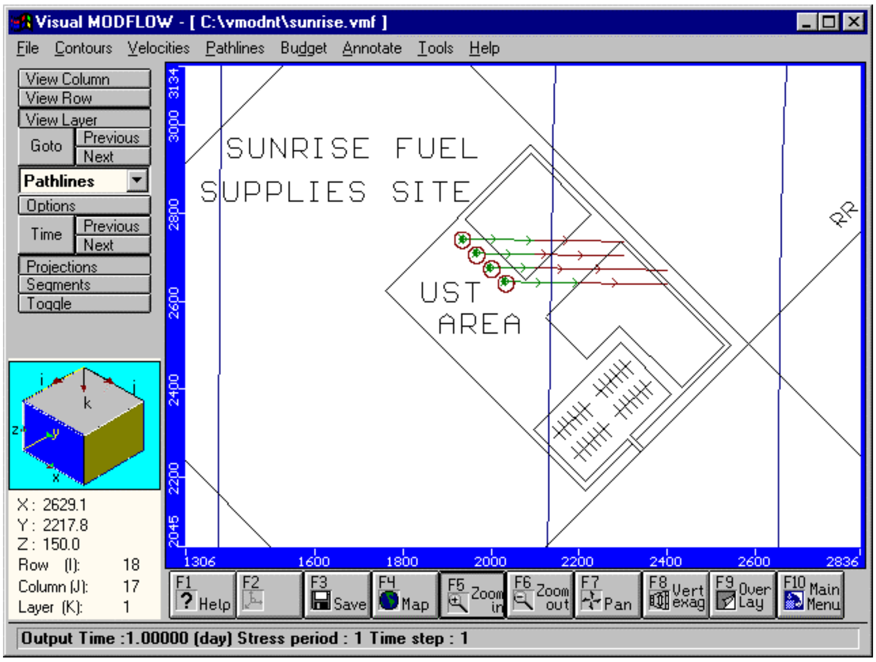
Change the settings to:

☞ ⦿ **Time Related**

type: **730** (in the box provided)

☞ **[OK]** (to accept these settings)

The new pathlines display should look similar to the figure below.



These pathlines show how far the conservative compounds in the groundwater plume have traveled after two years due strictly to advective transport mechanisms. These pathlines indicate the conservative elements of the contaminant plume have not likely migrated off-site. However, based on the pathline time markers contaminated groundwater plume will migrate off-site within the next 100 - 200 days.

Module 4: Simulating a Pumping Well

In this section of the exercise, you will simulate a pumping well to determine the pumping rate required to capture the existing plume and prevent any further off-site migration of the groundwater plume.

Section 1: Adding a Pumping Well

First, you must return to the Main Menu.

☞ **[F10-Main Menu]** (from the bottom menu bar)

☞ **Input** (from the top menu bar)

You will be transferred to the Grid Input screen by default.

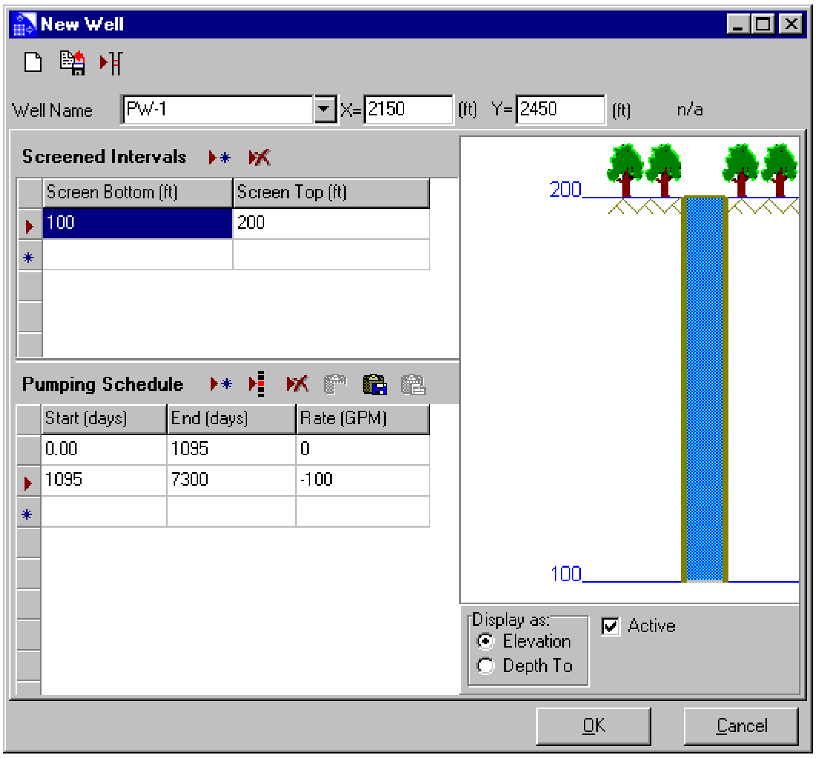
☞ **Wells** (from the top menu bar)

☞ **Pumping Wells**

You will then be transferred to the Wells input screen where you can graphically assign, edit, move, copy and delete pumping well locations and pumping schedules. To add a pumping well,

☞ **[Add Well]** (from the left-hand menu bar)

Using the grid co-ordinates in the bottom left-hand corner of the screen as a reference, move the mouse cursor to the grid location (row 16, column 12) and click the left mouse button. A **New Well** dialogue box will appear as shown below.



Enter the following information:

Well Name: **PW-1 ⇔**

X : **2150 ⇔**

Y : **2450**

Next you must enter the well screen interval. For this exercise you will screen the well across the entire depth of the aquifer.

☞ [▸ Ψ] (the **Add Screen** button beside the **Screened Intervals** text)

Move the mouse into the well bore, **RIGHT-CLICK** the mouse, and select **[Screen All]** from the drop-down menu.

Next you will enter the well pumping schedule. Since the USTs have been leaking for two years prior to the proposed installation of the pumping well, the well pumping schedule will consist of two time intervals. The first time interval will simulate the existing conditions at the site prior to the installation of the pumping well, while the second time interval will simulate the influence of the pumping well. When you are using MODFLOW, these time intervals are referred to as 'Stress Periods'.

It is estimated the design, approval and installation of the pump-and-treat remediation system will take a minimum of one year to complete. Therefore, the first time period for the simulation will be for the three years (1095 days) from when the UST leaks were first discovered to the time when the pump-and-treat system is installed. The second stress period will introduce pumping conditions at the well until a time of 7300 days (20 years). Enter the following pumping schedule for the remediation well (note the negative pumping rate for the extraction well).

Start (days): **0 (default)**

End (days): **1095 ⇔**

Rate (GPM): **0 ⇔**

Start (days): **1095 (default)**

End (days): **7300 ⇔**

Rate (GPM): **-100**

☞ **[OK]** (to accept the pumping well information)

☞ **[F6 – Zoom out]** (to view the entire model domain)

A red well symbol will appear and the grid cell will be shaded red indicating the presence of an active pumping well.

Section 2: Running the Modified Model

Now run the new simulation with the proposed pumping well operating.

☞ **[F10 - Main Menu]** (from the bottom menu bar)

☞ **Yes** (to save well data)

☞ **Run** (from the top menu bar)

You will be transferred to the Run Module and you will be prompted to select the run type. Since you now have two stress periods for the pumping well, you will need to run a transient simulation to account for the different system conditions.

☞ **⊙ Transient**

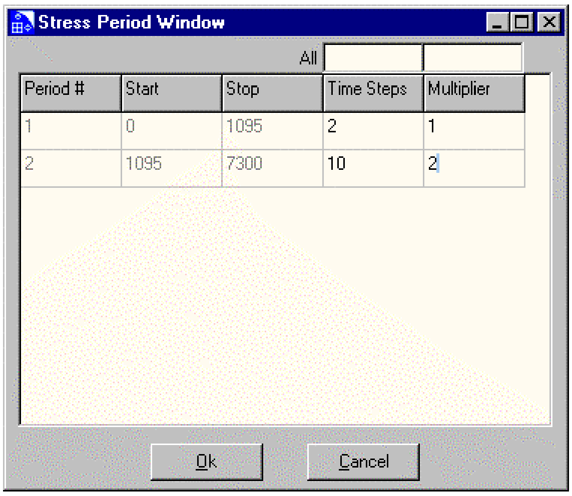
☞ **[OK]**

☞ **MODFLOW**

☞ **Time Steps**

The **Time Steps** option is active only for transient simulations and it allows you to customize the number of time steps for each time period, and to specify a multiplier for the time steps increment.

A Stress Period Window dialogue box will appear showing the available time settings and **default** number of time steps (10) and time step multiplier (1.2) for both time periods. MODFLOW will calculate the heads and drawdown for each of the time steps in each stress period and MODPATH uses these head values to determine the transient particle tracking pathlines.



Using the above figure as a reference, enter the required information in the **Stress Period Window** dialogue box.

Since the first stress period is essentially at steady state with regards to the existing conditions (initial heads) at the site, it is not necessary to calculate the results with very many time steps. However, when the well is turned on after three years, the flow field will change rapidly near the beginning of the stress period (rapid water table drawdown). Therefore, a time step multiplier >1 has been set to provide more information earlier in the stress period when the most rapid changes are occurring.

☞ **[OK]**

This will calculate the heads and drawdown for two time intervals in the first stress period (0 to 1095 days), and 10 time intervals in the second stress period (1095 to 7300 days).

For this run, you will be interested in seeing the influence of the pumping well on drawdown levels and particle migration from the UST Area. However, in order to calculate the drawdown, the model needs to know what the initial conditions were prior to pumping. Therefore, you will import the initial head estimate for this simulation from the previous Visual MODFLOW simulation.

☞ **MODFLOW** (from the top menu bar)

☞ **Initial Heads** (from the drop-down menu)

☞ **⊙ Previous Visual MODFLOW Run**

☞ **[OK]**

An **Import head from MODFLOW run** dialogue box will appear prompting you to select the appropriate file.

☞ **sunrise.HDS**

☞ **[Open]**

☞ **[OK]** (to accept default values)

The final step, prior to running the model, will be to set the output control options to calculate drawdown for each time step.

☞ **MODFLOW** (from the top menu bar)

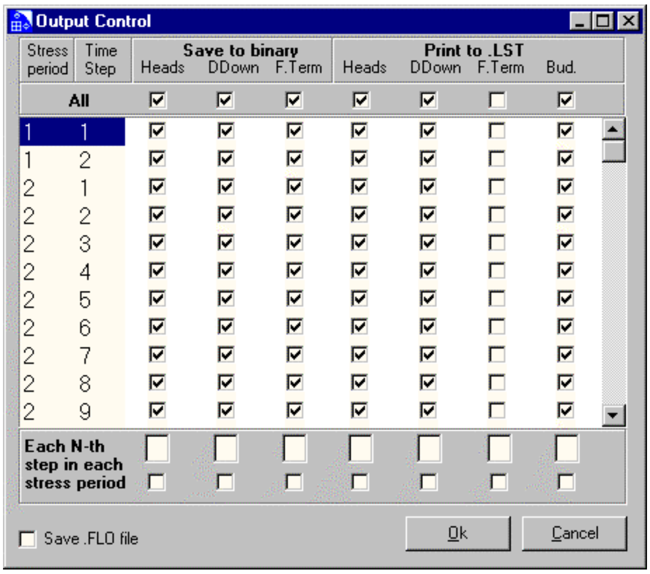
☞ **Output Control** (from the drop-down menu)

An **Output Control** dialogue box will appear listing the available output information to calculate and print to the listing (**.LST**) file, and the time steps at which they can be activated. The default settings indicate that heads will only be calculated at the end of each stress period (i.e. at 1095 and 7300 days). However, for transient simulations, MODPATH requires the heads to be calculated at each time step. In addition, you may also want to observe the transient development of the drawdown cone of depression around the well for each time step.

☞ **All** ☑ ☑ ☑ (in the 'Save to Binary' column)

☞ **All** ☑ ☑ ☐ ☑ (**Heads**, **DDown**, and **Bud.** in the 'Print to .LST' column)

Your Output Control dialogue box should look the same as the following window.



☞ **[OK]**

This will calculate heads and drawdown data at each time step for both stress periods. Now you are ready to run the model.

☞ **Run** (MODFLOW and MODPATH should still be selected)

☞ **[Translate & Run]**

Visual MODFLOW will then begin translating the Visual MODFLOW files and will open up the Win32 MODFLOW Suite to run the MODFLOW and MODPATH simulation. When the

MODFLOW and MODPATH calculations are complete (as indicated by the blue checkmarks) select **[Exit]** to close the Win32 MODFLOW Suite.

Section 3: Visualizing the Effects of a Pumping Well

When the simulation is complete, Visual MODFLOW will return to the Main Menu.

☞ **Output** (from the top menu bar)

☞ **[F6 – Zoom Out]** (from the bottom button bar)

Visual MODFLOW will display a plot of the head contours for the first time step of the first stress period (time = 547.5 days). When you move the mouse cursor into the model domain, the simulation time is displayed on the status bar along the bottom of the screen. Notice that the heads at 547.5 days are the same as the initial simulation. This result is for the first stress period where the well has not yet started pumping (the initial steady-state conditions for the water table at the site).

In the latter part of this exercise, you will need an ASCII (x, y, h) file containing the head values for the non-pumping condition at this site. The purpose for the ASCII file will be explained later.

☞ **[Export Layer]** (from the left-hand menu bar)

☞ **[Active Only]**

An **Export Head Layer - ASCII** dialogue box appears requesting you to enter a Filename for the ASCII (x, y, h) file. Enter the following:

Filename: sun_ini.asc

☞ **[Save]**

An ASCII (x, y, h) file named **sun_ini.asc** will be created in the C:\VMODFLOW directory.
To display a listing of all available time steps,

☞ **[Time]** (from the left-hand menu bar)

A **Select Output Time** dialogue box will appear listing the available output time steps. Notice the time steps for the first stress period (0 to 1095 days) are divided into two equal time intervals (547.5 days), while the time steps for the second stress period (1095 to 7300 days) are more frequent in the early stages of the stress period.

☞ **1095**

☞ **[OK]**

This will display the heads for a time of 1095 days, just prior to the pumping well being activated.
To display the head contours for the first time step after pumping,

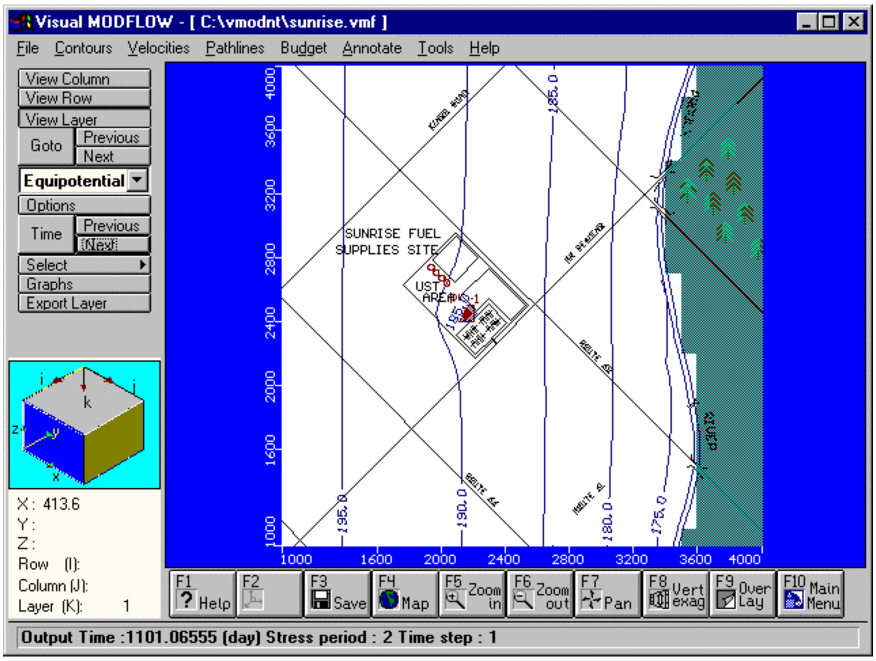
☞ **[Next]** (beside the Time button)

Notice the small deformation of the head contours in the vicinity of the pumping well as shown in the figure on the following page. Continue to step through the remaining time steps by selecting

[Next] until the heads reach a steady state condition (i.e. the heads no longer change significantly).

Note: *To see the time after clicking the **[Next]** button, move the mouse into the model domain to read the Output time, Stress Period and Time Step from the lower display bar.*

A steady-state condition appears to be achieved after approximately 1865 days (770 days after turning the pumping well on). Therefore, it will take more than two years of pumping at 100 US GPM before the aquifer approaches a steady-state drawdown condition.



Now return to a heads output display time of 1101 days.

☞ **[Time]** (from the left-hand menu)

☞ **1101**

☞ **[OK]**

The head contours should appear as shown above. To display the drawdown contours,

☞ **Contours** (from the top menu bar)

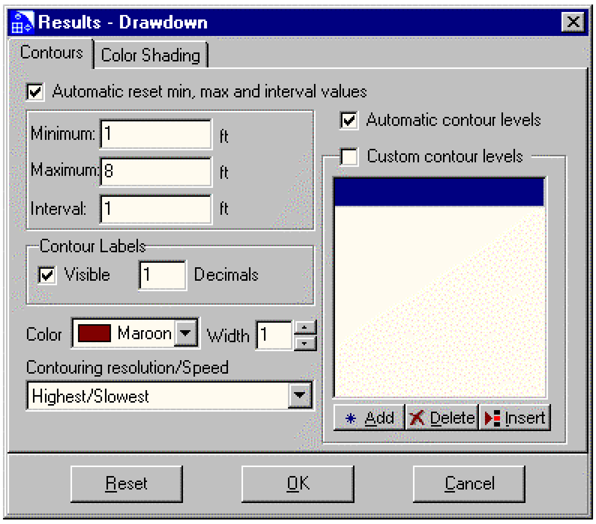
☞ **Drawdown** (from the drop-down menu)

You will be transferred to the Drawdown Output screen where the head contours are replaced by drawdown contours. Notice there is zero drawdown closest to the edges of the model domain where the heads are fixed due to boundary conditions. To 'clean-up' the display of drawdown contours, you should set the range of drawdown values to just above zero.

☞ **[Options]** (from the left-hand menu)

Before altering values in the **Results – Drawdown** dialogue window that appears, take a minute to familiarize yourself with the functionality of the following features:

1. The ☑ **Automatic reset min, max and interval values** defaults to active indicating every time you advance to a new layer, the minimum, maximum and interval of the head contours are re-calculated.
2. The ☑ **Automatic contour levels** defaults to active which means that the contours displayed on screen will be set according to the values indicated in the boxes labeled Minimum, Maximum, and Interval.
3. The ☐ **Custom contour levels** defaults to inactive indicating that custom contours levels will not be displayed.
4. The **Contour Labels** box allows you to set the number of decimal places for each contour value.
5. The **Contouring Resolution/Speed** option allows you to select the desired resolution of the contours and the corresponding speed at which the contours are calculated. This is a particularly useful option when you are modeling very large grids (200 x 200 cells). Visual MODFLOW defaults to the highest resolution of contouring, which is set to **Highest/Slowest**.
6. The **Reset** button allows you to manually reset the minimum, maximum and interval value of the contours displayed on the present screen. This button is only applicable when the Automatic reset minimum, maximum and interval values option is de-activated.



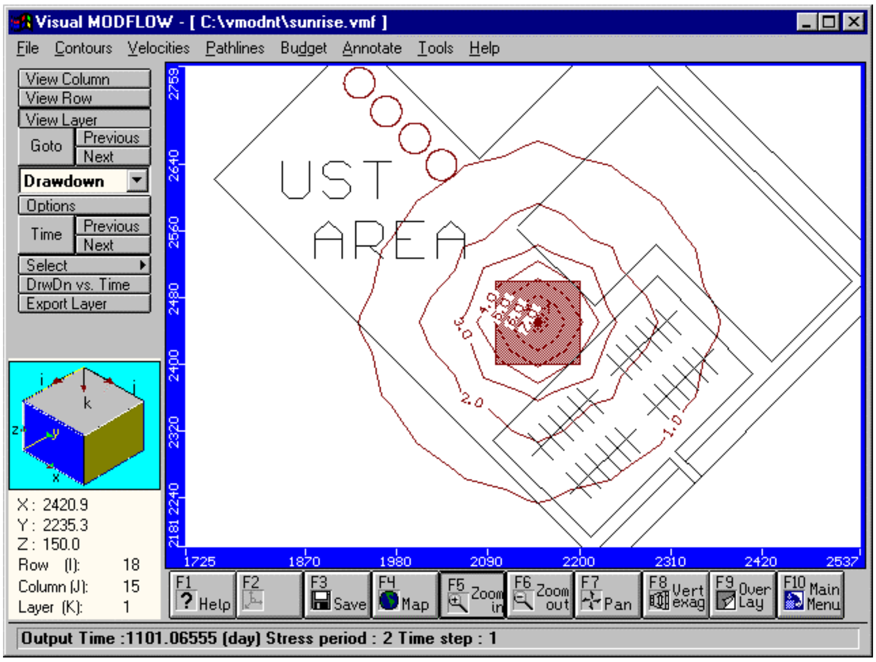
Now, change the **Minimum Contour** value to **1.0**. By default, the **Maximum Contour** value should be **8.0** and the **Interval** value should be **1.0**.

The drawdown contours will be plotted for a range of 1.0 - 8.0 ft (0.3 to 2.4 m) with a one-foot interval.

☞ **[OK]**

Zoom in to the site area to examine the gradual development of the drawdown cone of depression as it extends radially outwards from the well.

☞ **[F5 - Zoom In]**



To examine the drawdown contours for the next time step,

☞ **[Next]** (from the side menu bar)

Remember the range of contour values and interval between contours is being recalculated at each time step. Therefore, although the cone of depression appears the same for the next time step, the range of contour values has increased and the contour interval is now 2.0.

Continue to advance through each time step to watch the cone of depression spread out radially from the well. It is interesting to note the drawdown contours near the pumping well are not as smooth as the contours further away from the well. This is due to the coarse grid spacing of the model near the pumping well. This effect will be examined in more detail later on.

☞ **[F6 - Zoom Out]**

☞ **[F9 – Overlay]**

☞☞ **Results - Drawdown** (removes * from selection)

☞ **[OK]**

The next step is to evaluate the effectiveness of the pumping well to see if it will capture the groundwater contamination plume and prevent further off-site migration of contaminants.

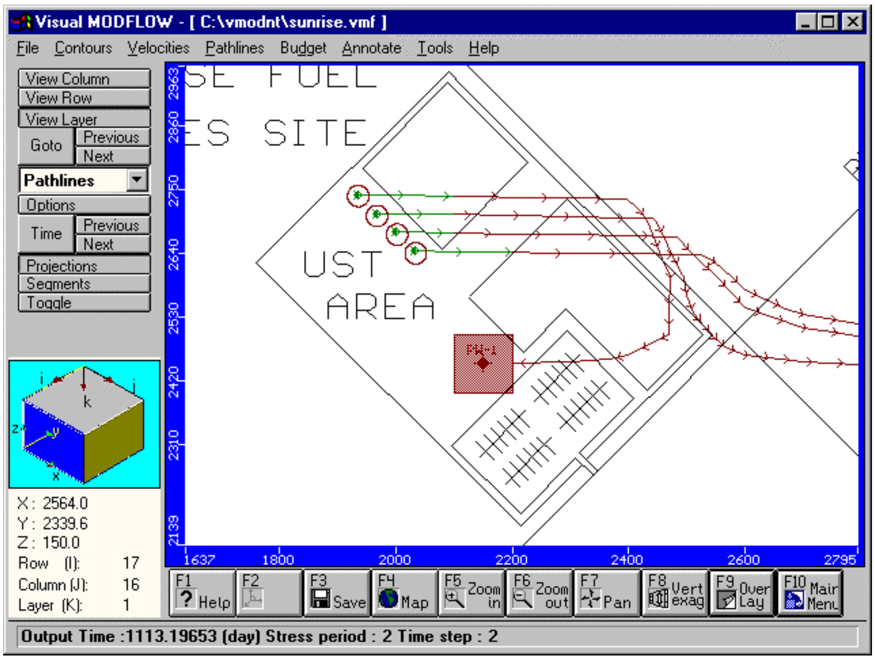
☞ **[Pathlines]** (from the top menu bar)

☞ **[Options]** (from the side menu bar)

☞ **⊙ Steady State**

☞ **[OK]**

Using the **[F5 – Zoom in]** button from the bottom toolbar, select an area similar to the figure shown below.



Notice the pathlines represent flow directions for the entire simulation time from time zero to 7300 days. Therefore, unlike heads and drawdown, the pathlines display will not change with each time step.

These results indicate the well pumping rate is not high enough to capture the entire contaminated groundwater plume migrating from the fuel storage tanks. Therefore, the pumping rate must be increased or the well must be moved.

To view the site in cross-section,

☞ **[View Row]** (from the left-hand menu bar)

Move the mouse into the model domain and a horizontal red bar will highlight each grid row as you move the mouse up and down. Select a cross-section profile along the row in which the pumping well is located by clicking the left mouse button on **Row 16**. A relatively flat model layer will appear on the screen. To enlarge the vertical perspective of the cross-section you must assign a vertical exaggeration to the model.

☞ **[F8 - Vert Exag]** (from the bottom menu bar)

type: **20**

☞ **[OK]**

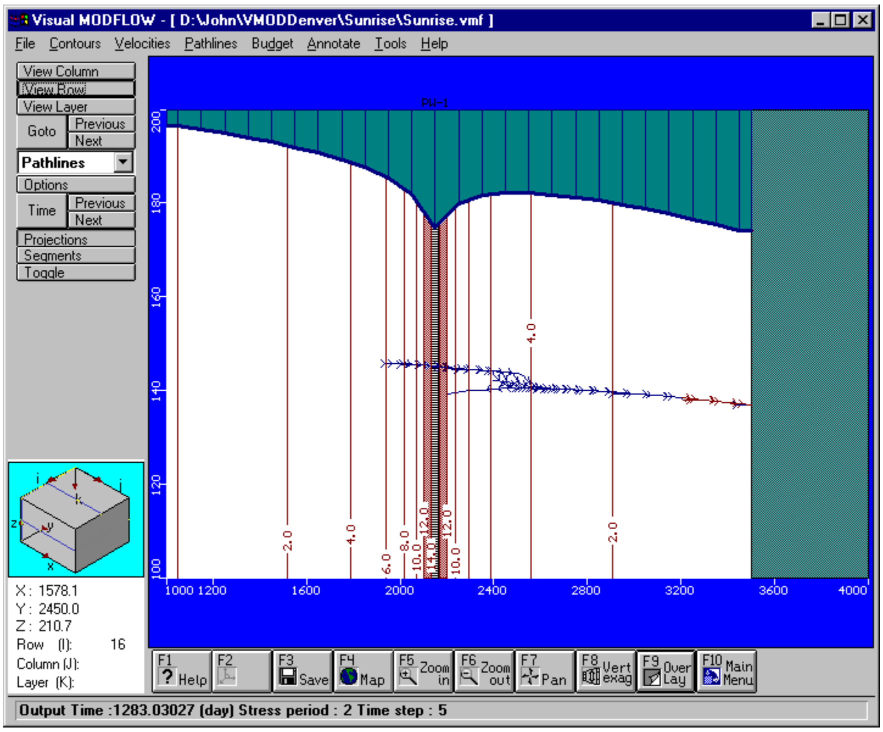
As well, you need to add the Drawdown Overlay to the display. To accomplish this task,

☞ **[F9 - Overlay]** (from the bottom menu bar)

☞☞ **Results - Drawdown** (adds an * beside the selection)

☞ **[OK]**

The cross-section on your screen should look similar to the figure below.



Now return to the plan view display of the model domain.

☞ **[View Layer]** (from the left menu)

When you move the mouse into the cross-section, the entire model layer will be highlighted. Click the left mouse button to return to the plan view display of the model.

When you are using a groundwater model to study the groundwater flow for a site, it is often necessary to report the calculated or predicted heads at a precise location or for a particular grid cell. This can be accomplished using the Cell Inspector.

☞ **Tools** (from the top menu bar)

☞ **Cell Inspector**

A **Cell Inspector** dialogue box will appear. On the **Options** tab, click **[All On]**

☞ **Cell Values** (available tab selection)

A **Cell Inspector** dialogue box will appear listing the available output results that can be inspected on a cell-by-cell basis. Re-size the dialogue box by moving the mouse to the edge of the box (changes to an arrow) and dragging the window to the appropriate size. This allows you to view all of the available results without having to use the window scroll bar.

Move the mouse pointer into the model domain and the cell-by-cell information will be displayed in the Cell Inspector dialogue box. Move the cursor to the cell containing the pumping well. The head at the pumping well location should be approximately 177 ft.

☞ **[X]** (top right-hand corner of Cell Inspector to close application)

☞ **[F10 - Main Menu]** (to return to the Main Menu)

Module 5: Refining the Model Grid

Next, we will examine the effects of refining the model grid in the vicinity of the pumping well.

Section 1: Refining the Grid

☞ **[Input]**

☞ **[F9 - Overlay]**

☞☞ **[HBC – Pumping Wells]** (* indicates the overlay has been activated)

☞ **[OK]**

Zoom in to the site area around the pumping well.

☞ **[F5 - Zoom In]**

☞ **[Edit Grid]** (from the left-hand menu)

☞ **[Edit Rows]**

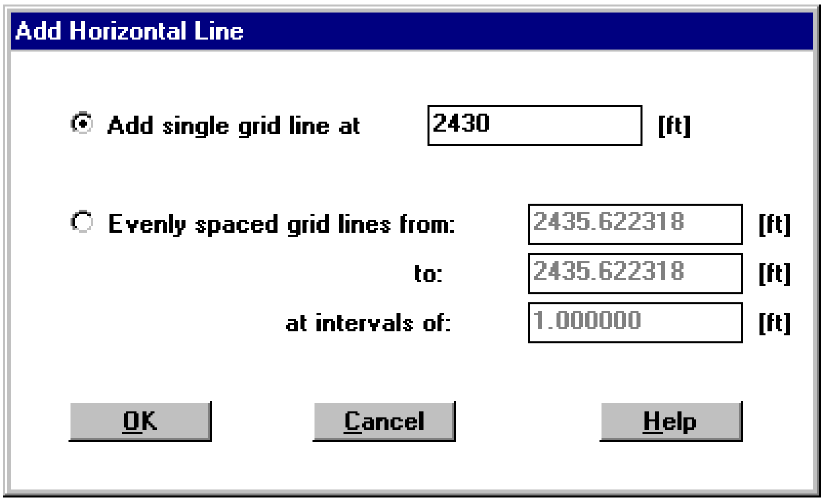
☞ ⊙ **Refine by 2** (in the **Rows** dialogue box)

Move the mouse into the model domain. The Y co-ordinate of the cursor will be displayed in the left-hand corner of the screen below the navigator cube. Click the mouse on the row corresponding to a Y-location of approximately 2200 ft, and click again at a Y-location of approximately 2700 ft. This will double the number of gridlines between these two locations.

Now you will add a few more grid lines closer to the pumping well.

☞ ⊙ **Add** (in the **Rows** dialogue box)

RIGHT CLICK the mouse anywhere in the model domain, producing an **Add Horizontal Line** dialogue box in the display window similar to the figure below.



☞ ⊙ **Add single grid line at**

type: **2430**

☞ **[OK]**

REPEAT this procedure for three more grid lines at Y-locations of 2445, 2455 and 2470 ft.

Finally, delete the gridline, which passes directly through the pumping well.

☞ ⊙ **Delete** (click on the row at Y=2450)

☞ **[Close]** (to accept these grid modifications)

Now do the same thing for the columns.

☞ **[Edit Grid]** (from the left-hand menu)

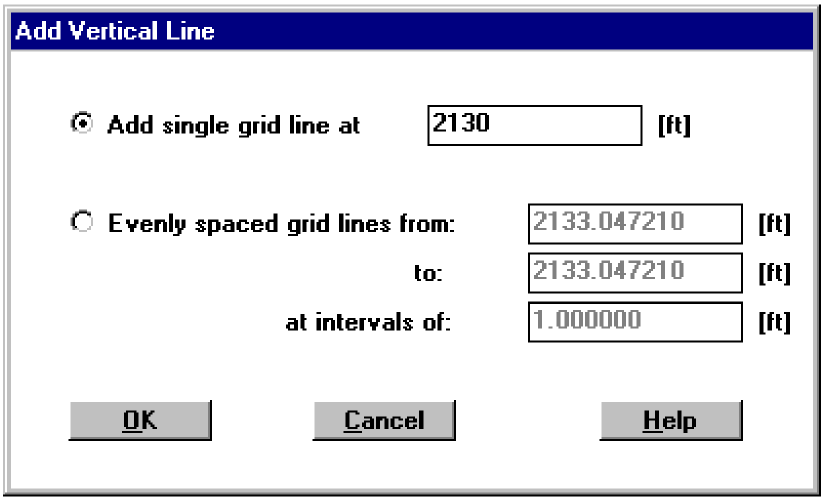
☞ **[Edit Columns]**

☞ ⊙ **Refine by 2**

Move the mouse into the model domain and highlight the column X = 1900 and click the left mouse button. Then move the mouse to highlight the column X = 2400 and click the left mouse button again to refine the grid between these two columns. Now add a few more gridlines closer to the pumping well.

☞ ⊙ **Add** (to refine the grid in the well cell)

RIGHT CLICK anywhere in the model domain and an **Add Vertical Line** dialogue box similar to the figure below will appear.



☞ ⊙ **Add single grid line at**

type: 2130

☞ **[OK]**

REPEAT this for three more grid lines at X-locations of 2145, 2155 and 2170 ft.

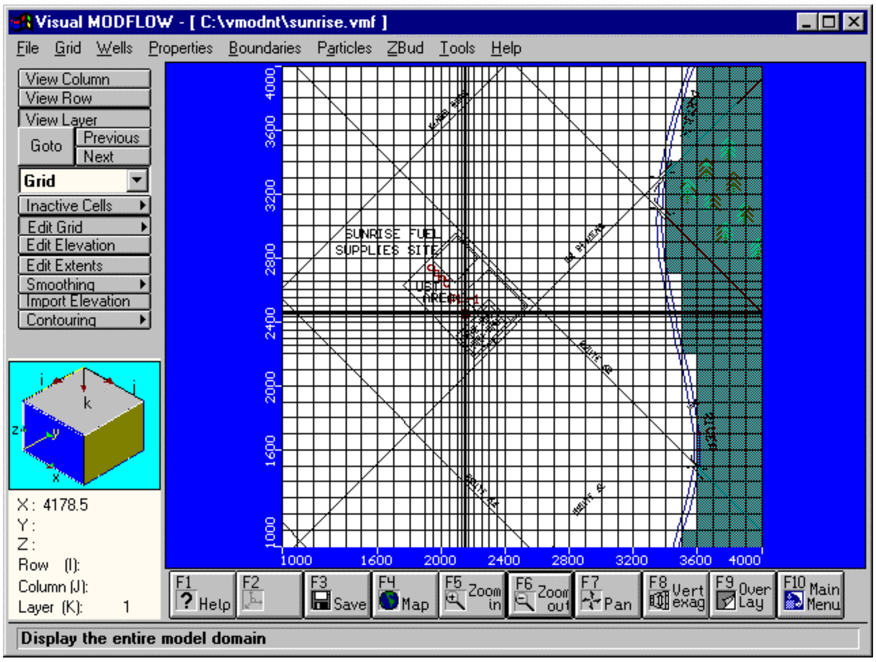
Finally, delete the gridline, which passes directly through the pumping well.

☞ ⊙ **Delete** (click on the column at X=2150)

☞ **[Close]** (to accept these grid modifications)

☞ **[F6 - Zoom Out]**

The refined model grid should appear similar to the following figure.



Now run the model again to see how these grid refinements will alter the results.

☞ **[F10 - Main Menu]** (to return to the Main Menu)

☞ **[Yes]** (to save the grid data)

Section 2: Running the Refined Grid Model

☞ **Run** (from the top menu bar)

☞ **[OK]** (to accept a transient simulation)

Since you have refined the model grid, you can no longer use the initial head estimates from a previous Visual MODFLOW run because the **.HDS** file will not correlate to the new grid dimensions. Therefore, the initial head estimate must be obtained from a different source. This is why you created the '**sun_ini.asc**' file earlier. It contains the simulated head values for the model prior to pumping.

☞ **MODFLOW**

☞ **Initial Heads**

☞ **⊙ Import from ASCII File**

☞ **[OK]**

An **Import heads from ASCII file** dialogue box will appear listing the available ASCII (**.ASC**) files in the C:\VMODFLOW directory.

☞☞ **sun_ini.asc**

The next dialogue box appears listing the number of nearest data points that it will use to interpolate the data to the model grid. Enter the following:

 # Nearest: **1**

☞ **[OK]**

☞ **[F3 – Save]**

Now run the model simulation for the refined model grid.

☞ **Run**

☞ **[Translate & Run]** (to run MODFLOW and MODPATH)

Once this is completed, press **[Exit]** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

Section 3: Visualizing the Model Output

From the Main Menu,

☞ **Output** (from the top menu bar)

Visual MODFLOW will display a plot of the head contours for the first time step of the first stress period (Time = 547.5 days). Notice the head contours do not look noticeably different than the previous simulation with the coarse grid.

Zoom in to the area around the pumping well to see whether the grid refinement had a significant impact on the heads and flow pathlines,

☞ **[F5 - Zoom In]** (from the bottom button bar)

Select an output time step at the end of the simulation time (e.g. ~7300 days).

☞ **[Time]** (from the left-hand menu)

☞ **7300**

☞ **[OK]**

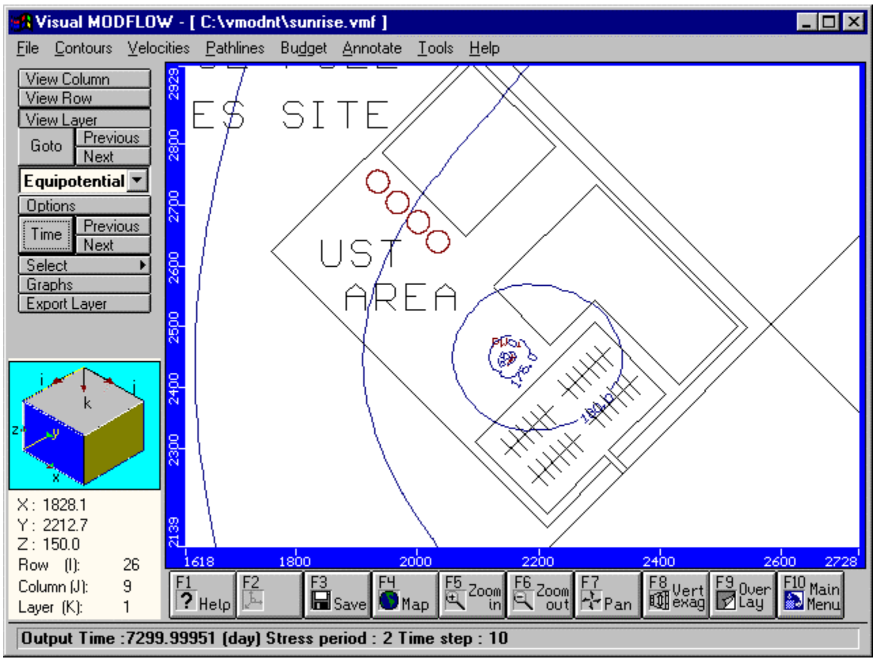
The displayed heads should look similar to those in the following figure, as the contours closer to the well are much smoother and the minimum contour value at the well is now 165 ft. In the previous simulation, however, the minimum contour at the well was much higher at 175 ft.

Use the cell inspector to determine the exact calculated head in the cell where the pumping well is located.

☞ **Tools** (from the top menu bar)

☞ **Cell Inspector**

☞ **Cell Values** (available tab selection)



Move the mouse cursor to the location of the well (row 20, column 16) and check the Cell Inspector dialogue box for the calculated head in the cell. The calculated head in the cell should be approximately 161.3 ft (this may vary slightly depending on the grid spacing).

Compare this head value to the head value calculated in the first simulation before the grid refinements (174.7 ft). Clearly, grid refinements have a significant impact on the heads calculated by the model, particularly in areas where there are steep gradients. The larger the cell size, the more MODFLOW over-predicts the true head level. Therefore, refining your grid in critical modeling areas is very important.

This is further illustrated by viewing a cross-section through the well. Close the Cell Inspector dialogue box (click the **X** in the top right corner).

☞ **[View Row]**

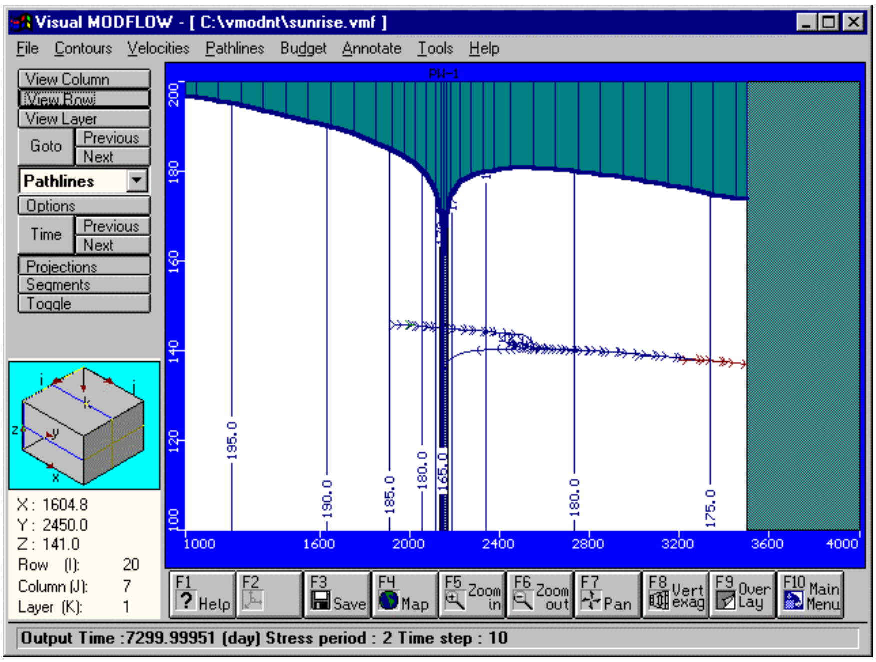
Move the mouse pointer into the model domain and click on **Row 20** to view a cross-section profile passing through the pumping well location. The cross-section should be similar to that shown below.

his cross-sectional view of the water table drawdown at the well clearly demonstrates the impact of grid refinement on the water levels of the aquifer.

Now return to the plan view display of the model domain.

☞ **[View Layer]**

Note: These modeling results can be displayed and animated in three-dimensional view using the Visual MODFLOW 3D-Explorer by clicking the [F2] button. Although there currently no instructions for using the Visual MODFLOW 3D-Explorer, the User's Guide is provided in the Documentation directory of your Visual MODFLOW working directory.



Section 4: Increasing the Pumping Rate

Next, you will increase the pumping rate to 125 US GPM (7.88 L/s) to capture and contain the off-site migration of the groundwater plume.

☞	**[F10 - Main Menu]**	(from the bottom menu bar)
☞	**Input**	(from the Main Menu bar)
☞	**Wells**	(from the top menu bar)
☞	**Pumping Wells**	

You will be transferred into the Well Input screen where you can add, delete, copy and move well locations within the model domain.

☞	**[Edit Well]**	(from the left-hand menu bar)

Place the mouse cursor directly over the well symbol and click the left mouse button to edit the pumping well information. A **Well Edit** dialogue box will appear displaying the existing well data.

Change the pumping rate in the well from **-100** US GPM (6.2 L/s) to **-125** US GPM (7.88 L/s).

☞	**[OK]**	(to accept the changes to the well pumping schedule)
☞	**[F10 - Main Menu]**	
☞	**[Yes]**	(to save the well data before exiting)
☞	**Run**	(from the Main Menu)
☞	**[OK]**	(to accept a Transient simulation)
☞	**Run**	

☞ **[Translate & Run]** (to run MODFLOW and MODPATH)

Visual MODFLOW will then begin translating the Visual MODFLOW files and will activate the Win 32 MODFLOW Suite. Again, click on **[Exit]** when both MODFLOW and MODPATH have finished calculating.

When the simulation is complete, Visual MODFLOW will return to the Main Menu.

☞ **Output** (from the top menu bar)

Visual MODFLOW will display a plot of the head contours for the first time step of the first stress period (time = 547.5 days).

If you are not already zoomed in, zoom in to the site area to examine the particle pathlines.

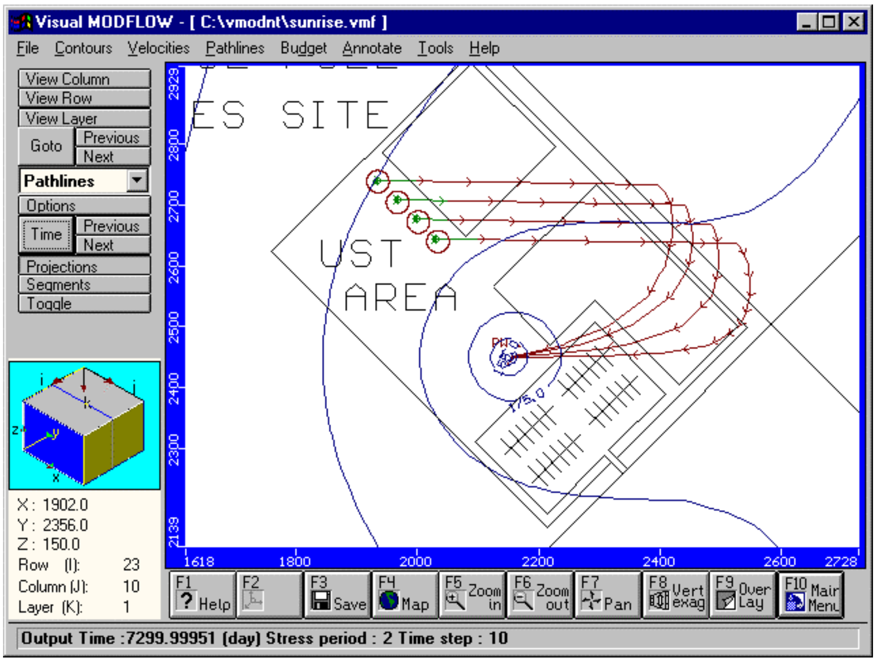
☞ **Pathlines** (from the top menu bar)

☞ **[Time]**

☞ **7300** (to see the results at the end of 20 years)

☞ **[OK]**

The results should indicate the increased pumping rate of 125 US GPM (7.88 L/s) successfully captures the particle flow pathlines migrating off-site (see the following figure).



This concludes the model construction phase of the Sunrise laboratory. If you have time left, try examining the drawdown contours as you did previously in this exercise, or look at the flow velocity vectors by selecting **[Velocities]** from the top menu bar.

☞ **[F10 - Main Menu]** (to return to the Main Menu)

Module 6: Calibrate to Steady-State, Non-Pumping Conditions

Section 1: Adding Observation Wells

To start this exercise, you have to enter the input screen and then you can begin entering the observation well data into Visual MODFLOW.

☞ **Input** (from the Main Menu)

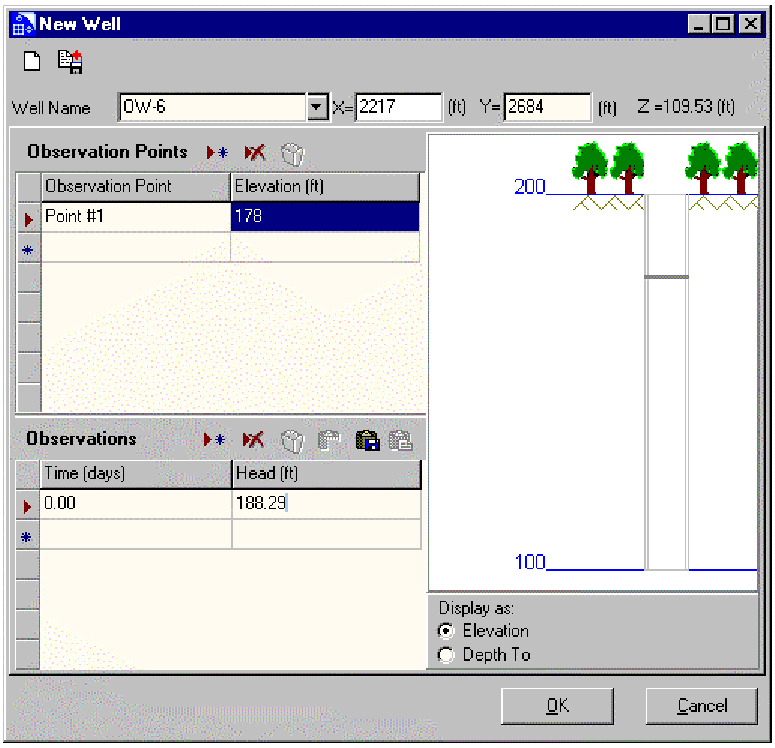
☞ **Wells** (from the top menu bar)

☞ **Head Observation Wells**

You will be transferred to the Head Observation Input screen where you can add, delete and move observation wells or import observation well data from a text file.

☞ **[Add Obs.]** (from the left menu bar)

Move the mouse pointer into the model domain and click the left mouse button to add an observation well at any location. A **New Well** dialogue box will appear, as shown in the figure below, which will allow you to enter the observation well information.



Using the figure above as a reference, enter the following observation well information:

Well Name: **OW-6 ⇔**

X: **2217 ⇔**

Y: **2684**

Elevation: **178** (in the Observation Points Table)

Head: **188.29** (in the Observation Table)

☞ **[OK]** (to accept these values)

A green well symbol will appear on the screen indicating the location of the observation well.

Since there is a total of 18 observation wells (see Table 1), it would be a somewhat tedious task to graphically enter all of the observation wells. With this in mind, Visual MODFLOW allows you to import a space delimited ASCII text file (**.TXT**) of the observation well data with a data format similar to Table 1 (Well Name, X and Y coordinates, Screen elevation, Average Observation Head).

Well Name	X-Location (feet)	Y-Location (feet)	Screen Elev. (feet amsl)	Avg. Obs. Head (feet amsl)
OW-1	2116	2485	150	190.26
OW-2	2162	2453	175	189.99
OW-3	2134	2433	182	189.92
OW-4	2186	2434	178	189.29
OW-5	2148	2511	174	189.76
OW-6	2217	2684	178	188.29
OW-7	2443	2750	171	186.90
OW-8	1919	2624	169	191.41
OW-9	1799	2626	184	192.73
OW-10	2087	2637	182	190.39
OW-11	2527	2534	181	186.45
OW-12	1147	3662	187	196.20
OW-13	1138	2150	191	196.40
OW-14	2876	2737	170	182.89
OW-15	2737	2550	176	184.12
OW-16	2530	2330	181	186.39
OW-17	3237	2141	170	177.31
OW-18	3213	2625	170	177.72

Table 1: Observed Heads - Steady State, Non-Pumping Conditions

☞ **[Import Obs]** (from the left menu)

An **Import Observations** dialogue box will appear on the screen prompting you to select the **.TXT** file containing the observation well information.

☞ **sun_obs1.txt**

☞ **[Open]**

Visual MODFLOW will notify you that OW-6 already exists within the database. You are prompted with a number of options on how Visual MODFLOW should proceed with the file import.

☞ **[Yes]** (to replace the information for OW-6)

The calibration information for the steady-state, non-pumping condition is now completed. To incorporate the observation well data for model calibration purposes, the steady-state, non-pumping simulation must be re-calculated.

Section 2: Model Run in Steady-State, Non-Pumping Conditions

☞ **[F10 - Main Menu]** (from the bottom button bar)

☞ **Run** (from the Main Menu)

☞ ⊙ **Steady State Flow**

☞ **[OK]**

At this point, you may be wondering whether the pumping well that you assigned will be used in this steady-state simulation. If you recall, there were two stress periods specified in the well pumping schedule; the first stress period specified a pumping rate of 0.0 US GPM (i.e. it was not pumping); and the second stress period specified a pumping rate of 125 US GPM. Since this is a steady-state simulation, *Visual MODFLOW will only use the information from the first stress period* for each pumping well and time-varying boundary conditions specified in the model.

☞ **Run** (from the top menu bar)

☞ **[Translate & Run]** (to execute MODFLOW and MODPATH)

Once this is completed, press **[Exit]** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

Section 3: Visualizing the Model Results

In order to visualize these new results,

☞ **Output** (from the top menu bar)

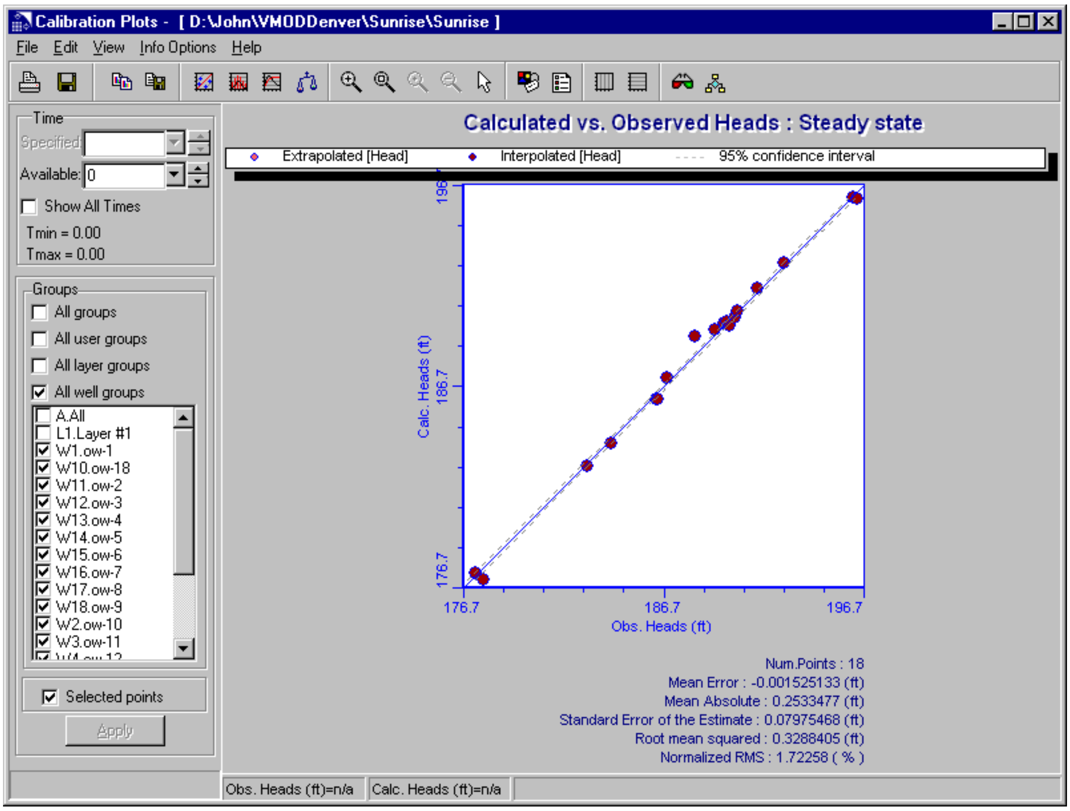
☞ **[Graphs]** (from the left menu bar)

Once the **Calibration Plots** dialogue box is presented,

☞ ☑ **All well groups**

☞ **[Apply]**

A Calculated Heads vs. Observed Heads graph will be displayed as shown in the following figure with calibration statistics provided below the graph.



The values for calculated heads are plotted along the left-hand axis, while the values for observed heads are plotted along the bottom axis. The points represent the head observation well data points. A perfect fit would have all of the data points directly on the 45° line. If a data point is above the line then the model is over-predicting the heads in the system, and if the data point is below the line then the model is under-predicting the heads.

A rough 'rule of thumb' is to have a **Normalized RMS** of 10% or less. Also, the **Mean Error** should be low as most modelers like to see a value less than 0.2, although there is no general consensus on this.

This example demonstrates a very good fit between the observed and calculated heads, which would lead you to believe that it must be the correct solution. However, for a simplistic steady-state model like this one, it is rather easy to find a number of input parameter combinations that will provide an equally good fit between the calculated and observed data. This point will be demonstrated in the following steps where you will reduce the conductivity values and Recharge by an order of magnitude and still get the same results.

☞ **File**

☞ **Exit**

☞ **[F10 - Main Menu]** (from the bottom button bar)

☞ **Input** (from the top menu bar)

☞ **Properties** (from the top menu bar)

☞ **Conductivity**

You will be transferred to the Conductivity Input screen where you can edit the aquifer hydraulic conductivity properties.

☞ **[Database]** (from the left menu bar)

A K Property Database dialogue box will appear which allows you to edit the Kx and Kz values of the aquifer material(s). Double-click in the box labeled Kx and enter the following:

Kx (ft/d): **1** ⇔ (0.3 m/d)

Ky (ft/d): **1** ⇔ (0.3 m/d)

Kz (ft/d): **0.1** (0.03 m/d)

☞ **[OK]**

The hydraulic conductivity value for the aquifer has been reduced by an order of magnitude. Intuitively, this will produce higher head levels in areas **NOT** under the influence of a pumping well, as water entering the system through recharge will encounter more resistance to flow before it discharges to the river. However, areas under the influence of a pumping well will experience the opposite effect. In this simulation,

To offset the increased water table elevation you can reduce the amount of recharge.

☞ **[Boundaries]** (from the top menu bar)

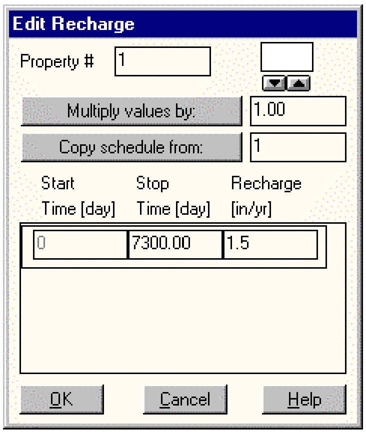
☞ **[Recharge]**

You will then be transferred to the Recharge Input screen.

☞ **[Edit]** (from the left menu bar)

☞ **[Property]**

An Edit Recharge dialogue box containing the recharge input value will appear as shown in the figure below.



Click in the box labeled **Recharge [in/yr],** and enter the following:

Recharge [in/yr]: **1.5**

☞ **[OK]** (to accept this new value)

Essentially, what you are doing is reducing the recharge by an order of magnitude to compensate for reducing the hydraulic conductivity by an order of magnitude (i.e. if you reduce the ability of the system to transmit groundwater, you must compensate by reducing the amount of water entering the system). The effect of the recharge/conductivity ratio with respect to steady-state flow system calibration will be explained in greater detail in the 'How to Build a Model' lecture.

Now return to the Main Menu to run the simulation again.

☞ **[F10 - Main Menu]**

☞ **[Yes]** (to save the property data before exiting)

☞ **Run** (from the Main Menu)

☞ **[OK]** (to accept a Steady State Run Type)

☞ **Run**

☞ **[Translate & Run]** (to execute MODFLOW and MODPATH)

Once this is completed, press **[Exit]** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

☞ **Output**

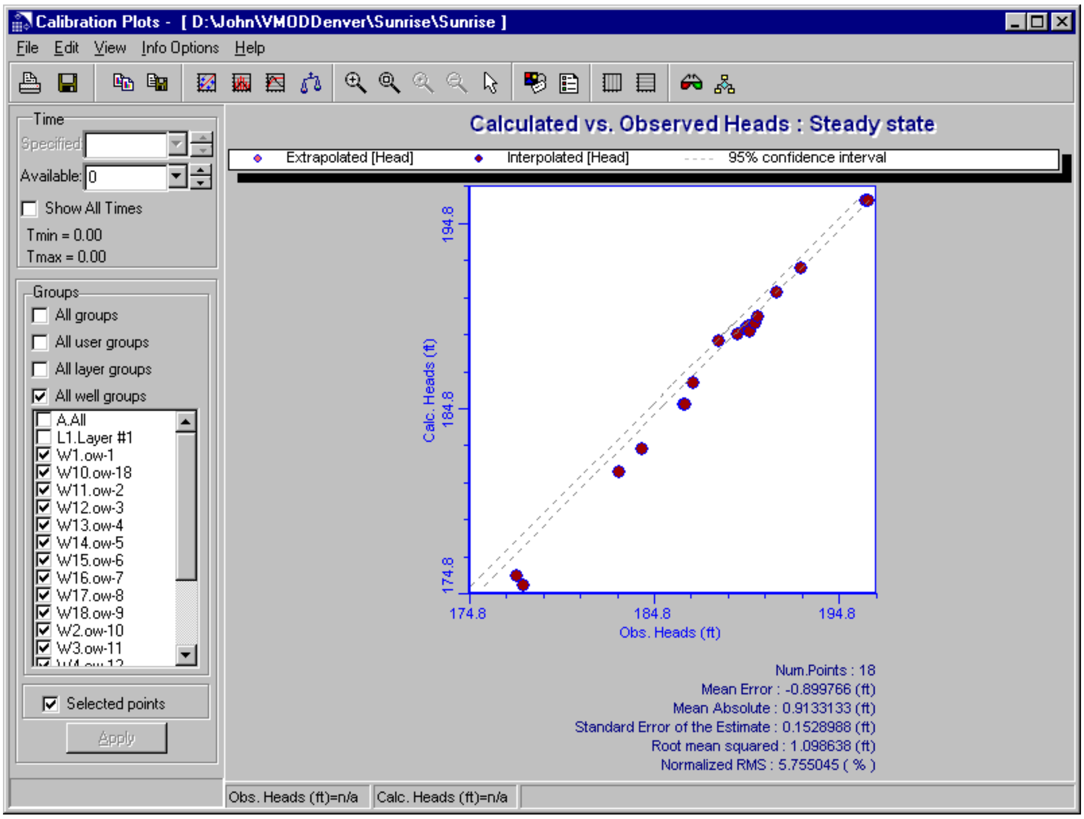
☞ **[Graphs]** (from the left menu bar)

Once the Calibration Plots dialogue box is presented,

☞ **☑ All well groups**

☞ **[Apply]**

A Calculated Heads vs. Observed heads graph will be displayed similar to the figure below.



From a calibration point of view, this simulation is still within acceptable **Normalized RMS** levels (i.e. < 10%). The **Mean Error** levels are greater than the desired 0.2 value, however, this model is still considered calibrated.

These results are nearly the same as the previous simulation in spite of the radically different parameter values. Clearly, this calibrated model does not represent a unique solution to the groundwater flow at this site.

Steady-state groundwater models are not unique and one must rely on using professional judgement in the final selection of model input parameters (i.e. numerical values for these properties must make sense and must correlate with the field data).

☞ **File**

☞ **Exit**

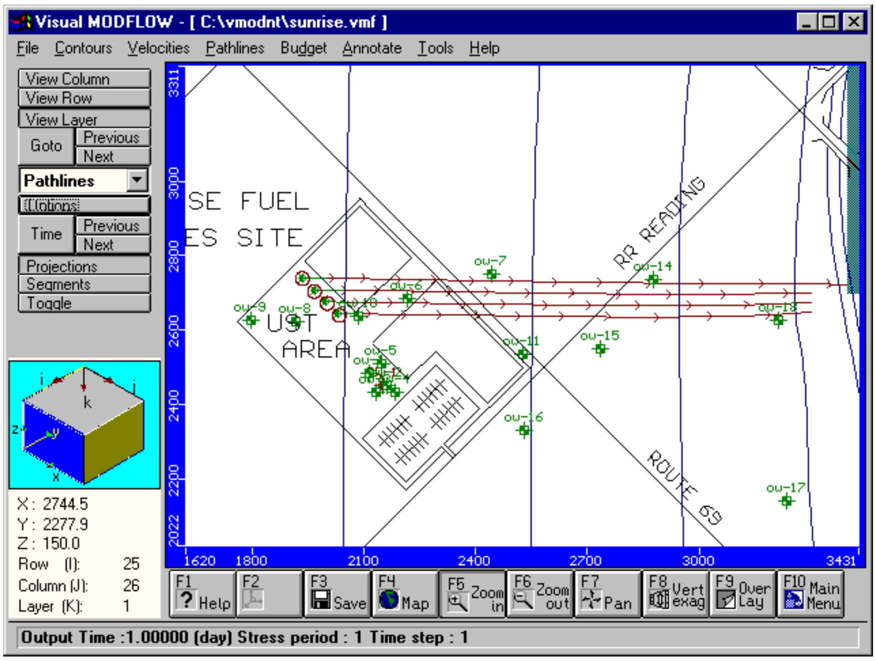
Changes to the hydraulic conductivity and recharge values did not significantly affect the calculated head levels. However, hydraulic conductivity does have a strong influence on pathline time markers and time-related capture zones, as groundwater velocities are directly proportional to hydraulic conductivity.

☞ **Pathlines** (from the top menu bar)

☞ **[F5 - Zoom In]** (zoom in to the area around the Sunrise Fuel Supplies site)

To determine the time interval of the pathline time markers,

☞	**[Options]**	(from the left-hand menu bar)
type:	**2000**	(⦿ Regular every ___ days.)
☞	**[OK]**	(to accept these settings)



For this model, the groundwater plume will require a much longer period of time to migrate off-site than the previous simulation (about 10 times longer!).

Pathline travel times are also very sensitive to the effective porosity of the soil material(s). For the purpose of calculating particle travel times, given the hydraulic conductivity, the most important property is the effective porosity. Effective porosity is defined as a portion of the total porosity through which flow actually occurs (i.e. neglecting dead-end pores). It is a common mistake for modelers to assign the porosity of the soil according to typical soil porosity values, which are usually significantly greater than the values for effective porosity for flow. Given that the effective porosity can often be one-half, or less, of the total porosity, the value entered for effective porosity is a significant influencing factor on the calculated groundwater flow velocities.

If you have time at the end of this exercise, try changing the effective porosity values to see how the pathlines are influenced.

Module 7: Calibrate to Steady-state, Pumping Conditions

In this module, you will examine the influence of the hydraulic conductivity and recharge parameters under steady-state pumping conditions.

Section 1: Adding a Pumping Well

For this part of the exercise, you will assume the proposed pumping well was originally an industrial water supply well that has been operating continuously for the past 5 years at an average pumping rate of 35 GPM (2 L/s). The revised water level monitoring data at the site (to account for the industrial water supply well) is provided in Table 2.

Return to the Main Menu to begin the next part of this exercise.

☞ **[F10 - Main Menu]** (from the bottom button bar)

☞ **Input** (from the top menu bar)

Following the same steps as described before, change the K properties and the recharge boundary condition back to their original values using the Input menu:

Kx (ft/d):	**10** ⇔	(3.04 m/d)
Ky (ft/d):	**10** ⇔	(3.04 m/d)
Kz (ft/d):	**1**	(0.3 m/d)
Recharge (in/yr):	**15**	(381 mm/yr)

Next, you will assign the well information and pumping schedule for the industrial water supply well.

☞ **Wells** (from the top menu bar)

☞ **Pumping Wells**

☞ **[Yes]** (to save the property data before exiting)

Zoom in to the area in the near vicinity of the industrial water supply well if you are not already zoomed in.

☞ **[F5 - Zoom In]**

☞ **[Edit Well]**

Move the mouse pointer over the existing pumping well (Row 20, Column 16) and press the left mouse button. An Edit Well dialogue box will appear with the existing well information for pumping well, PW-1.

Click the mouse in the second line of the pumping schedule and press **[▸ 8]** (the **Delete Schedule Item** button) to delete the second stress period in the pumping schedule.

Change the remaining well information as follows:

End (days): **7300** ⇔

Rate (GPM): **-35** (2.2 L/s)

☞ **[OK]**

Note: *You will be running a steady-state simulation, so the time you enter for the Stop Time is irrelevant. However, a value of 7300 days was used simply to be consistent with the stop times specified for the model boundary conditions.*

Next, you will import the revised water level monitoring data to account for the steady-state pumping conditions at the industrial water supply well.

☞ **Wells** (from the top menu bar)

☞ **Head Observation Wells**

☞ **[Import Obs]** (from the left menu bar)

☞ **sun_obs2.txt** (from the list of available .txt files)

☞ **[Open]**

Again you will be prompted to confirm replacement of the well information

☞ **[All]**

The new observation well data, as defined in Table 2, will be imported and will replace the previous values.

Well Name	X-Location (feet)	Y-Location (feet)	Screen Elev. (feet amsl)	Avg. Obs. Head (feet amsl)
OW-1	2116	2485	150	184.64
OW-2	2162	2453	175	182.30
OW-3	2134	2433	182	183.40
OW-4	2186	2434	178	183.82
OW-5	2148	2511	174	185.21
OW-6	2217	2684	178	186.09
OW-7	2443	2750	171	184.29
OW-8	1919	2624	169	189.12
OW-9	1799	2626	184	190.85
OW-10	2087	2637	182	187.03
OW-11	2527	2534	181	183.89
OW-12	1147	3662	187	196.20
OW-13	1138	2150	191	196.40
OW-14	2876	2737	170	180.37
OW-15	2737	2530	176	181.18
OW-16	2530	2330	181	183.29
OW-17	3237	2141	170	175.84
OW-18	3213	2625	170	175.19

Table 2: Observed Heads - Steady State, Pumping Conditions

☞ **[F3 - Save]** (from the bottom menu bar)

Now run the model simulation.

☞ **[F10 - Main Menu]**

☞ **Run**

☞ **[OK]** (to accept steady state)

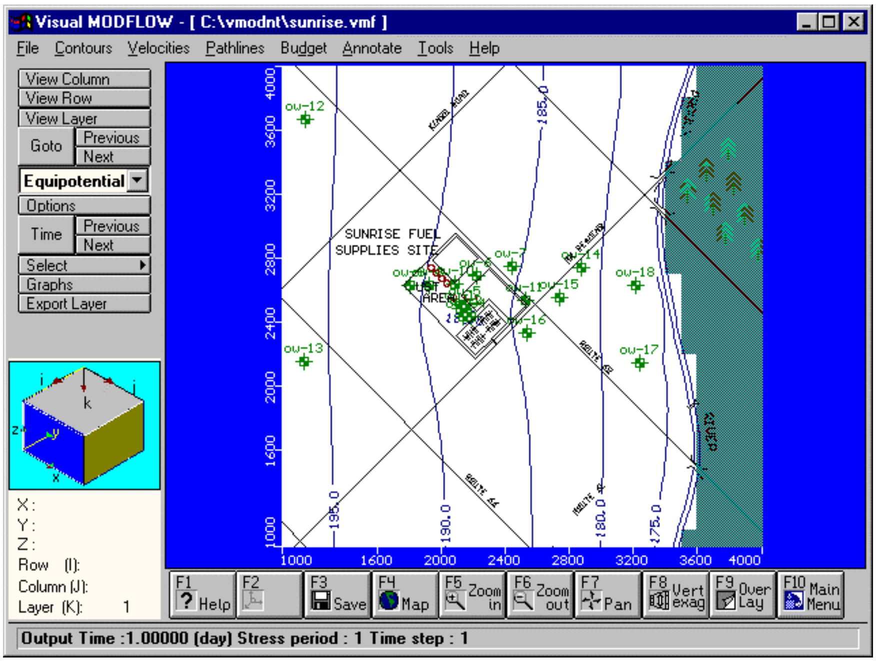
☞ **Run**

☞ **[Translate & Run]** (to execute MODFLOW and MODPATH)

Once this is completed, press **[Exit]** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

☞ **Output** (from the top menu bar)

A contour plot of the steady state heads will be displayed as shown in the figure on the following page.



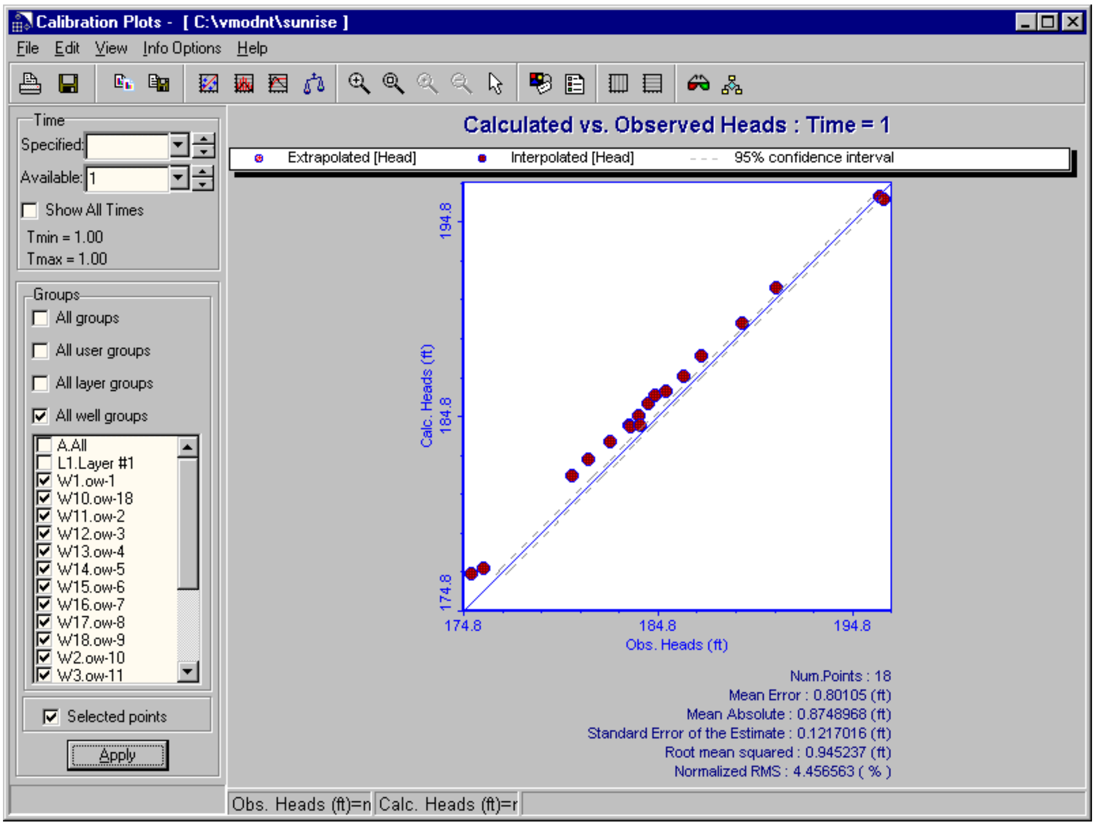
☞ **[Graphs]** (from the left menu bar)

Once the Calibration Plots dialogue box is presented,

☞ ☑ **All well groups**

☞ **[Apply]**

A Calculated Heads vs. Observed heads graph will be displayed that shows a good fit between the observed and calculated heads (see the figure on the following page). The **Mean Error** levels are greater than the desired 0.2 value, however the **Normalized RMS** value is less than 10%. Therefore, this model has been calibrated to match the observation well data.



Visual MODFLOW also allows you to select specific observation wells for plotting the Calculated vs. Observed data. To see the calibration data for the observation wells around the pumping well, return to the display of heads.

☞ **File**

☞ **Exit** (return to the Output screen)

Zoom in to the area around the pumping well, using the **[F5 - Zoom in]** bottom from the bottom tool bar. Then,

☞ **[Select]** (from the left menu bar)

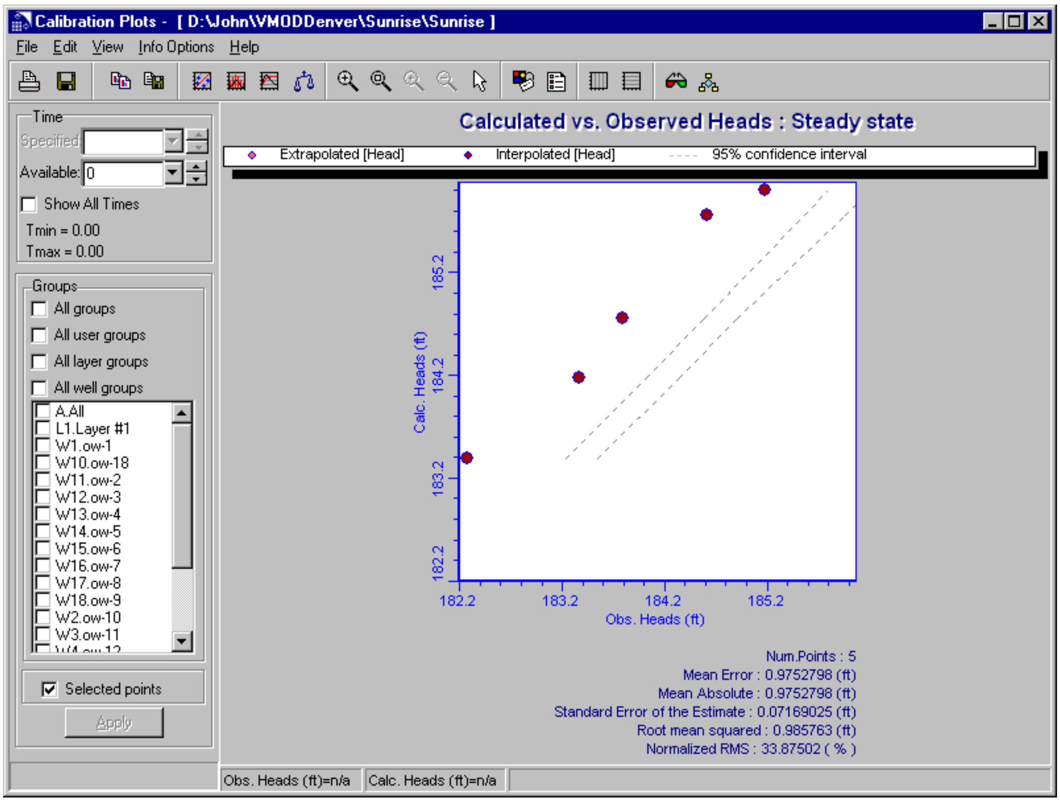
An additional pop-up menu bar will appear prompting you to choose a tool for selecting the desired observation wells.

☞ **[Box]**

Now use the mouse to draw a box around the **FIVE** observation wells in the near vicinity of the pumping well (click the left mouse button to anchor the starting location of the box, then stretch the box around the observation wells and click again to close the box). The observation wells located within the box will be highlighted blue, indicating that they have been selected.

☞ **[Graphs]** (from the left menu bar)

Once the Calibration Plots dialogue box is presented, the Calculated vs. Observed heads graph will be displayed automatically. Only the selected wells will be plotted. This option allows you to determine how the model is calibrating in specific zones of the model and is particularly useful in more complex, multi-layered models with heterogeneous property distributions.



As you can see from the figure above, the **Mean Error** and **Normalized RMS** values are greater than acceptable levels (0.2 and < 10%, respectively). Therefore, although this simulation was deemed calibrated when viewing the entire data calibration set, you can see that the Observed vs. Calculated heads in the near vicinity of the pumping well are drastically different. Therefore, although this model is calibrated, you need to alter your input parameters to find a 'better fit' between the Observed vs. Calculated head levels.

☞ **File**

☞ **Exit** (to return to the Output screen)

Section 2: Calibrating the Pumping Model

Now that you have seen that the model is calibrated to the steady-state pumping conditions, try adjusting the K values and recharge as you did previously to see how these parameters influence the results.

☞ **[F10 - Main Menu]**

☞ **Input**

☞ **Properties**

☞ **Conductivity**

☞ **[Database]** (from the left-hand menu bar)

Change the K properties to the following, to increase the K property for the model:

Kx (ft/day): 11.5 ⇔ (3.5 m/d)

Kx (ft/day): 11.5 ⇔ (3.5 m/d)

Kz (ft/day): 1.15 (0.35 m/d)

☞ **[OK]**

Based on the results of your initial model run, your model is calculating a head distribution that is generally greater than that observed on the site (see figure page 53). To refine your calibration, you need to LOWER the model-calculated heads to achieve a better fit.

You may note intuitively that an increase in the hydraulic conductivity within the area influenced by the pumping well will raise the model-calculated heads, and conversely lower the heads outside the area of influence. As a first step toward improving your calibration, you could INCREASE the hydraulic conductivity across your model. While this will help to achieve a better data fit within the area influenced by the pumping well (see figure on page 54), it will also result in an increase in the head distribution in the area outside the of the model – resulting overall in a worse data fit to the observed values.

By reducing the recharge in concert with increasing the hydraulic conductivity, you may be able to improve the overall data fit. To reducing the recharge,

☞ **Boundaries**

☞ **Recharge**

☞ **[Edit]** (from the left-hand menu bar)

☞ **[Property]**

Click in the box labeled **Recharge [in/yr]** and enter the following information:

Recharge [in/yr]: **7.75**

☞ **[OK]**

Now re-run the model simulation with these new parameters.

☞ **[F10 - Main Menu]**

☞ **[Yes]** (to save the boundary data)

☞ **Run**

☞ **[OK]** (to accept steady state)

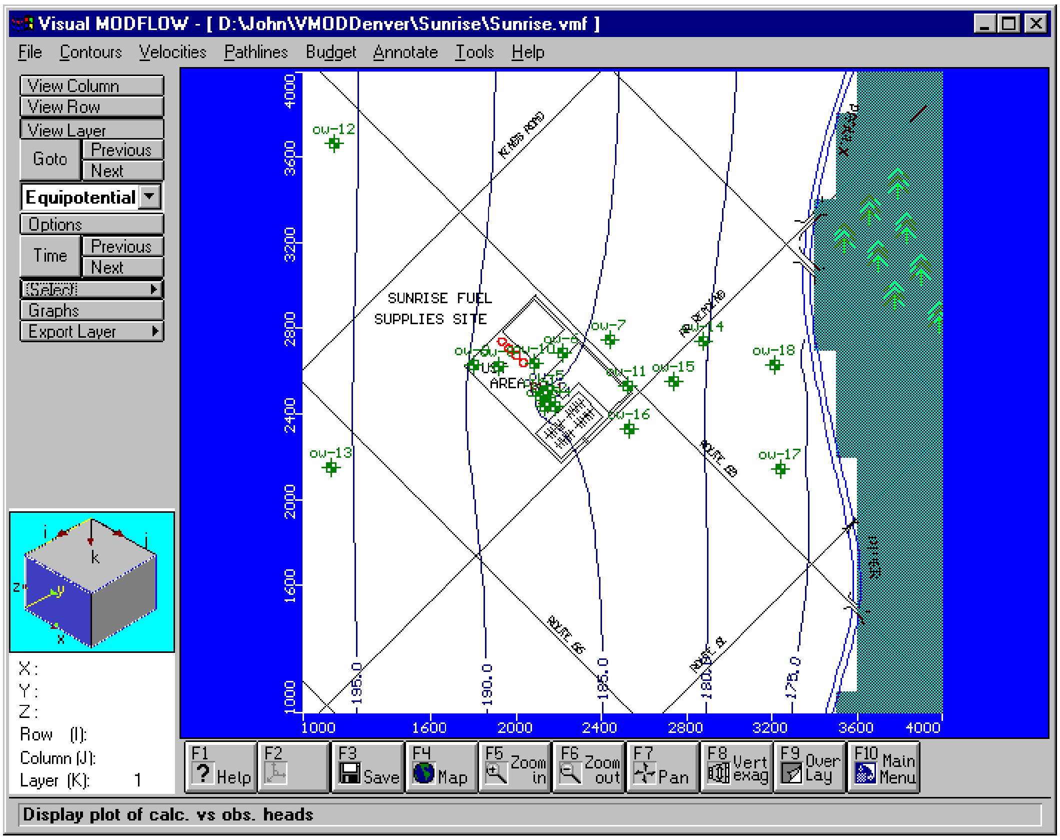
☞ **Run**

☞ **[Translate & Run]** (to execute MODFLOW and MODPATH)

Once this is completed, press **[Exit]** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

☞ **Output** (from the top menu bar)

A contour plot of the steady-state heads will be displayed as shown in the following figure.



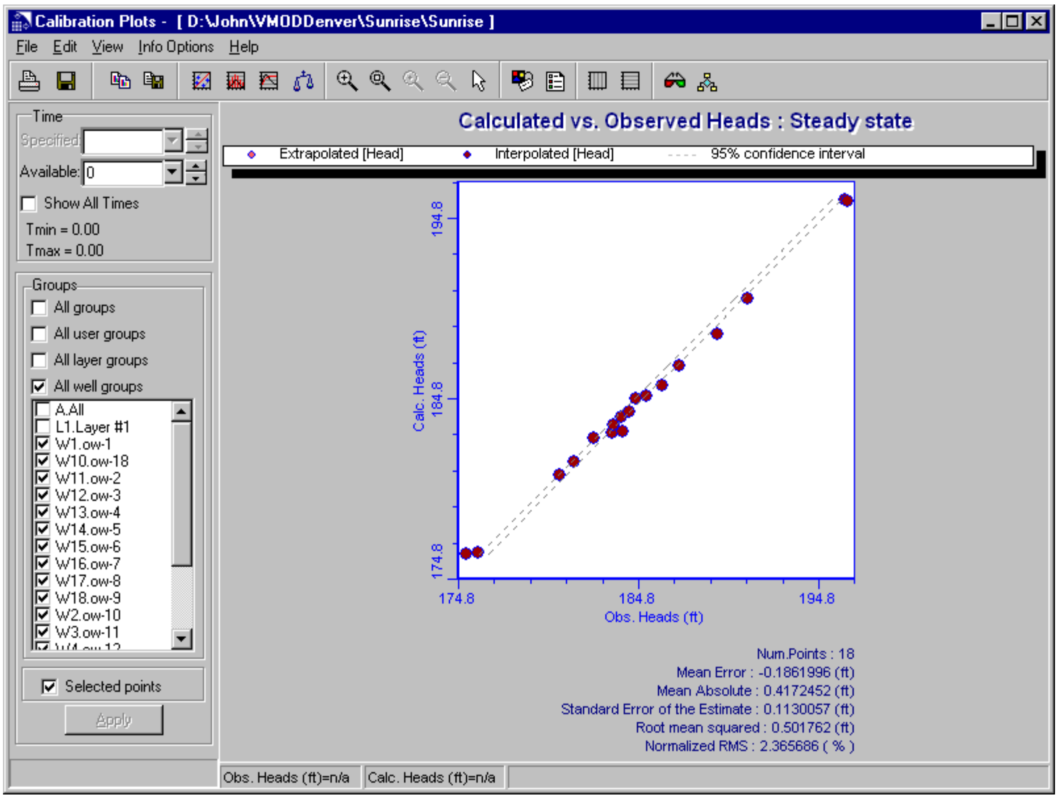
☞ **[Graphs]** (from the left menu bar)

Once the Calibration Plots dialogue box is presented,

☞ ☑ **All well groups**

☞ **[Apply]**

Compare the calculated vs. observed heads for this scenario.



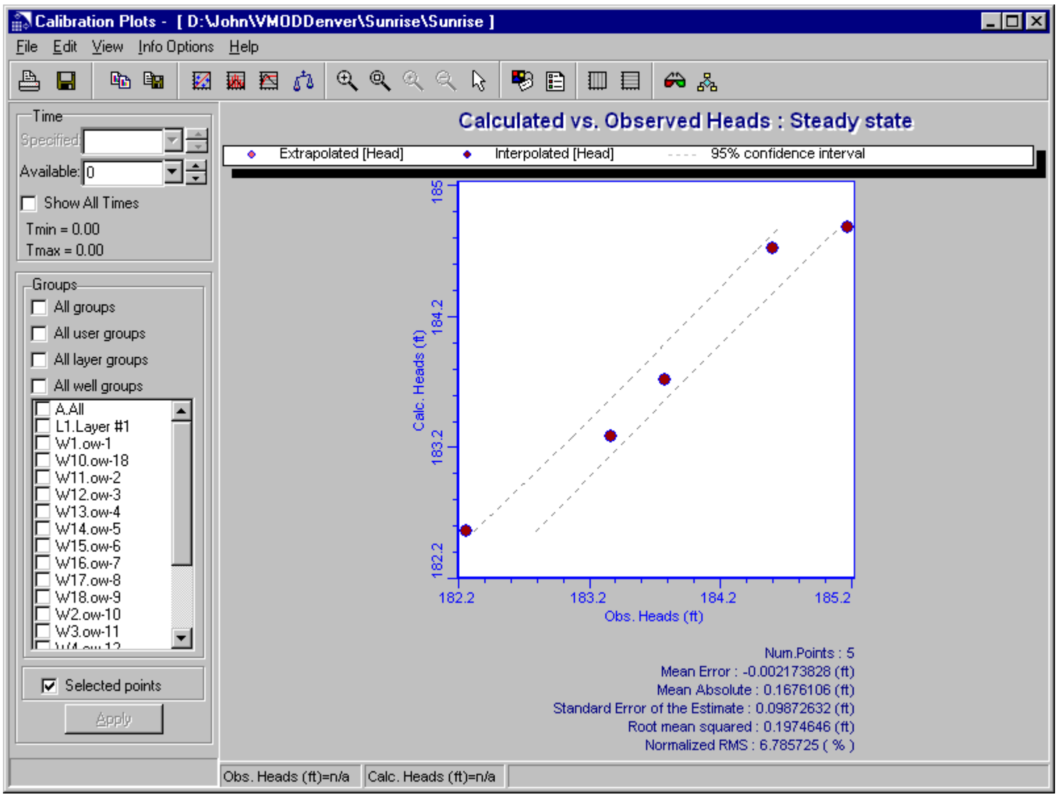
Notice that the fit between the calculated and observed heads is much better (compare to figure on page 53) as indicated by the lower **Mean Error** and **Normalized RMS** values.

As seen in the figure above, increasing the hydraulic conductivity while simultaneously lowering the recharge, resulted in an improved calibration. Let's take a closer look at the observation wells surrounding the pumping well to see how they reacted to the input parameter changes.

☞ **All well groups** (to remove the check mark)

☞ **Apply** (refreshes the graph display)

The FIVE wells that were selected in the previous scenario have been retained, and should be displayed as seen in the figure below.



As you can see from the figure above, the fit between the calculated and observed heads has also improved near the pumping well, as compared with the previous model simulation. The revised input parameter combination for hydraulic conductivity and recharge produced a better-calibrated model, both in the areas near to and outside of the pumping well.

To return to the Visual MODFLOW Output window,

☞ **File**

☞ **Exit**

It is important to recognize that groundwater models do not have unique solutions (combinations of input parameters) that produce a reasonable calibration. This means that there exists an infinite number of input combinations of model parameter values that could produce a reasonable "quantitative" model calibration. And this is the reason why any successful calibration must also include a "qualitative" review of the modeling results, where the hydrogeologist applies his knowledge of the local groundwater hydrology and flow system dynamics to determine which combination of input parameters best represents actual site conditions.

This concludes the Visual MODFLOW Student Version Tutorial Guide. To exit Visual MODFLOW,

☞ **File / Main Menu**

☞ **File / Exit** (from the Main Menu)