Visual MODFLOW

Student Version

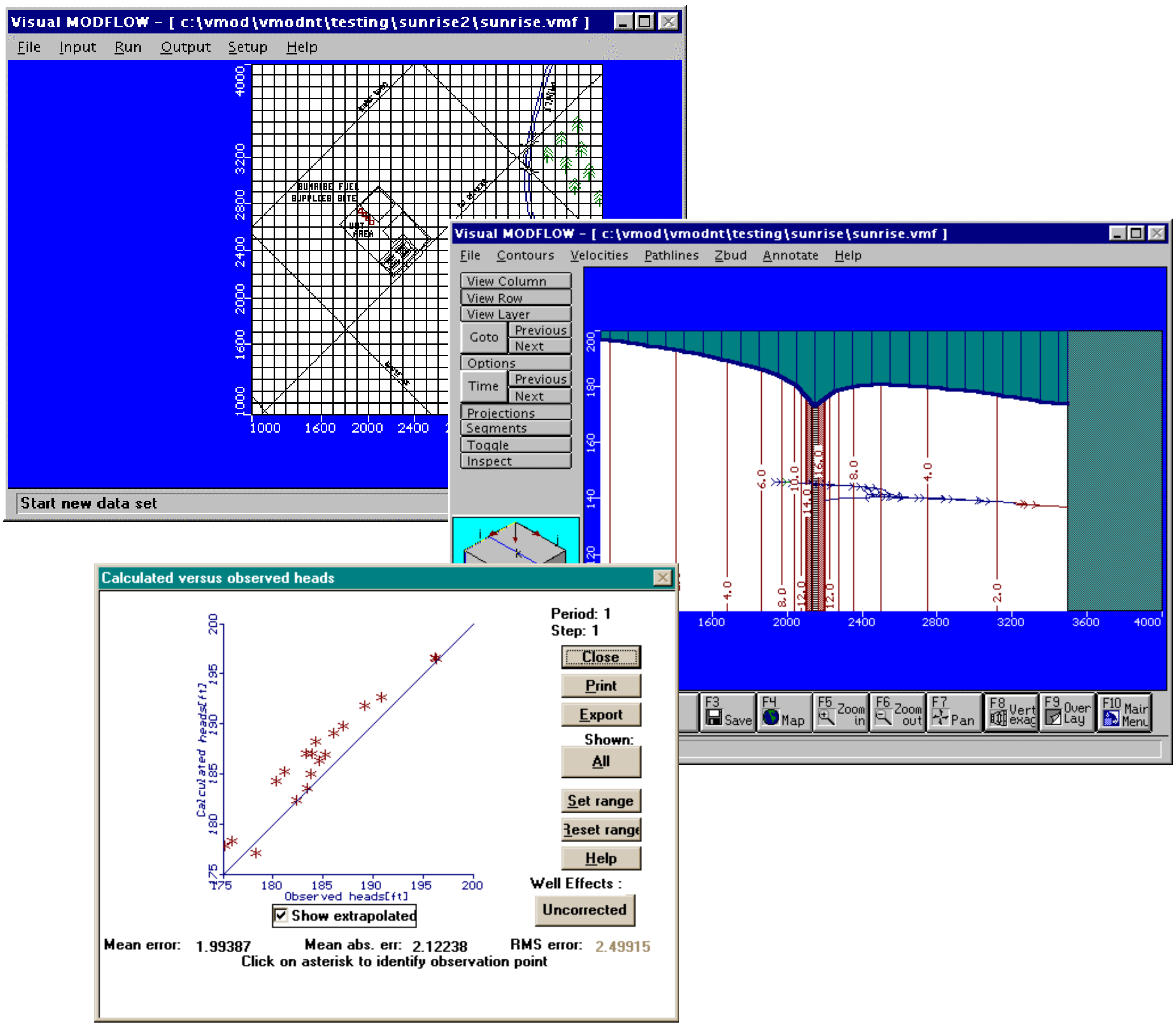


Table of Contents

INTRODUCTION

ABOUT VISUAL MODFLOW STUDENT VERSION

Visual MODFLOW is the most complete and easy-to-use modeling environment for practical applications in three-dimensional groundwater flow and contaminant transport simulations. This fully integrated package combines MODFLOW, MODPATH, and MT3D with the most intuitive and powerful graphical interface available. The innovative menu structure allows you to easily dimension the model domain and select units, conveniently assign model properties and boundary conditions, run model simulations (MODFLOW, MODPATH and MT3D), calibrate the model, and visualise the results with line contours or color shading. The model grid, input parameters and results can be visualized in cross-section or plan view at any time during the development of the model or while displaying the results. For complete three-dimensional groundwater flow and contaminant transport modeling, Visual MODFLOW is the only software package you will need.

Visual MODFLOW was first released in August of 1994 and has quickly become the standard modeling environment for more than 3000 users at consulting firms, educational institutions, and government agencies worldwide. It is used by the United States Geological Survey (USGS), the United States Environmental Protection Agency (USEPA) and is the featured model in many continuing education courses including:

- The NGWA's Visual MODFLOW Short Course;
- The NGWA's IBM-PC Applications in Risk Assessment, Remediation and Modeling;
- The Princeton Remediation Course;
- The University of Bradford Visual MODFLOW Course in West Yorkshire, U.K.;
- Applied 3-D Groundwater Modeling (Angewandte 3D Grundwassermodellierung) in Koln, Germany.

Visual MODFLOW Student Version is a fully functional version that includes all the flexibility and utilities of the Professional Version. The Student Version limits the user to:

- 40 rows by 40 columns and 3 layers;
- 3 property zones; and
- 2 pumping wells

If the functionality of Visual MODFLOW Student Version does not meet the needs of your specific application, please contact Waterloo Hydrogeologic, Inc. to inquire about purchasing a Professional Version.

TERMS AND CONDITIONS OF USE

Visual MODFLOW Student Version can be installed on any number of computers and can be applied by any number of users at licensed educational institutions. This version of Visual MODFLOW is to be used exclusively for educational purposes and is not to be used for any type of professional consulting application or sold for proprietary gain.

ABOUT THE INTERFACE

The Visual MODFLOW interface has been specifically designed to increase modeling productivity and decrease the complexities typically associated with building three-dimensional groundwater flow and contaminant transport models. The interface is divided into three separate modules, the Input Module, the Run Module, and the Output Module. When you open or create a file, you will be able to seamlessly switch between these modules to build or modify the model input parameters, run the simulations, calibrate the model, and display results (in plan view or full-screen cross-section).

The Input Module allows the user to graphically assign all of the necessary input parameters for building a three-dimensional groundwater flow and contaminant transport model. The input menus represent the basic "model building blocks" for assembling a data set for MODFLOW, MODPATH, and MT3D. These menus are displayed in the logical order to guide the modeller through the steps necessary to design a groundwater flow and contaminant transport model.

The Run Module allows the user to modify the various MODFLOW, MODPATH, and MT3D parameters and options which are run-specific. These include selecting initial head estimates, setting solver parameters, activating the re-wetting package, specifying the output controls, etc. Each of these menu selections has default settings, which are capable of running most simulations.

The Output Module allows the user to display all of the modeling and calibration results for MODFLOW, MODPATH, and MT3D. The output menus allow you to select, customize, and overlay the various display options for presenting the modeling results.

SUGGESTED REFERENCE BOOKS

A list of books that may useful for hydrogeology theory and applications are listed below. The full bibliographic references are included in the Bibliography Section.

- Applied Ground Water Modeling: simulation of Flow and Advective Transport, Anderson, M.D. and W.W. Woessner, 1992.
- Physical and Chemical Hydrogeology, Domenico, P.A. and F.W Schwartz, 1990.
- Applied Hydrogeology; 3rd edition, Fetter, C.W. Jr., 1994.
- Practical Applications of Groundwater Models, National Ground Water Association, published Biannually.
- ASTM Standard on Analysis of Hydrologic Parameters and Ground Water Modeling, ASTM, 1996.

HOW TO CONTACT WATERLOO HYDROGEOLOGIC INC.

If you would like to contact Waterloo Hydrogeologic Inc. to purchase the professional version of this software, please contact us at the address below:

Waterloo Hydrogeologic Inc.
180 Columbia Street West - Unit 1104
Waterloo, Ontario, CANADA
N2L 3L3
Phone (519) 746 1798
Fax (519) 885 5262
Email: Info@flowpath.com
Web: www.flowpath.com

OTHER PRODUCTS BY WHI

FLOWPATH

The most complete two dimensional, steady-state, groundwater flow and pathline model. It computes hydraulic heads, pathlines, travel times, velocities and water balances (verified against the USGS MODFLOW, approved by the US EPA, and recommended by the IGWMC).

FLONET/TRANS

A powerful yet easy-to-use two dimensional, steady-state groundwater flow and transient contaminant transport model. Calculates and displays equipotential distribution, streamlines, flow nets, velocity vectors, and temporal graphs of concentration at multiple observation points.

AIRFLOW/SVE

The only comprehensive soil-vapour extraction model to simulate the coupled process of soil-vapour flow and multi-component vapour transport in the unsaturated zone.

PRINCE

A compilation of the 10 Princeton Analytical Models which include 7 mass transport models (one-, two-, and three-dimensional) and 3 two-dimensional flow models.

AquiferTest

An easy-to-use, graphically oriented package for estimating transmissivity, hydraulic conductivity and storage properties for a variety of aquifer types. The program contains analytical solutions for pumping and slug test for confined, unconfined, and leaky confined aquifers.

Visual Groundwater

The first software package to combine state-of-the-art graphical technology for 3-D visualization and animation capabilities with an easy-to-use graphical interface designed specifically for environmental project applications.

We are continually developing additional models with interfaces comparable to Visual MODFLOW. For more information please contact us.

WATERLOO HYDROGEOLOGIC INC. TRAINING AND CONSULTING

Waterloo Hydrogeologic Inc. offers individually tailored training courses in the use and application of all software products. Our modeling courses emphasize practical applications, the interpretation of results (calibration, prediction, etc.), and extensive hands on coverage of the mechanics of using our groundwater models. Courses can be arranged by contacting Waterloo Hydrogeologic Inc.

Waterloo Hydrogeologic Inc. also offers expert consulting services and reviewing service for all numerical modeling problems concerning groundwater flow and mass transport. For further information please contact us.

HARDWARE REQUIREMENTS

To run Visual MODFLOW you will need the following minimum system configuration:

- 486 DX or Pentium computer
- 8 Mbytes of RAM (in excess of the operating system requirements), with approximately 400 Kbytes free in lower memory;
- A high-density (1.44 Mbytes), 3 1/2" floppy drive for software installation;
- A hard drive, with at least 40 Mbytes free;
- A VGA graphics card and a suitable monitor;
- A Microsoft or compatible mouse;
- Dos 5.0 or higher, Windows 3.x, Windows 95, or Windows NT depending on the version of Visual MODFLOW.

The following options make the use of Visual MODFLOW more convenient or efficient, however they are not required:

- A printer with graphics capabilities;
- More than 8 Mbytes of RAM (in excess of the operating system requirements).

If you have any problems with your particular system configuration, please make sure that you followed the installation instructions precisely (see below in **Installing Visual MODFLOW**). If the problem is still unresolved, see **Setup** or **Trouble Shooting**, and contact your hardware experts. Finally, see the section about contacting Waterloo Hydrogeologic Inc.

INSTALLING VISUAL MODFLOW

Visual MODFLOW must be installed on your hard disk to run. Please read the section on hardware requirements to ensure that your system meets the requirements before performing the installation. The following executable file is used to install Visual MODFLOW:

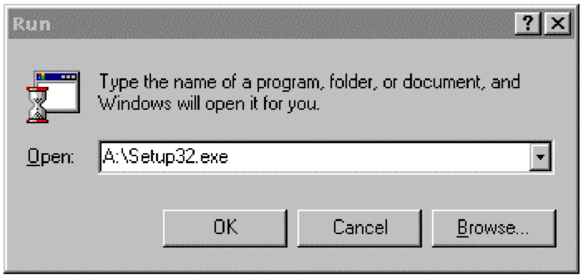
setup32.exe Installs Visual MODFLOW for Windows in Windows 95/NT.

The installation procedure outlined here assumes that Visual MODFLOW will be installed from drive A: (use your CDROM drive if you are installing this from CDROM) to drive C: (destination drive). The default installation directory is \vmodnt, but may be modified. To install Visual MODFLOW in the Windows 95 or Windows NT 4.0 environment follow the succeeding steps.

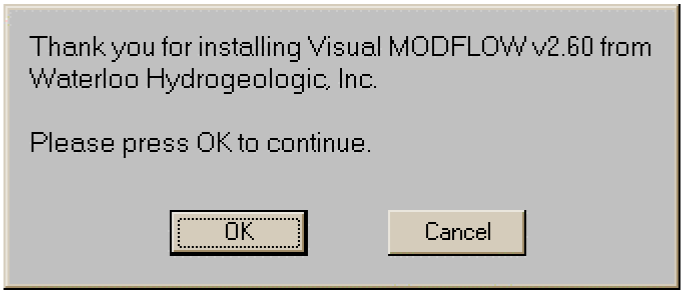
[1] Insert disk #1 into your disk drive.

[2] Enter Windows unless you are already in Windows.

[3] Run the Install Program by clicking on Start in the lower toolbar, then click on Run in the pop up menu and type **A:\Setup32.exe** (or the CDROM drive if you are installing this from CDROM) in the dialogue box. This will start the Visual MODFLOW installation.



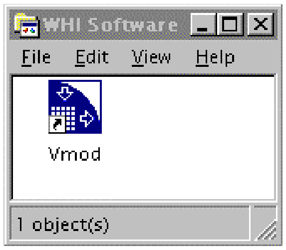
[4] Click on OK to proceed past the title screen.



[5] Enter the path to install Visual MODFLOW or to accept the default path click on [**OK**] to start copying files to the specified path.

[6] After the installation is complete, click on [**OK**] to complete the installation.

You should now see a program group called "WHI Software" on your screen with the VMOD icon inside this window.



STARTING VISUAL MODFLOW

To start up Visual MODFLOW, from Windows or DOS follow the following procedures accordingly. Follow the Windows 95 / Windows NT 4.0 installation procedure. Then simply double click on the Visual MODFLOW shortcut within the WHI Program Group or click on Start/Programs/WHI Software/Vmodflow.

GETTING AROUND

The integrated environment in Visual MODFLOW consists of four major screens: Main, Input, Run, and Output plus the system Setup and Help. The Main Screen contains the following menu options:

File Select a file utility or exit Visual MODFLOW.

Input Go to the Input Screen to modify the current Visual MODFLOW data set.

Run Go to the Run Screen to modify run time parameters and run numerical simulations.

Output Go to the Output Screen to post-process results from numerical simulations.

Setup Choose the numerical engines you want to use.

Help Main help screen and general information on Visual MODFLOW.

Your Microsoft compatible mouse buttons performs as follows:

Left button: This is the regular "click" button. By holding it down on top of an input box and dragging the mouse, it will also highlight the characters that will be overwritten by the next typed characters.

Right button: This button has different functions depending on the context. For example, it closes polygons or ends a line during the assignment of properties, boundaries, etc. During grid design, it also locates grid rows or columns on precise co-ordinates.

The number and letter keys are active only when numerical or text input is required; all other keys are ignored by Visual MODFLOW.

After starting Visual MODFLOW you will be in the main screen. To proceed you must click on File, which leaves you with three options:

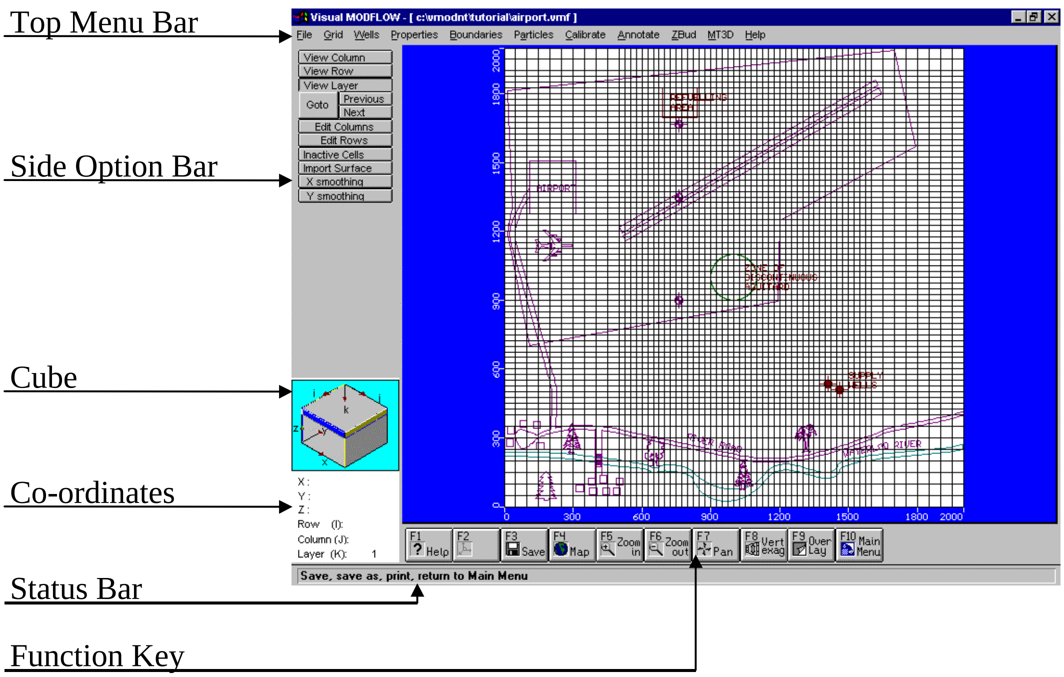
1. **New** to create a new data set, by typing in a new file name;

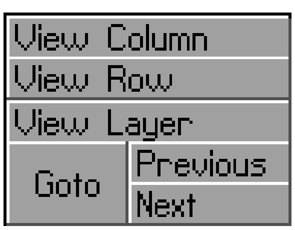
2. Click on **Open,** to open a pre-existing file; or
3. Click on **Import,** to import an existing MODFLOW model or select one of the pre-existing files by highlighting the file path at the bottom of the drop-down menu.

Choose a file by clicking on the file name and then pressing the `Enter` key or double clicking on the file path.

SCREEN LAYOUT

After opening a File and selecting either Input, Run, or Output a screen similar to the figure below will appear.





Top Menu Bar: provides options, which vary depending on the particular section.

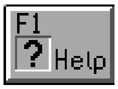
Side Option Bar: contains the view options plus functions particular to the current screen or module. The view options are as follows:

[View Column]	View a cross-section along a column.
[View Row]	View a cross-section along a row.
[View Layer]	Switch from cross-section to plan view.
[Goto]	View a specified row, column or layer.
[Previous]	View previous row, column or layer.
[Next]	View next row, column or layer.

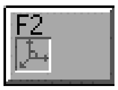
Navigator Cube: provides a simplified 3D representation of the model domain with cross-hairs for locating your spatial location.

Co-ordinates Area: shows the current location of the cursor in real world co-ordinates, and shows the current cell on which the cursor is located.

Function Keys: common functions to the Input, Run, and Output screens.

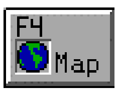


F1 Help on specific screen or highlighted area.

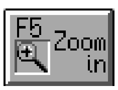


F2 Reserved for future 3-D options.

F3 Save the file as previously named.

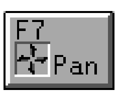


F4 Map pops-up a File Selection window where you can select a .DXF format map file for importing. The map can be toggled on and off in the Overlay window.

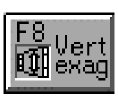


F5 Zoom In allows you to extend a zoom window over the screen with the left mouse button. The right mouse button pops-up a window that allows you to specify the co-ordinates of the zoom area for consistent size plot windows.

F6 Zoom Out resets the screen image to the model extent.

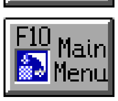


F7 Pan allows the user to click the left mouse button once on the model and drag this point to its new location and click again.



F8 Vertical Exaggeration allows the user to specify the amount of vertical exaggeration seen in the row or column view.

F9 The Overlay window allows you to toggle on or off the various plot and map features (see below).



F10 Main Menu allows you to return to the Main Screen.

Status Bar: a description of each button functionality and use.

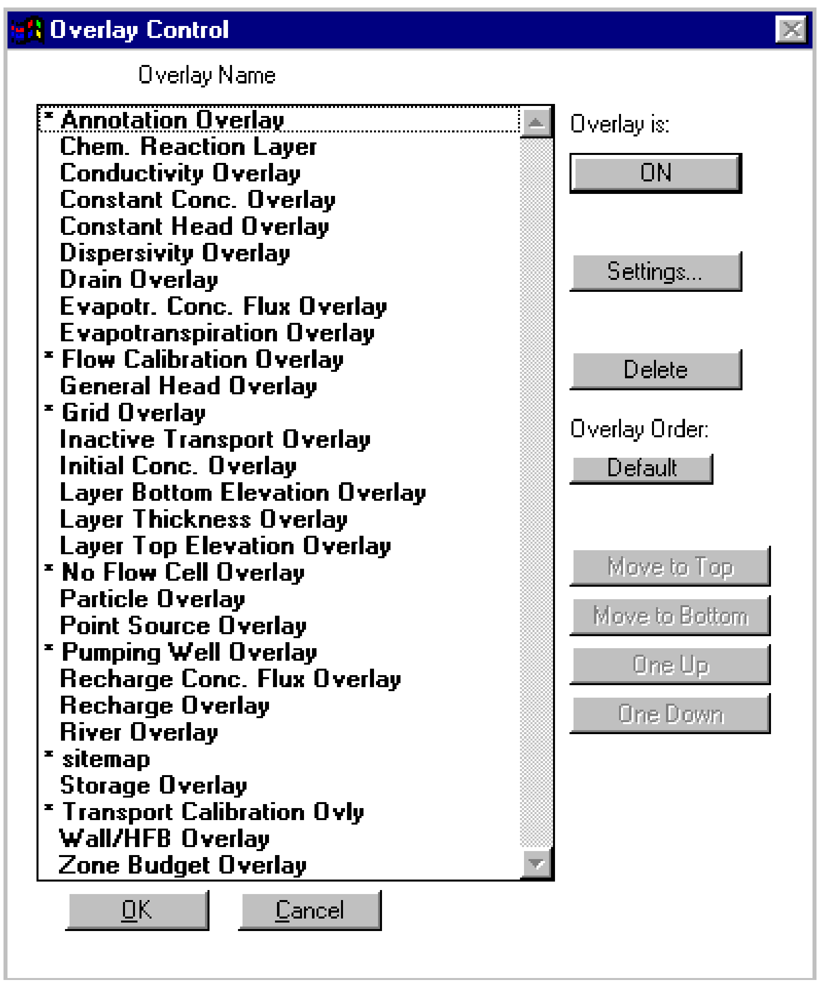
The CAD (Computer Aided Design) environment is one of Visual MODFLOW's most powerful features. It allows fast and easy setup of complicated problems and at the same time allows the user to visually inspect input, thereby eliminating errors that frequently occur during the input of data.

For the CAD environment, a Microsoft (or compatible) mouse is necessary. In all CAD environments, the following keys are active:

Function Keys	As described above.
Enter Key	Select the currently highlighted area.
Esc Key	Cancels current window.
Cursor (arrow) keys	Limited functionality (such as choosing menu options).
Tab Key	Moves from cell to cell.

OVERLAY FEATURES

Various features, such as gridlines, observation wells and conductivity zones, can be turned on or off in the Overlay window. The Overlay window appears when F9 is pressed or when the F9 button at the bottom of the screen is clicked on.



The various overlays allow you to cross check your data or to add or remove features from your plots to make them more readable. Maps that are imported can also be deleted from the Overlay window.

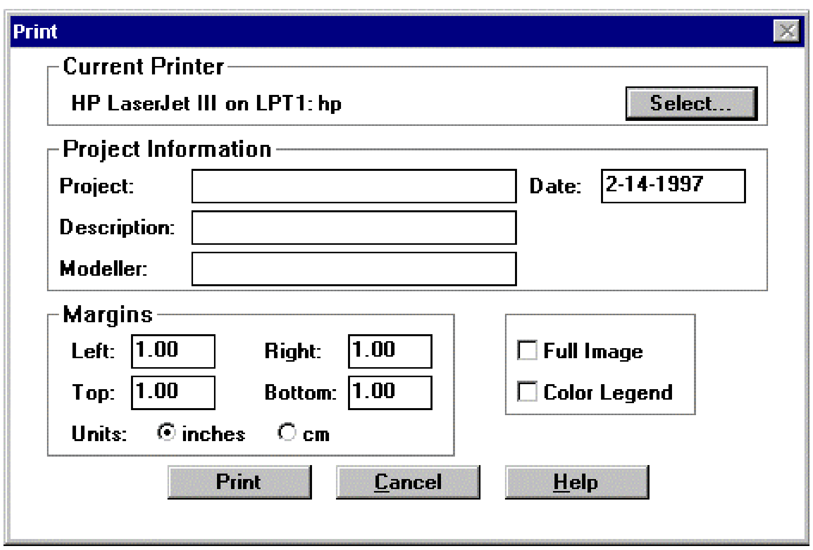
The Overlay dialogue in the Input screen contains all of the overlay features, which can be seen before a model is run. The Overlay dialogue in the Output screen includes the Input screen overlays, as well as the overlays available for visualizing the model results.

The Settings button enables you to modify the display characteristics of some of the overlay features. Zone style features, such as conductivity and recharge, can be displayed as either solid zones or outlines. Colours for the .DXF maps, as well as, the options for calculated features, such as contour lines and velocity vectors can be set with the Settings button.

The order in which the overlays appear in the list will determine the drawing order of the overlays on the screen. The drawing order is reverse to the list order (i.e. the feature at the top of the list is the last feature drawn on the screen). After toggling the **[Default]** button to **[User Defined],** the Overlay list can be modified using the group of four buttons located on the lower right hand side of the Overlay Control dialogue box.

PRINTING IN VISUAL MODFLOW

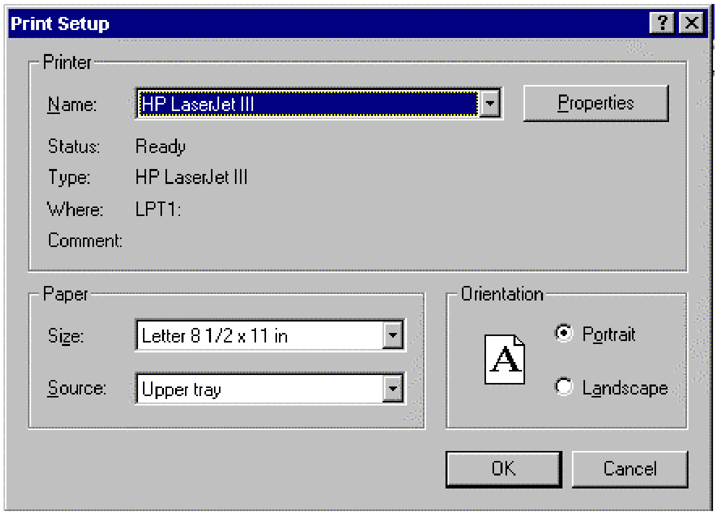
Visual MODFLOW allows you to print the model you are currently viewing. From the Main, Input, Run or Output Screen, select **[File]** and then **[Print]**. You will be prompted with the following dialogue box:



Project information provides the text data for the title block of the plot. The Full Image toggle has been disables in the Student Version so that the title block will always be visible.

For a consistent plotting window, use the right mouse button after selecting Zoom In to specify the viewing area for your model. This feature takes the top and bottom grid co-ordinates and then scales the left and right grid co-ordinates to form a rectangular viewing domain. It also allows the user to revert back to the previous zoom window or to alter the current zoom window.

The default Windows printer is automatically selected as the current printing device. To change the current printer, click on **[Select]**. You will be prompted with the following screen.

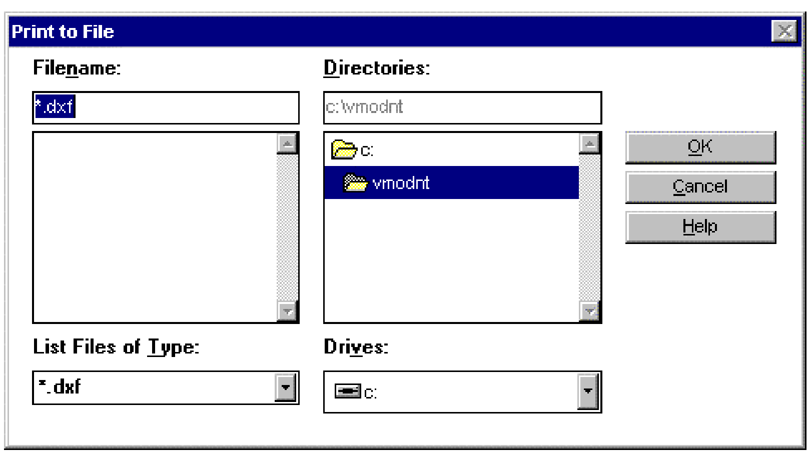


By selecting the arrow beside the Name input box you can switch the current printer to any currently installed Windows printer. Selecting “Properties” brings up the standard Windows print options, which allow you to modify paper, font, and device settings.

PRINTING TO DXF FILE

If you want to export your output to a .DXF file for post processing in a CAD program such as AutoCAD, you can “print” your output to a .DXF file.

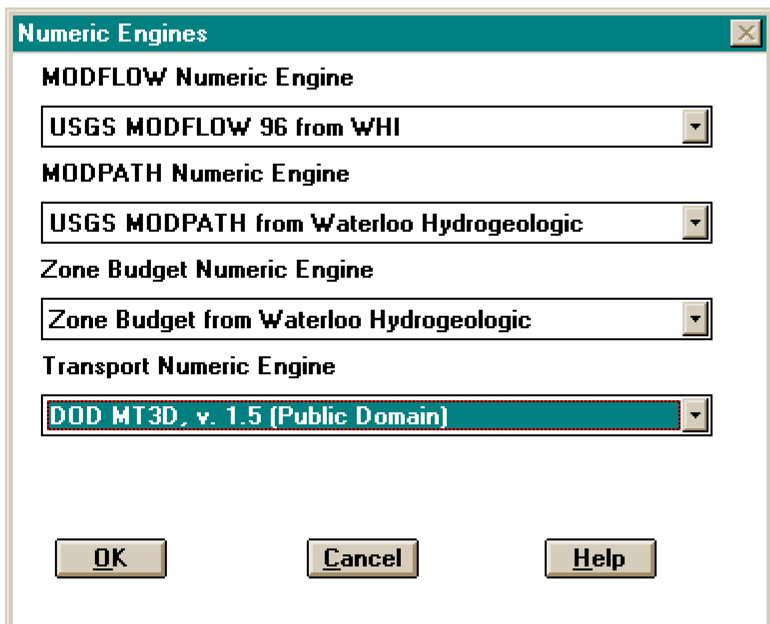
To export output to a .DXF file, select **[File]** from the main menu and then select **[Print to DXF file]**. The following dialogue menu will prompt you for the .DXF filename.



NUMERIC ENGINES SETUP

The Setup option is for selecting different numeric engines for the different Visual MODFLOW packages. To enter Setup, click on [**Setup**] in the top menu. A drop down menu will appear.

Visual MODFLOW allows the user to select different numeric engines (FORTRAN executables) for MODFLOW and MT3D. After selecting [Setup], select [Numeric Engines] from the drop down menu and the **Numeric Engines** dialogue box appears.



There is only one MODFLOW numeric engine available, which is USGS MODFLOW 96 from WHI

For both MODPATH and Zone Budget there is only one numeric engine available: USGS MODPATH from Waterloo Hydrogeologic and Zone Budget from Waterloo Hydrogeologic respectively.

There are four Transport Numeric Engines available:

- EPA MT3D, v.1.1 (Public Domain);
- DOD MT3D, v. 1.5 (Public Domain);

WINDOWS95/NT SWAP FILE

Windows 95/NT manages the systems virtual memory. To change these settings please see your system administrator.

COMMON QUESTIONS AND ANSWERS

Q. Why does Visual MODFLOW sometimes become sluggish?

A: On large data sets, when Visual MODFLOW runs out of RAM to work with, it begins using your hard disk as a temporary area. Moving unused data to and from the temporary area adds additional time to many operations, making more RAM available to Visual MODFLOW decreases the need to use this temporary area. A quick remedy may be to reduce the size of any disk cache such as SMARTDRV.EXE to a smaller value, such as 512k or 1 megabyte in size. This will release additional RAM for Visual MODFLOW. Alternatively, you may want to purchase additional RAM for the computer.

Q. Why is my DXF map HUGE when compared to the model domain?

A: Visual MODFLOW requires that the units of your DXF map to be in decimal units. The DXF file that you are trying to import is either set up to use Engineering or Architecture scale of Feet and Inches. To prepare your DXF file for use with Visual MODFLOW, you will need to edit your DXF file with AUTOCAD or other suitable CAD software. First, you will need to change the units to DECIMAL. This will convert your drawing to decimal inches. This however, will cause the drawing to be 12 times too large. To remedy this, you need to use the SCALE command to scale down the entire drawing by a factor of 0.0833333 (1/12th). After the scaling has completed, you need to save the drawing again using the DXFOUT or similar command. The file should now be successfully imported into Visual MODFLOW at the right scale. A future release of Visual MODFLOW will recognize this situation and scale the drawing automatically.

We want your feedback…

At Waterloo Hydrogeologic, we are dedicated to providing a product that is tailored to our customer’s needs. Therefore, we are very interested in your thoughts on Visual MODFLOW and welcome your suggestions and comments. We cannot promise to include all your suggestions in the next release, but you have our assurance that all suggestions will be carefully considered. Please indicate a priority or an importance ranking for each of your suggestions.

Visual MODFLOW Tutorial: Sunrise Fuel Supplies Co.

INTRODUCTION

The following laboratory exercise consists of two parts that correspond to the lecture material, which will be covered during the course. These exercises include:

- Part A - Building a Groundwater Flow Model
- Part B - Calibrating a Model (Steady-State and Transient)

The purpose of these exercises will be to introduce you to the basic steps necessary to build a groundwater flow and contaminant transport model using Visual MODFLOW. The models which you will be building and modifying during these exercises are simplified examples which 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.

DESCRIPTION OF THE PROBLEM

Approximately 2 years ago the company inventory manager discovered that the 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 the USEPA and a detailed site investigation was initiated to determine the extent 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 over the course of 6 months

DESCRIPTION OF THE SITE

The site is relatively flat with an average elevation of approximately 200 feet above mean sea level (AMSL). The local topography slopes from northwest to southeast towards the Proulx River, which is located about 2000 feet east of the site. The Proulx River flows from north to south with a surface water elevation that ranges from 174 - 176 ft AMSL in the north, to 169 -

171 ft AMSL in the south (depending on the time of year). The average depth of the Proulx River is approximately 5 ft.

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 below ground surface. The results of the slug test analyses and pumping test analyses indicate a soil hydraulic conductivity, which ranges from 0.5 - 15 ft/day. The water level monitoring data at the site indicate 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.

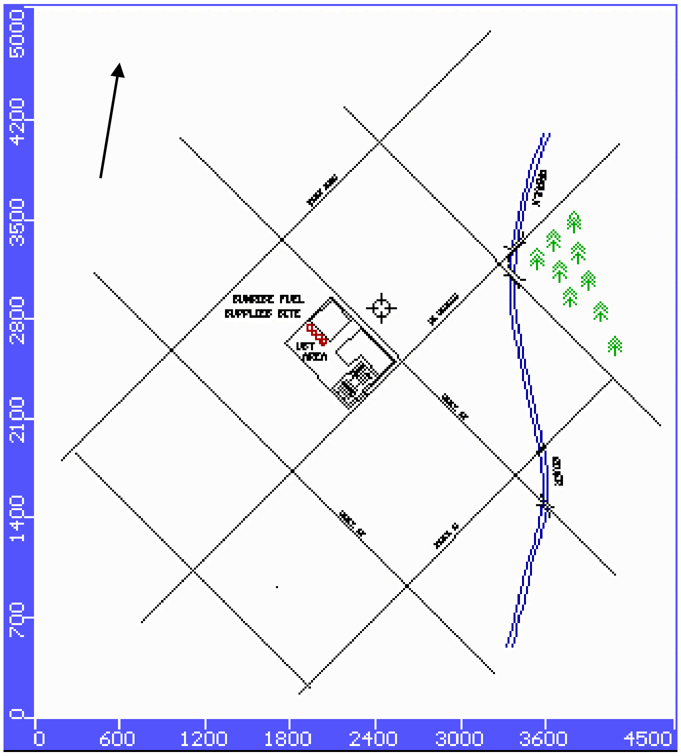


Figure 1: Map of Sunrise Fuel Supplies Site and Surrounding Area

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 potential extent of contamination of the site and to determine the feasibility of a pump and treat remediation system.

Part A

LEARNING OBJECTIVES

- To conceptualize a model and solution
- To dimension and build a model grid
- To input model properties and boundary conditions

- To assign particle tracking locations
- To run MODFLOW and MODPATH
- To visualize the model results
- To assign a pumping well
- To refine the model grid

MODEL CONCEPTION

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 the 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 - 198 ft AMSL throughout the year, with an average elevation of 196.5 ft AMSL. The groundwater flow direction across the site is predominantly west to east.

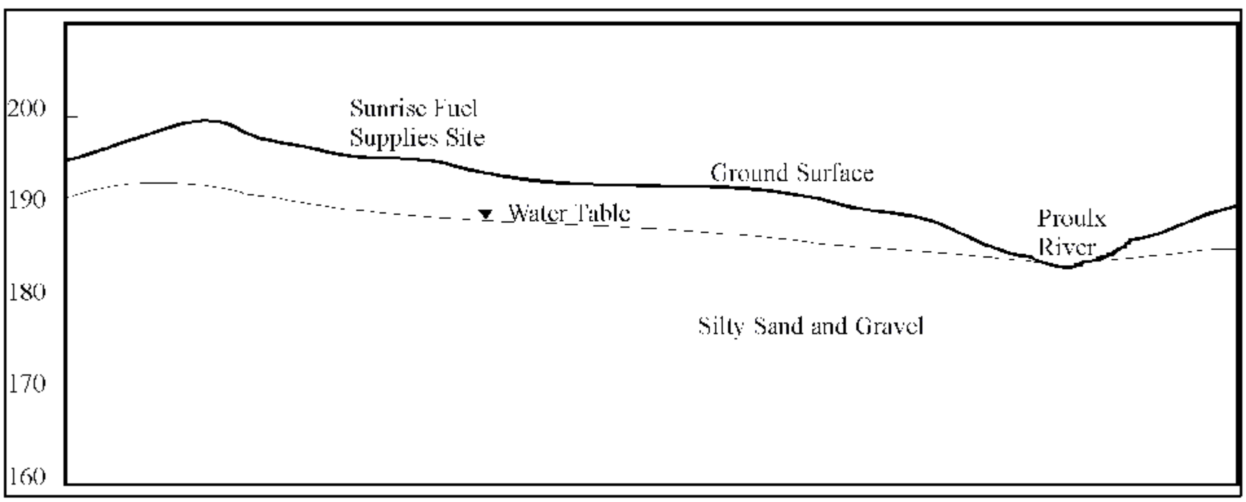


Figure 2: Cross-section through the Sunrise Fuel Supplies Site

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

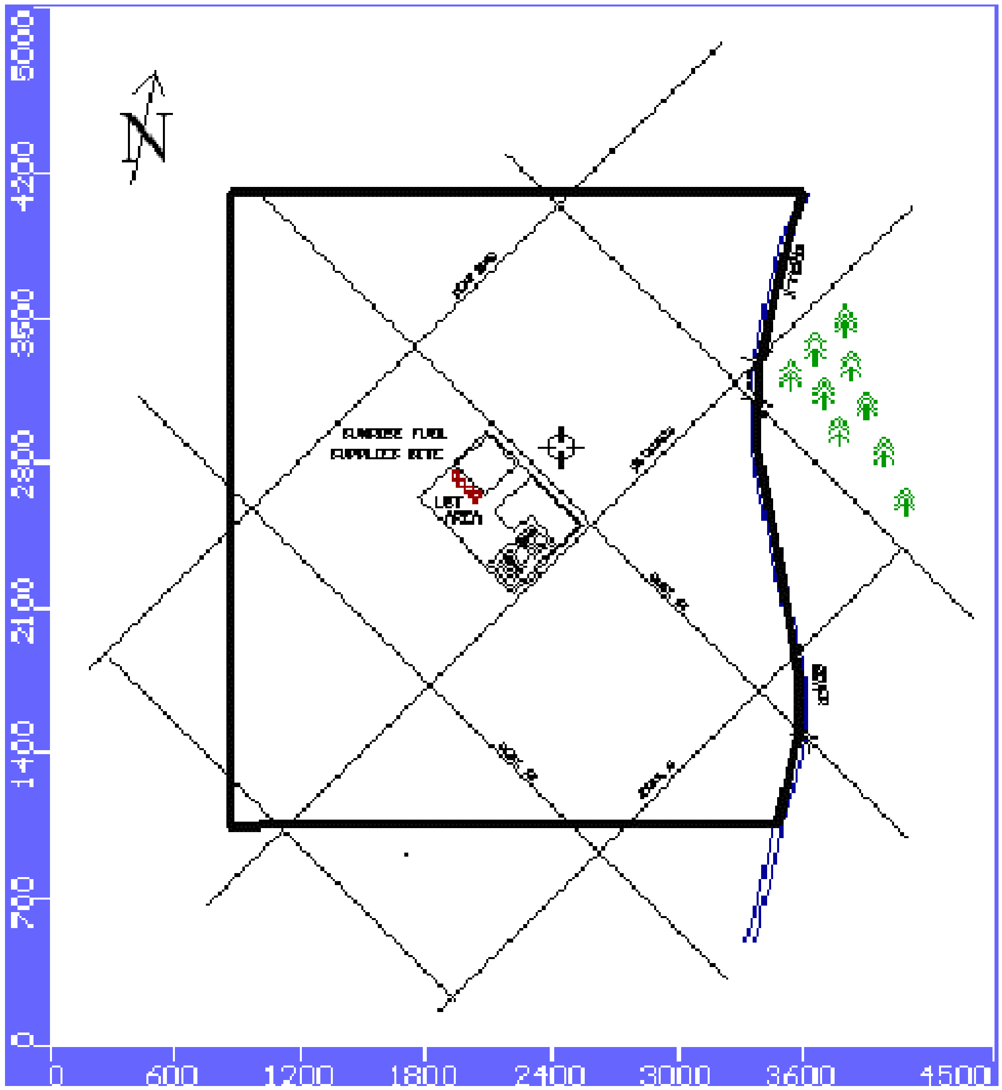


Figure 3: Model Domain Boundary for Sunrise Fuel Supplies Site

A Constant Head (196.5 ft AMSL) boundary condition was selected along 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 which are 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 such that they would not be influenced by any local stresses to the groundwater flow system within the site property.

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

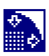
☞☞ - to double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click on or values to type in.

[….] denotes a button to click on, either in a window, or in the side or bottom menu bars.

STARTING VISUAL MODFLOW

On your Windows95 desktop, you will see an icon for Visual MODFLOW.

☞☞  Visual MODFLOW to start the program.

You are now in the opening screen of Visual MODFLOW. To proceed to the Main Menu,

☞ **[OK]**

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)

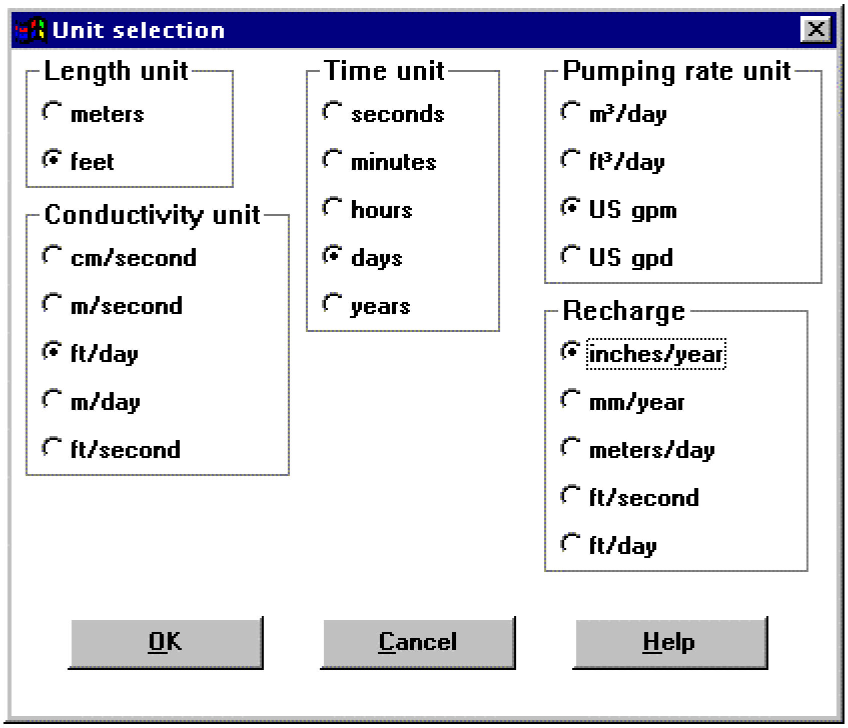
A Create New File window will appear prompting you to enter the filename of the Visual MODFLOW project.

type: sunrise (in the box labelled Filename)

☞ **[OK]**

A Unit Selection window will display all of the available units for the model input parameters.

Use the mouse to select the following units.



Length unit: ⊙ **feet**

Conductivity unit: ⊙ **ft/day**

Time unit: ⊙ **days**

Pumping rate unit: ⊙ **US gpm**

Recharge: ⊙ **inches/year**

☞ **[OK]** (to accept these units)

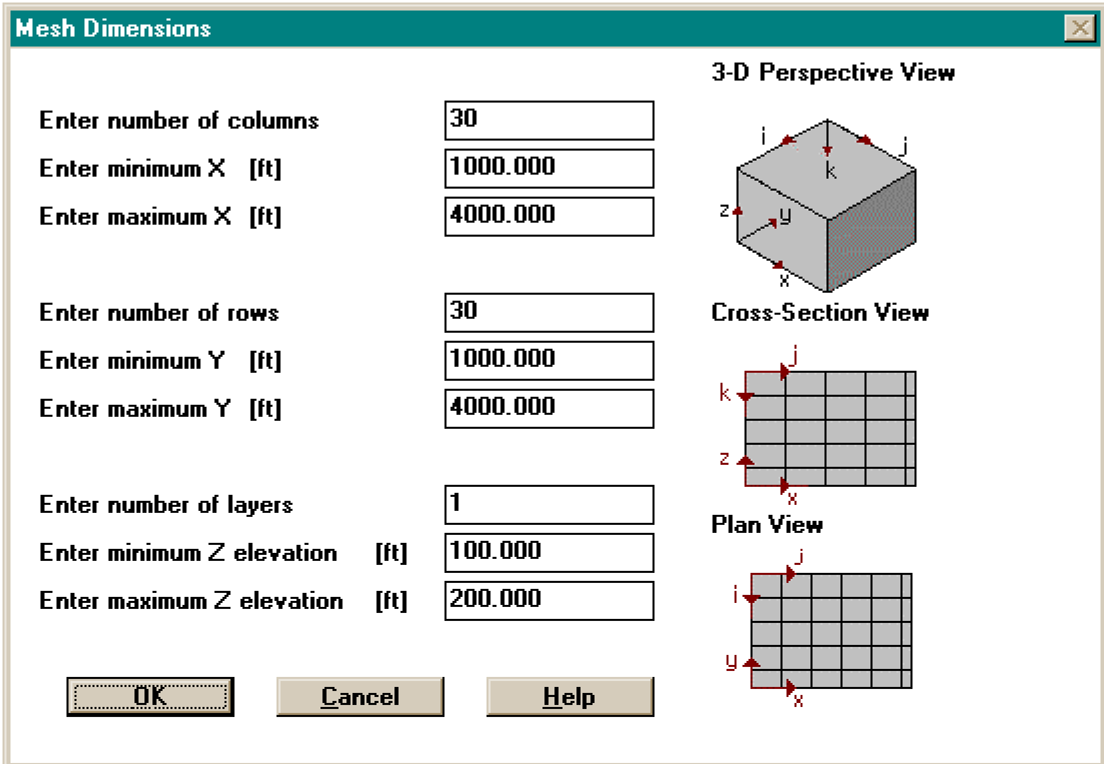
An MT3D Units window will then appear prompting you to select the desired units for a contaminant transport simulation.

☞ **[OK]** (to accept the default units)

A Use Map Co-ordinates window will then ask '**Retrieve Site Co-ordinates from a DXF MAP?**'

☞ **[Yes]**

A DXF File Selection window will provide a listing of the available .dxf files in the working directory. Use the mouse to double-click on the filename '**sunrise.dxf**' to select it for this project. Visual MODFLOW will read the minimum and maximum x and y co-ordinates for the model domain from the '**sunrise.dxf**' file and then a Mesh Dimension window will appear as shown in the figure below.



Enter the following values in the appropriate boxes:

Enter number of columns: **30** ↔

Enter minimum X (ft) **1000** ↔

Enter maximum X (ft) **4000** ↔

Enter number of rows **30** ↔

Enter minimum Y (ft) **1000** ↔

Enter maximum Y (ft) **4000** ↔

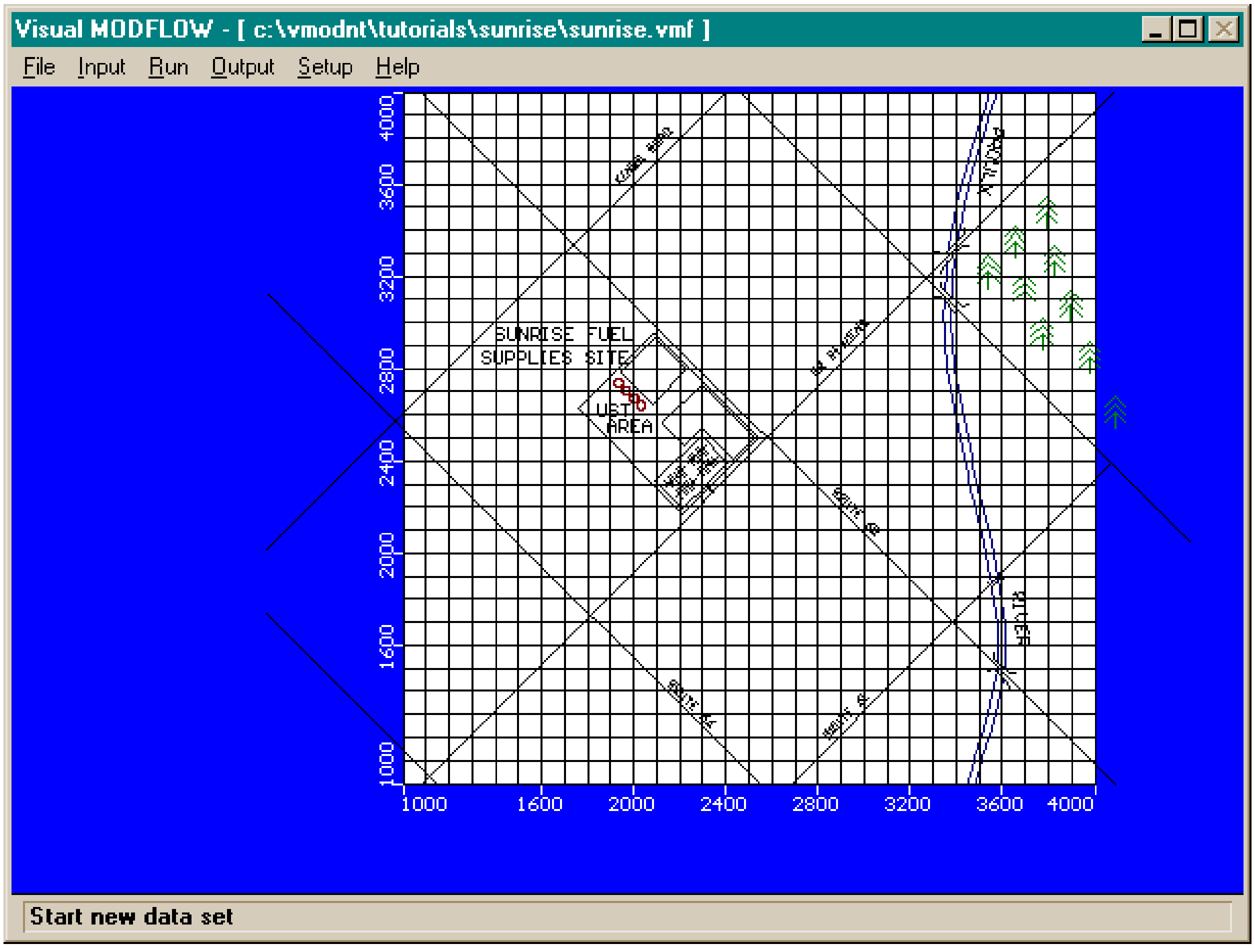
Enter number of layers **1** ↔

Enter minimum Z elevation (ft) **100** ↔

Enter maximum Z elevation (ft) **200** ↔

☞ **[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 as shown in the following figure.



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

☞ **Input** (from the top menu bar)

The graphical interface will then transfer you to the Input Menu, with all of the different types of model input parameters listed across the top menu bar. 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 the groundwater flow in these cells does not contribute to the groundwater flow in the model domain. In this example, we are assuming that 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 the groundwater flow in the area east of the river does not contribute to the groundwater flow in the area to the west of the river. Therefore, the model grid cells located to the 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)

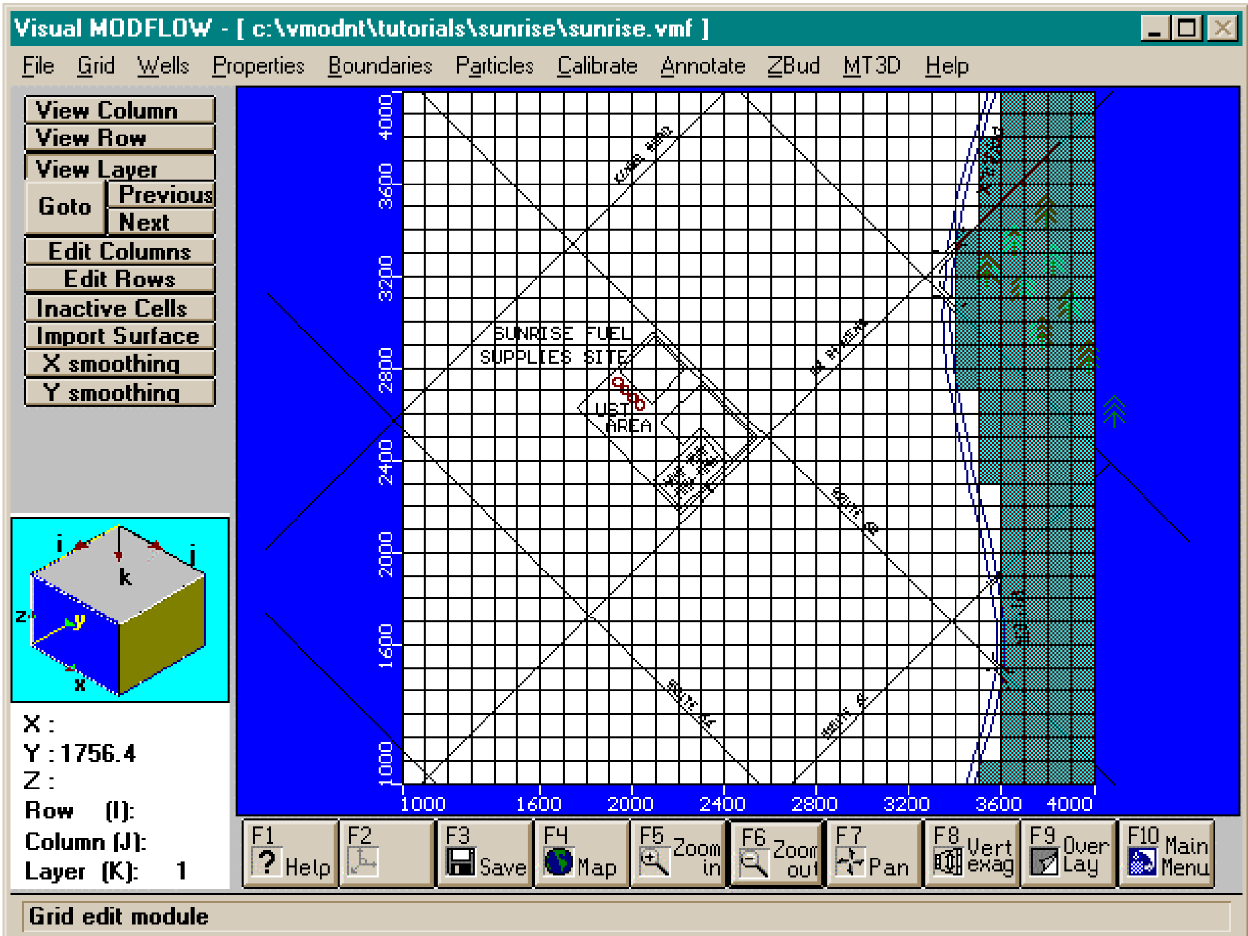
This graphical tool will allow you to digitize a polygon around an area to the east of the Proulx River. Move the mouse to the top right-hand corner of the model grid and click the left mouse button once to anchor the polygon. Then move the mouse down to the lower right-hand corner of the model grid 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 to the 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 below, indicating the cells, which have been designated, as inactive.

If the polygon, which you assigned, has missed some cells which 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, which you want to assign as inactive and click the left mouse button. To paint several cells inactive, press and hold the left mouse button while you drag the mouse across the cells. Alternatively, if you would like to re-activate cells which have been accidentally set as inactive, PRESS 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 to ensure that no grid cells are neglected when you are assigning the model properties using the graphical tools provided by Visual MODFLOW. Enter the following default property values:

Kx (ft/d) **10 ↔**

Ky (ft/d) 10

(Ky is automatically set equal to Kx)

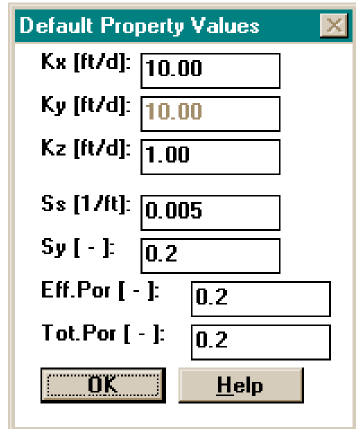
Kz (ft/d) **1.0 ↔**

Ss **0.005 ↔**

Sy **0.20 ↔**

Eff. Por. **0.20 ↔**

Tot. Por. **0.20**



☞ **[OK]**

You will then be transferred to the Properties Input screen which allows you to graphically assign hydraulic conductivity values to the model grid cells by painting individual cells, drawing polygons around multiple cells, or by stretching a window around multiple cells. For this particular example, the aquifer is being represented by a single layer, homogeneous model so you do not need to assign any heterogeneities 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 that you can assign to your model.

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

A Default Recharge window will appear (see below) 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).

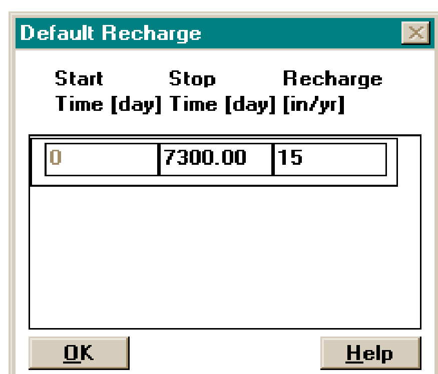
Enter the following default property values:

Start Time (days): 0.00 (default value)

Stop Time (days): **7300 ↔**

Recharge (in/yr): **15**

☞ **[OK]** (to accept these values)



You will be transferred to the Recharge Input screen which allows you to graphically assign spatially and temporally variable recharge rates. For this example, you will assume that a steady-state recharge rate of 15 in/yr 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)

A pop-up window will prompt you to '**Save property data before exiting?**'

☞ **[Yes]**

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 Line]** (from the left-hand menu bar)

Move the mouse to the top left-hand corner of the model grid and click once to anchor the line. Then move the mouse to the bottom left-hand corner of the model domain and click the RIGHT MOUSE BUTTON to close the line. The cells corresponding to the line will be shaded pink, indicating that they will be assigned a constant head boundary condition. An Assign Constant Head window will then appear prompting you to enter the required constant head information (see 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.

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 values:

Code #: **1**

☑ **Assign to appropriate layer**

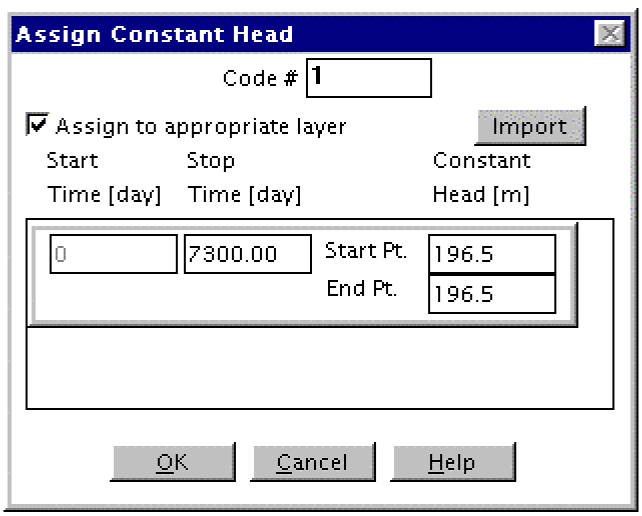
Start Time: **0.000 (default)**

Stop Time: **7300 ↔**

Start Point: **196.5 ↔**

End Point: **196.5**

☞ **[OK]** (to accept these values)



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.

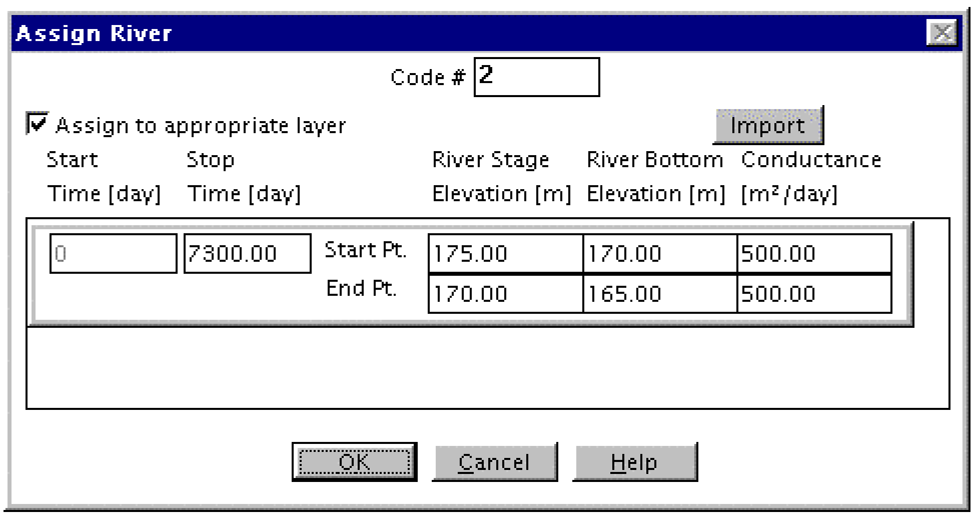
☞ **Boundaries** (from the top menu bar)

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

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

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

Move the mouse to the grid cell corresponding to northern most 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 that they will be assigned a river boundary condition (the following figure). An Assign River window will then appear prompting you to enter the required information.



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 is as follows:

$$C = \frac{K_v \times L \times W}{B}$$

where: C is the Conductance

K_v is the vertical hydraulic conductivity of the riverbed

L is the length of the river in the grid cell

W is the width of the river in the grid cell

B is the thickness of the riverbed

For the purpose of this exercise, we will calculate the Conductance as follows:

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

Enter the following values:

Code #: **2**

☑ **Assign to appropriate layer**

Start Time: **0.000 (default)**

Stop Time: **7300 ↔**

Start Point:

River Stage Elevation: **175 ↔**

River Bottom Elevation: **170 ↔**

Conductance: **500 ↔**

End Point:

River Stage Elevation: **170 ↔**

River Bottom Elevation: **165 ↔**

Conductance: **500**

☞ **[OK]** (to accept these values)

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 Area 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 available options for assigning particles within the model domain.

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

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

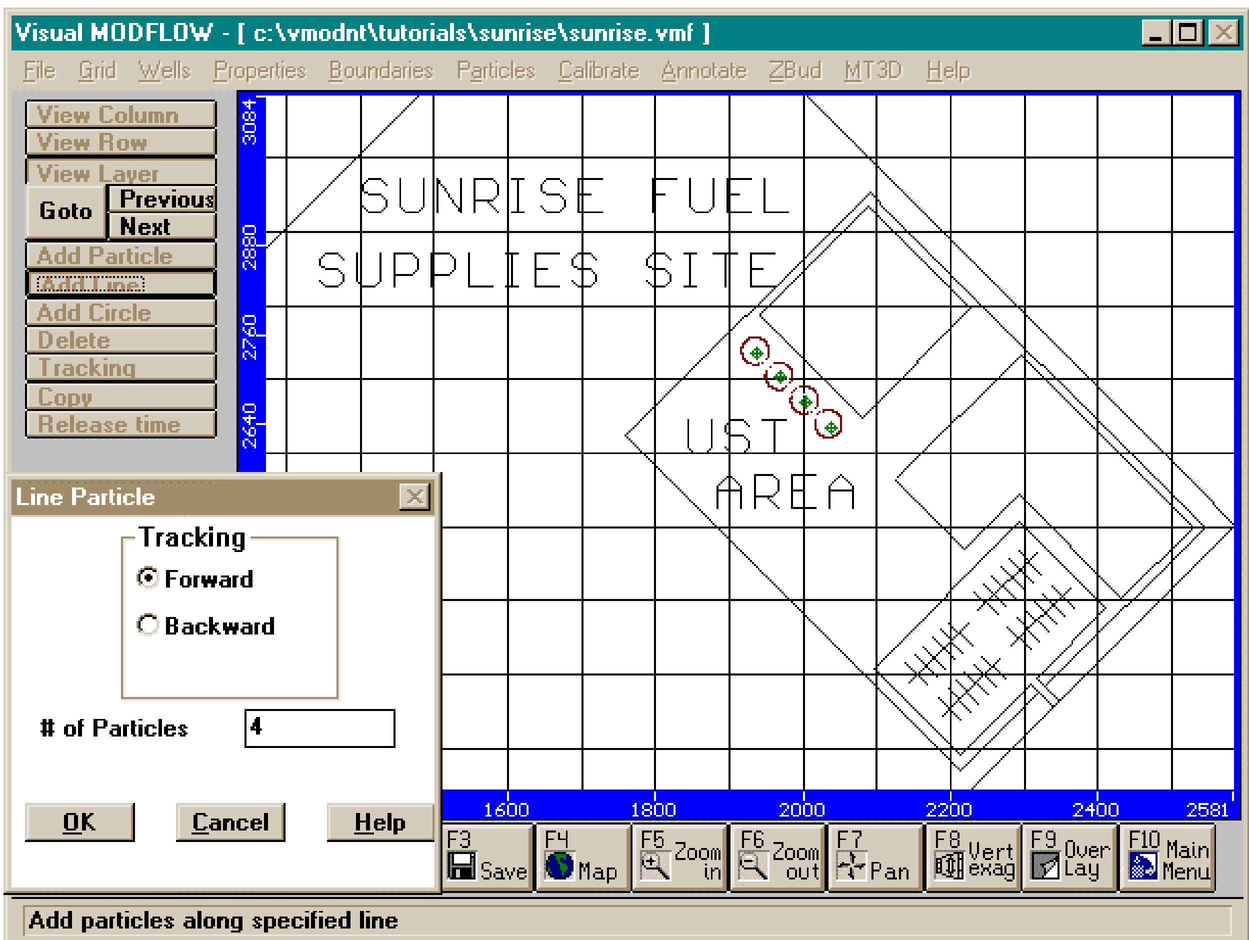
Move the mouse cursor into the model domain and click on a location to the north-west of the site to anchor the zoom window (see the figure below). Then stretch a box around the site and click again to close the zoom window. The screen display should appear similar to that shown in the following figure.

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

Move the mouse cursor to the center of the northern UST Area and click once on the northern circle to anchor the location of the particle line. Then stretch a line to the center of southern UST and click again to close the line. The particles should appear in the center of each UST as shown in the following figure. A Line Particle window will appear with default settings for the line of particles.

Change the **# of Particles** to '**4**'

☞ **[OK]** (to accept the Particle Line settings)



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 and MT3D.

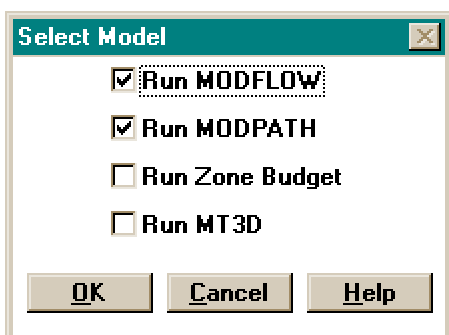
A Select Run Type window will prompt you to specify whether you will be running either a Transient or Steady State simulation. The default setting is ⦿ **Steady State.**

☞ **[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 pop-up window will appear,



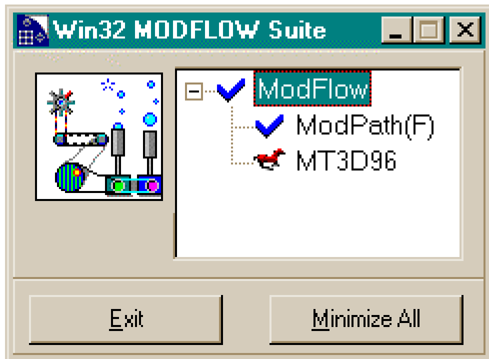
☞ ☑ **Run MODFLOW,** so a appears in the box next to it

☞ ☑ **Run MODPATH,** so a appears in the box next to it

☞ **[OK]**

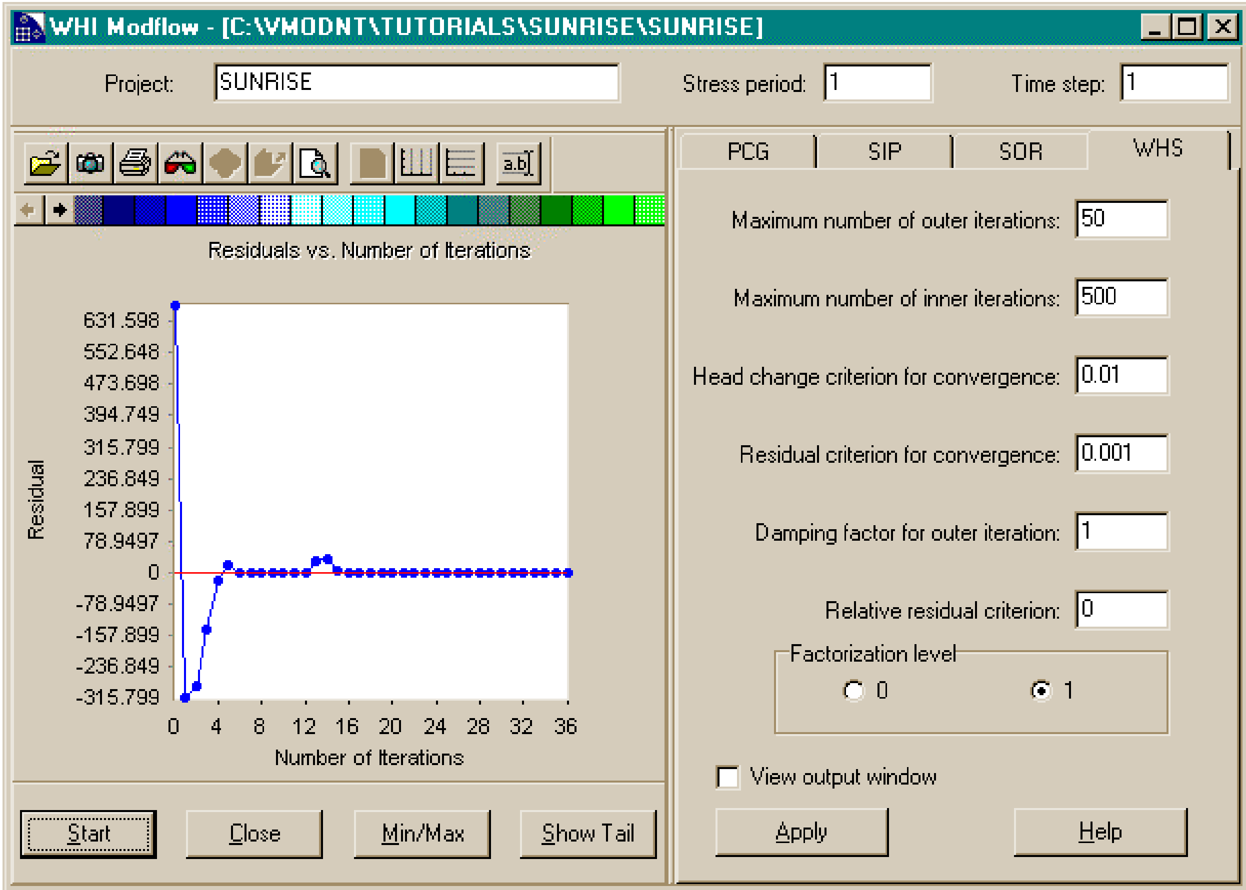
Visual MODFLOW will then create the necessary files and run the USGS MODFLOW program.

Visual MODFLOW version 2.51 or higher includes the Win32 MODFLOW Suite which contains MODFLOW, MODPATH, ZONE BUDGET and MT3D 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 solution is iterating.



When the models are executed, the Win32 MODFLOW Suite window will be displayed during the execution of each numeric model. This window keeps you up-to-date about which packages are running. A check mark indicates the numeric engine has completed running, a running horse indicates it is currently running, and a red circle indicates it is waiting to be run. Each engine will have an information window that displays simulation results and progress. These windows can be minimized by selecting **[Minimize All]** from the Win32 MODFLOW Suite window. These windows can be opened again by clicking on the specific model in the Win32 MODFLOW Suite window.

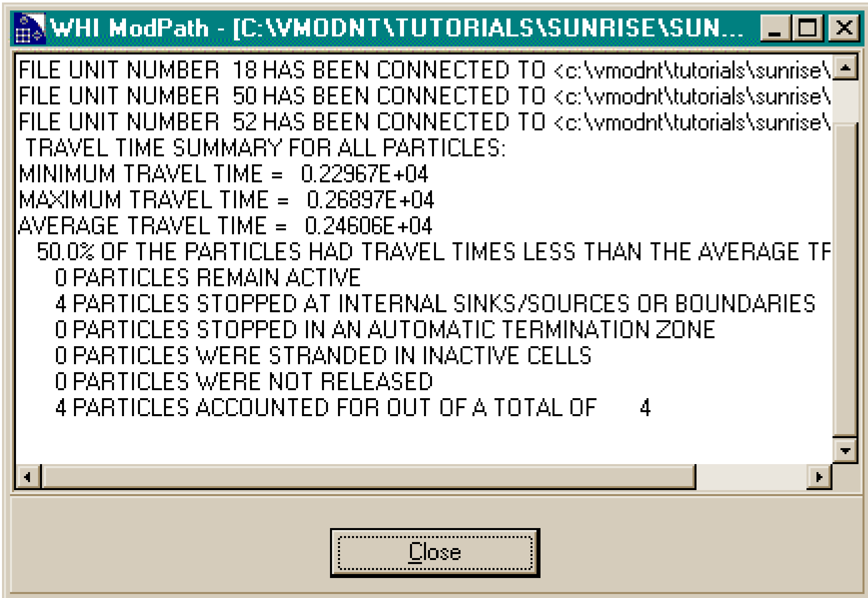
The following interactive screen is displayed for MODFLOW:



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. This plot is updated after each MODFLOW iteration. Numerical output can also be displayed by selecting ☑ **View Output Window**.

The MODPATH window looks like the following figure. This window 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), click on **Exit** in the Win32 MODFLOW Suite window, to close the Win32 MODFLOW Suite.

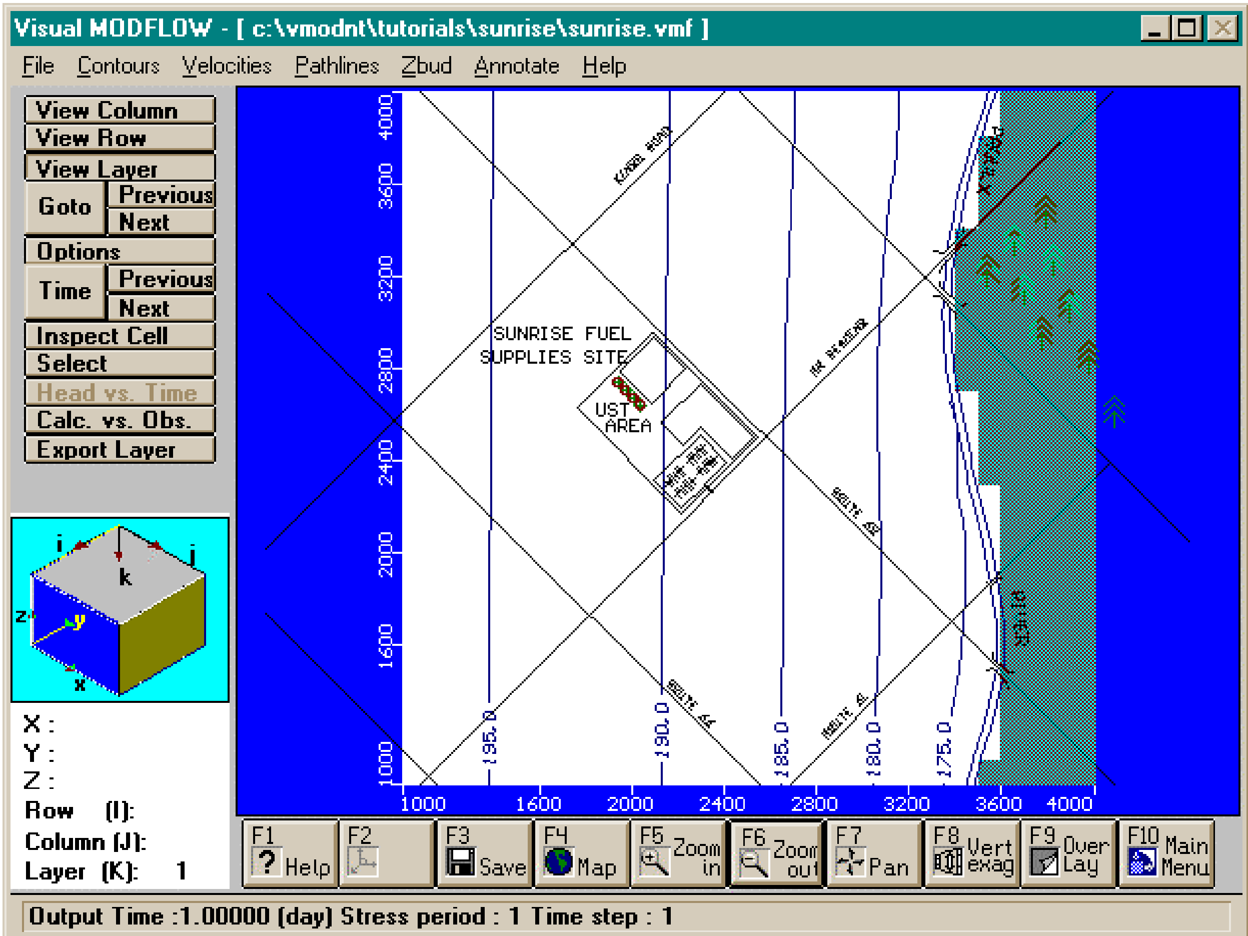
Module 3: Visualizing Model Results

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)



An aerial 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.

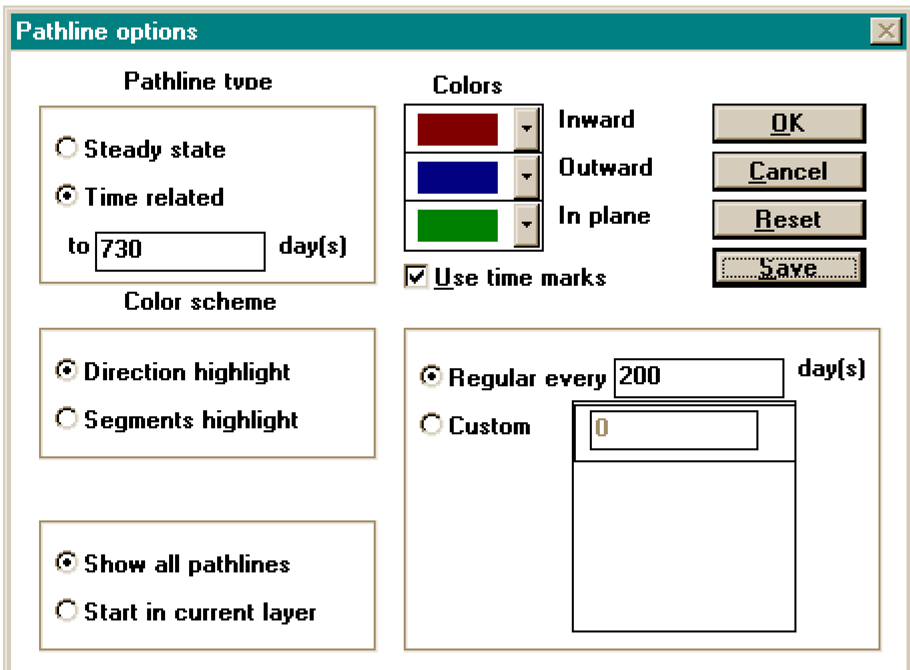
☞ **[F5 - Zoom In]** (from the bottom menu bar)

Move the mouse cursor to a location northwest of the site area and click the mouse button once to anchor the top left corner of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window.

To estimate how far the groundwater plume may have migrated from the UST Area, a conservative approach would be to assume that 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-hand menu bar)

A Pathlines Options window 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. The **Steady-state** pathlines setting will display flow pathlines for a steady state condition. The **Time-related** pathlines will display the 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 window. For this example, the default setting should say ⦿ **Regular every 200 days**.



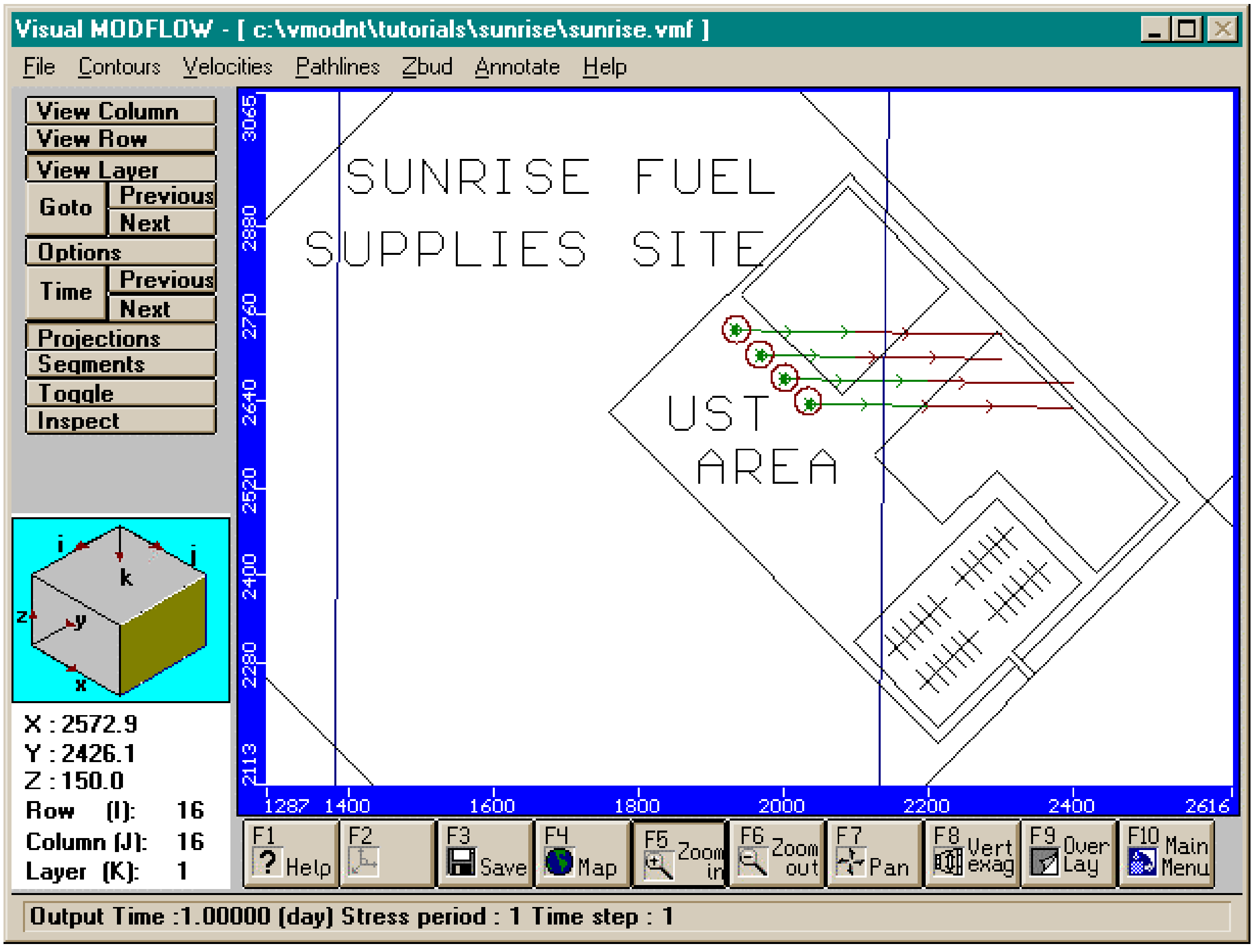
Change the settings to:

☞ ⦿ **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 travelled after two years due strictly to advective transport mechanisms. These pathlines indicate that the conservative elements of the contaminant plume have not likely migrated off-site. However, based on the pathline time markers, it is apparent that the 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 for capturing the existing plume and preventing 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)

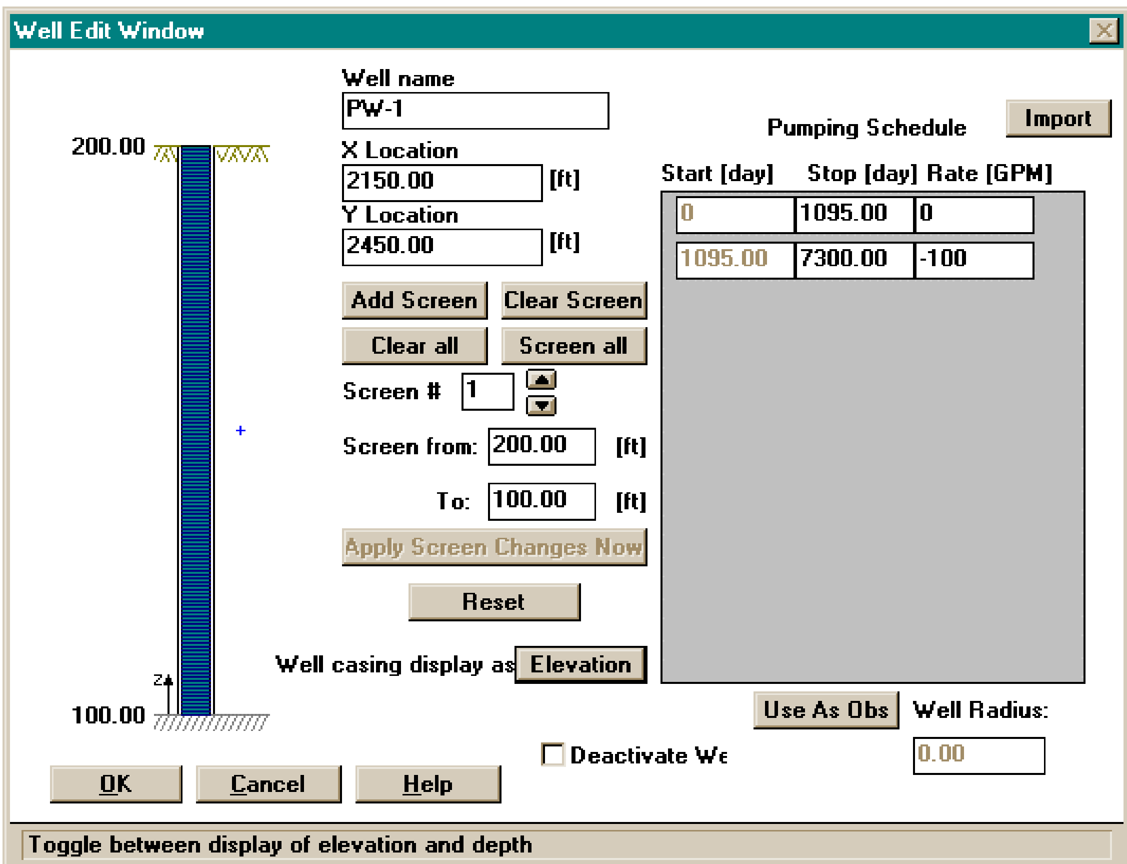
You will be transferred to the Grid Input screen by default.

☞ **Wells** (from the top menu bar)

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 Well Edit window will appear as shown below. This screen displays a representative well cross-section on the left side of the window.



Using the mouse, double-click in the box labelled **Well name:** and enter the following information:

Well name: **PW-1** ↔

X location: **2150** ↔

Y location: **2450 ↔**

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

☞ **[Add Screen]**

Move the mouse into the well bore and click the mouse near the ground surface to anchor the starting point of the well screen. A red bar will appear inside the well bore and will follow the vertical location of the mouse. Move the mouse down to the bottom of the well bore and click again to set the well interval. Alternatively, you could select the button labelled **[Screen All]**.

Next you will enter the well pumping schedule. Since the UST 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 that 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). Using Figure 14 as a guide, enter the following pumping schedule for the remediation well (note the negative pumping rate for the extraction well).

Start (days):	Stop (days):	Rate (gpm):
0.00	**1095 ↔**	**0.00 ↵**
1095	**7300 ↔**	**-100**

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

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 before exiting)

☞ **Run** (from the top menu bar)

You will be transferred to the Run Options input screen 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]**

You will then be transferred to the Run Options screen. For this run you will be interested in seeing the drawdown influence of the pumping well and influence of the pumping well on the 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.

☞ **Basic** (from the top menu bar)

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

An Initial Head window will prompt you to select the initial head estimate for the simulation. The default setting is ⦿ **Constant By Layer** which assigns a single value for each layer of the model.

☞ ⦿ **Previous Visual MODFLOW Run**

☞ **[OK]**

An Import Head From MODFLOW run window will appear prompting you to select the appropriate MODFLOW heads (.hds) file.

☞ **sunrise.hds**

☞ **[OK]**

A Select Output Time window allows you to select the output time from the .hds file. Since the first run was a steady state simulation, there will be only one time period to select.

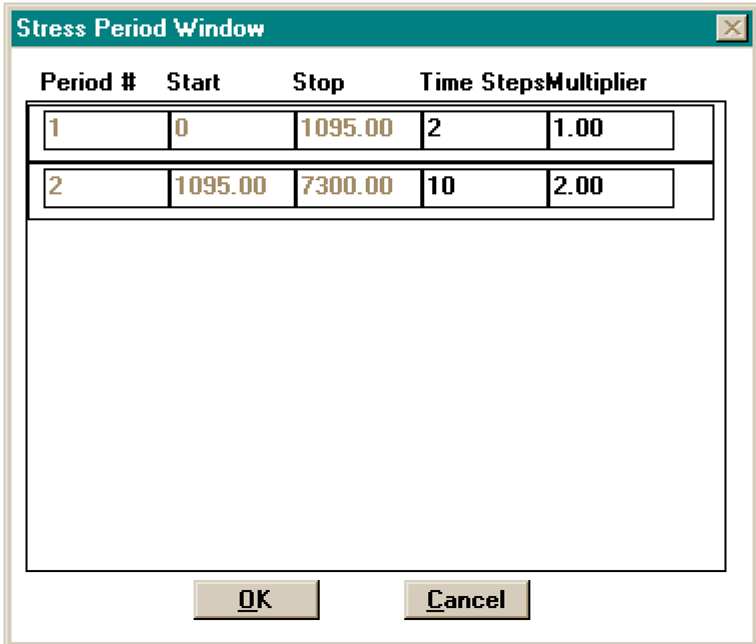
☞ **[OK]**

The other option under the **Basic** menu is **Time**. This 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.

☞ **Basic**

☞ **Time**

A Stress Period window will appear as shown below showing the available time settings and default values for the number of time steps (10) and the 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.



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 for 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, you should have a time step multiplier >1 to provide more information earlier in the stress period when the most rapid changes are occurring. Using the above figure as a reference, enter the required information in the Stress Period window.

☞ **[OK]** (to accept these values)

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).

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

☞ **OC** (from the top menu bar)

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

An Output Control window 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 the 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 Print On]** (from the bottom of the window)

☞ **[All Save On]** (from the bottom of the 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)

☞ **[OK]**

Visual MODFLOW will then begin translating the Visual MODFLOW files and will open up the Win32 MODLFOW to run the MODFLOW and MODPATH simulation (see page 16 for details). 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 then 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 a time of 547.5 days are the same as for the initial simulation. This result is for the first stress period where the well has not yet started pumping (this is 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 Equipotentials File window appears requesting you to enter a Filename for the ASCII (x, y, h) file. Enter the following:

Filename: sun_ini.asc

☞ **[OK]**

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 window will appear with a listing of the available output time steps.

Notice that 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]**

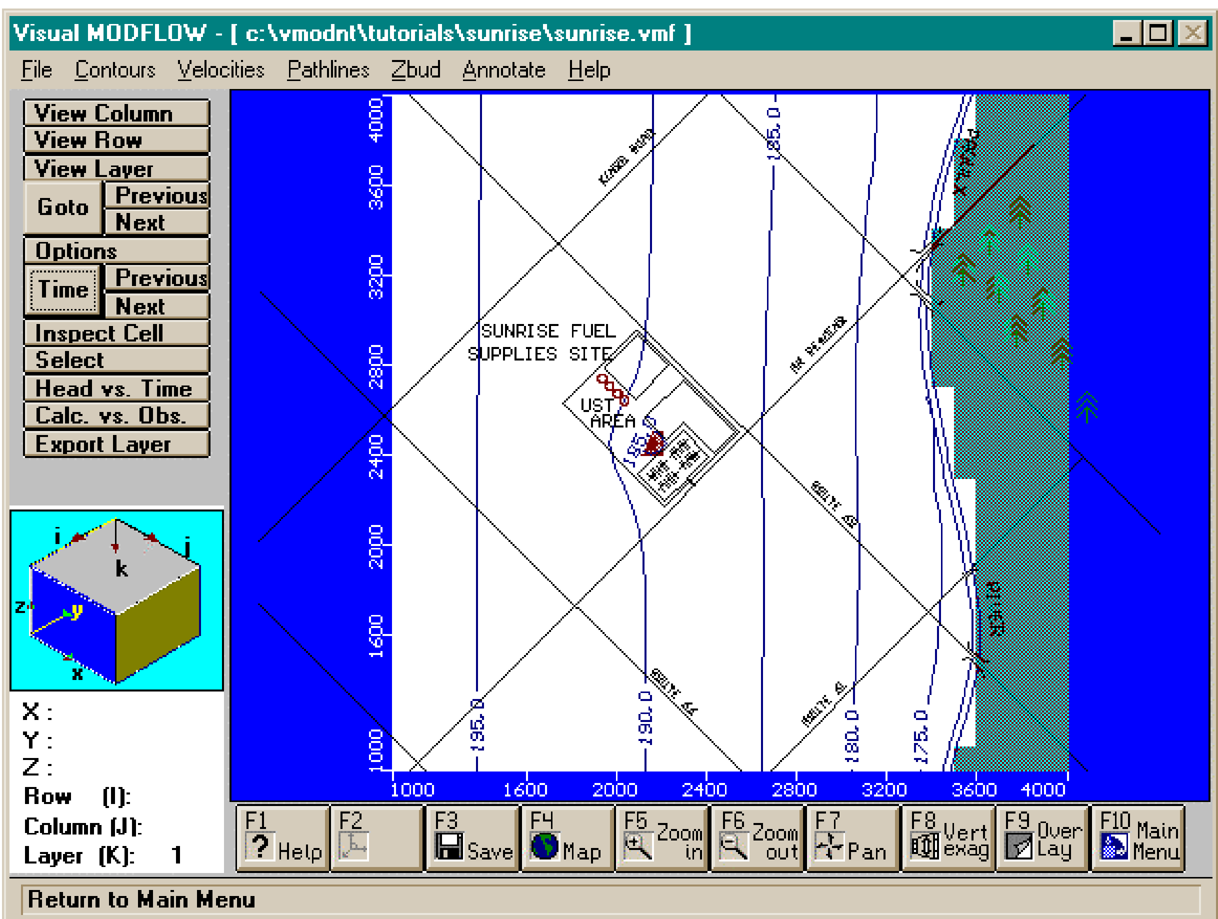
This will display the heads for a time of 1095 days, just prior to the pumping well being activated.

To display the head contours for the first time step after pumping,

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

Notice the small deformation of the head contours in the vicinity of the pumping well as shown below. Continue to step through the remaining time steps by selecting **[Next Time]** until the heads reach a steady state condition (i.e. the heads no longer change significantly).

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,

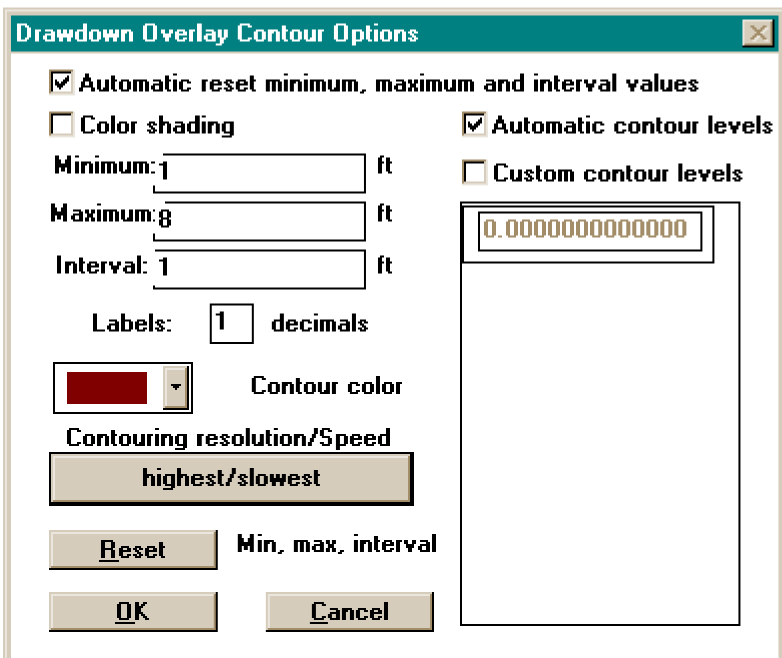
☞ **Contours** (from the top menu bar)

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

You will be transferred to the Drawdown Output screen where the head contours are replaced by the drawdown contours. Notice that 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)

A Drawdown Overlay Contour Options window will appear as shown in the following figure.



The ☑ **Automatic reset minimum, maximum and interval values** defaults to active (as indicated by ☑) which means that every time you advance to a new layer, the min., max. and interval of the head contours are recalculated.

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 labelled Minimum, Maximum, and Interval.

The ☐ **Custom contour levels** defaults to inactive (as indicated by ☐) which means that custom contours levels will not be displayed.

The ☐ **Color shading** defaults to inactive (as indicated by ☐) which means that color shading of contoured results will not be displayed.

The **Label:** box allows you to set the number of desired decimal places for each contour value.

The **Contour 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 (as indicated on the button labelled **[Highest/Slowest]**.

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.

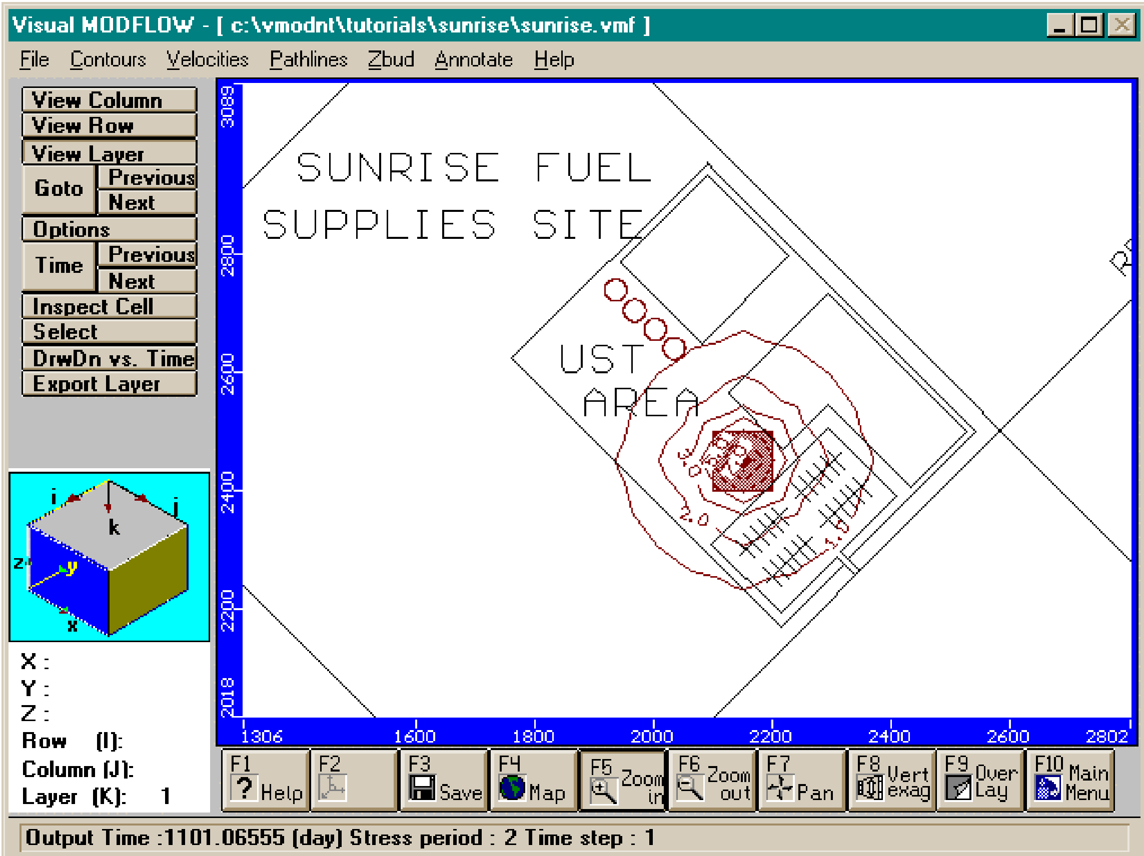
Change the minimum contour value to **1.0** and select **[OK].** The drawdown contours will be plotted for a range of 1.0 - 8.0 ft with a one-foot interval.

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]**

Move the cursor to a location northwest of the site area and click the mouse button once to anchor the starting location of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window. The screen display should look similar to that shown below

NOTE: When the mouse is pointing inside the model domain, the output time is displayed in the status bar along the bottom of the screen.



To examine the drawdown contours for the next time step,

☞ **[Next Time]** (from the bottom menu bar)

Remember that the range of contour values and the 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.

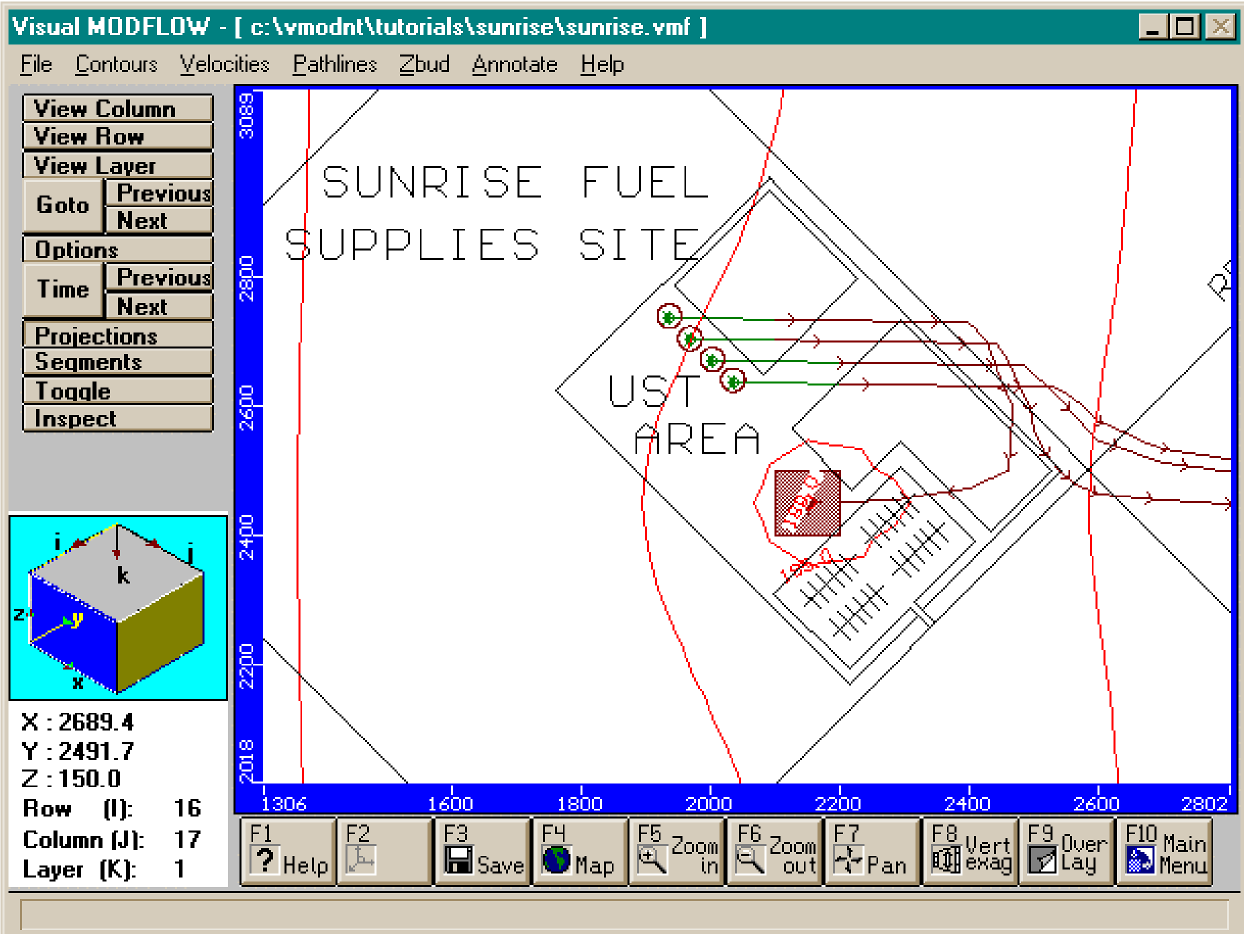
Continue to advance through each time step to watch the cone of depression spread out radially from the well.

It is interesting to note that 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.

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. First, zoom-in to the site area as shown in the following figure.

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

The transient flow pathline results will be plotted as shown below. The figure that you see on your screen may be different than that below depending on how the river was specified.



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

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

☞ **[F6 - Zoom Out]** (to display the entire model domain)

Advance to the final time step if you are not there already.

☞ **[Time]**

☞ **7300**

☞ **[OK]**

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.

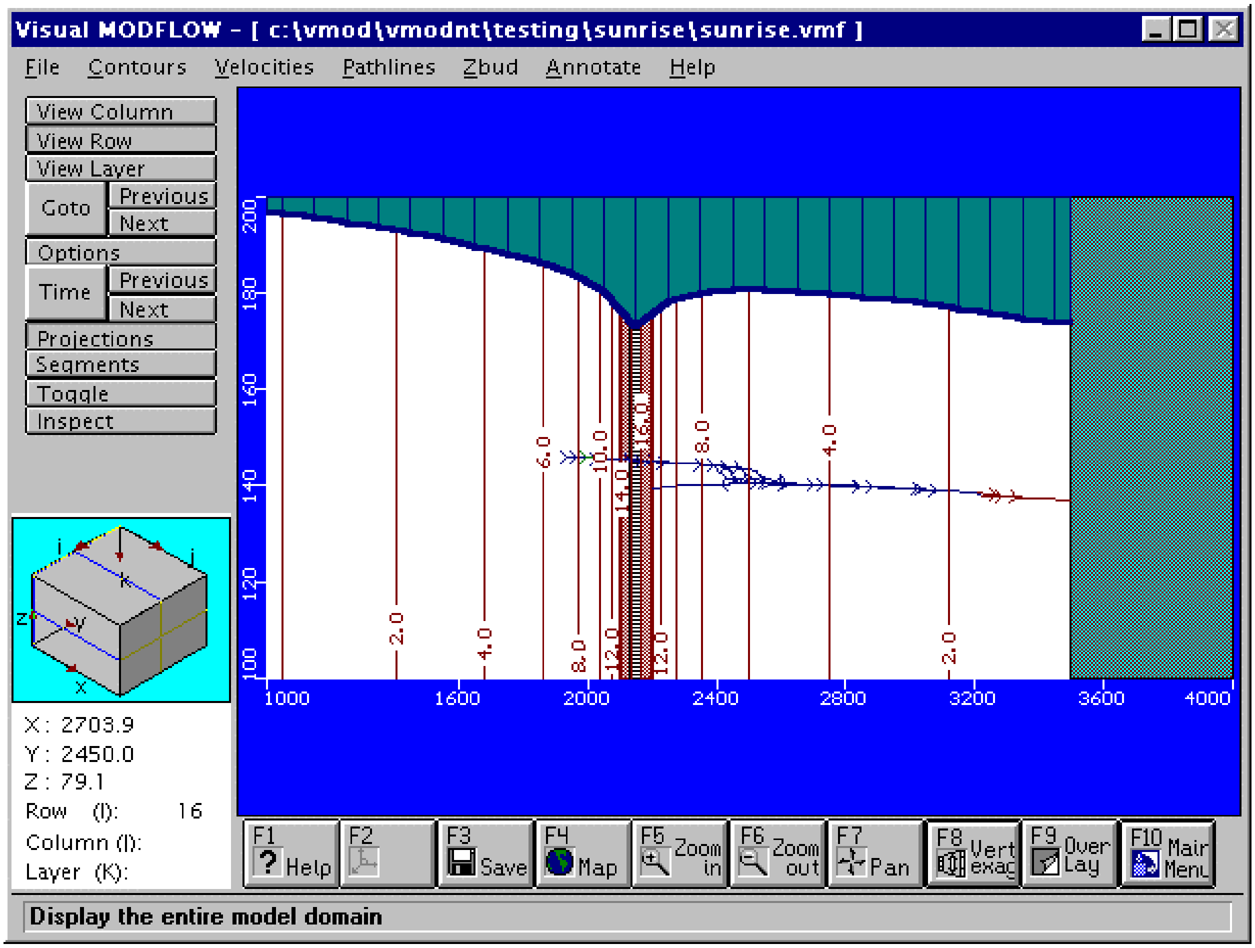
☞ **[F8 - Vert. Exag.]** (from the bottom menu bar)

A Vertical Exaggeration window will appear.

type: 15

☞ **[OK]**

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



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

☞ **[View Layer]** (from the left-hand 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.

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

A Cell Inspector window will appear listing the available output results that can be inspected on a cell-by-cell basis. Move the mouse into the model domain and the cell-by-cell information will be displayed in the Cell Inspector window. Move the cursor to the cell containing the pumping well. The head at the pumping well location should be approximately 173.25 ft.

☞ **[Close]**

Now return to the Main Menu.

☞ **[F10 - Main Menu]**

Module 5: Refining the Model Grid

The final step for this first exercise will be to examine the effects of refining the model grid in the near vicinity of the pumping well.

☞ **[Input]**

Section 1: Refining the Grid

Zoom in to the site area around the pumping well.

☞ **[F5 - Zoom In]**

Move the cursor to a location northwest of the site area and click the mouse button once to anchor the starting location of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window.

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

☞ **[Refine by 2]**

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, then click again at a Y-location of approximately 2700 ft. Then this will double the number of gridlines between these two locations.

Delete the gridline, which passes directly through the pumping well.

☞ **[Delete]** (click on the gridline that passes through the well)

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

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

CLICK THE RIGHT MOUSE BUTTON anywhere in the model domain and an Add Horizontal Line window will appear.

☞ ⦿ Add single line at

type: 2430

☞ **[OK]**

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

Then exit out of Edit Rows by

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

Now do the same thing for the columns.

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

☞ **[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.

Delete the gridline, which passes directly through the pumping well.

☞ **[Delete]** (click on the gridline that passes through the well)

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 a Add Vertical Line window will appear.

☞ ⦿ Add single line at

type: 2130

☞ **[OK]**

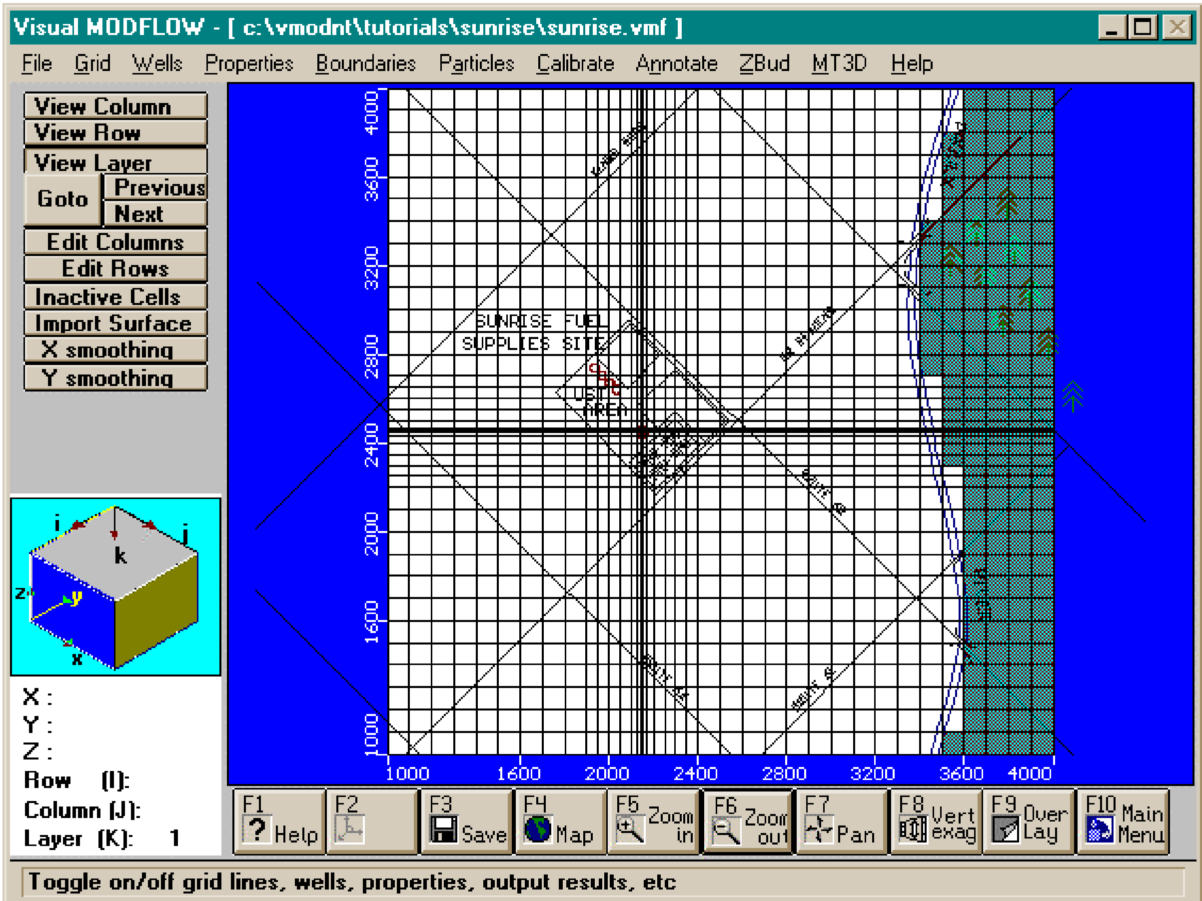
Repeat this for three more grid lines at X-locations of 2145, 2155 and 2170 ft.

Then exit out of Edit Rows by

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

☞ **[F6 - Zoom Out]**

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



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 before exiting)

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 form 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.

☞ **Basic** (from the top menu bar)

☞ **Initial Heads**

An Initial Heads window will appear listing the available options for initial head estimates.

☞ ⦿ **Import from ASCII file**

An Import heads from ASCII file window will appear listing the available ASCII (.asc) files in the C:\VMODFLOW directory.

☞ **sun_ini.asc**

☞ **[OK]** (to accept the file selection)

An Import Heads from ASCII file window appears listing the selected ASCII file, the corresponding model layer, and the # nearest data points that it will use to interpolate the data to the model grid. Enter the following:

 # Nearest: 1 (THIS IS NOT A MISPRINT! Change the value to '1')

☞ **[OK]**

Now run the model simulation for the refined model grid.

☞ **Run**

☞ **[OK]** (to run MODFLOW and MODPATH)

Visual MODFLOW will then begin translating the Visual MODFLOW files and will activate the Win 32 MODFLOW Suite (see page 16 for details). Once the MODFLOW and MODPATH calcualtions have been completed,

☞ **[Exit]** (to close the Win32 MODFLOW Suite)

Section 3: Visualizing the Model Output

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

☞ **Output** (from the top menu bar)

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

> **NOTE: The results that you obtain may be slightly different than the results shown in this exercise tutorial depending on the grid refinements which you have performed.**

Zoom in to the site 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)

Move the cursor to a location northwest of the site area and click the mouse button once to anchor the starting location of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window.

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

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

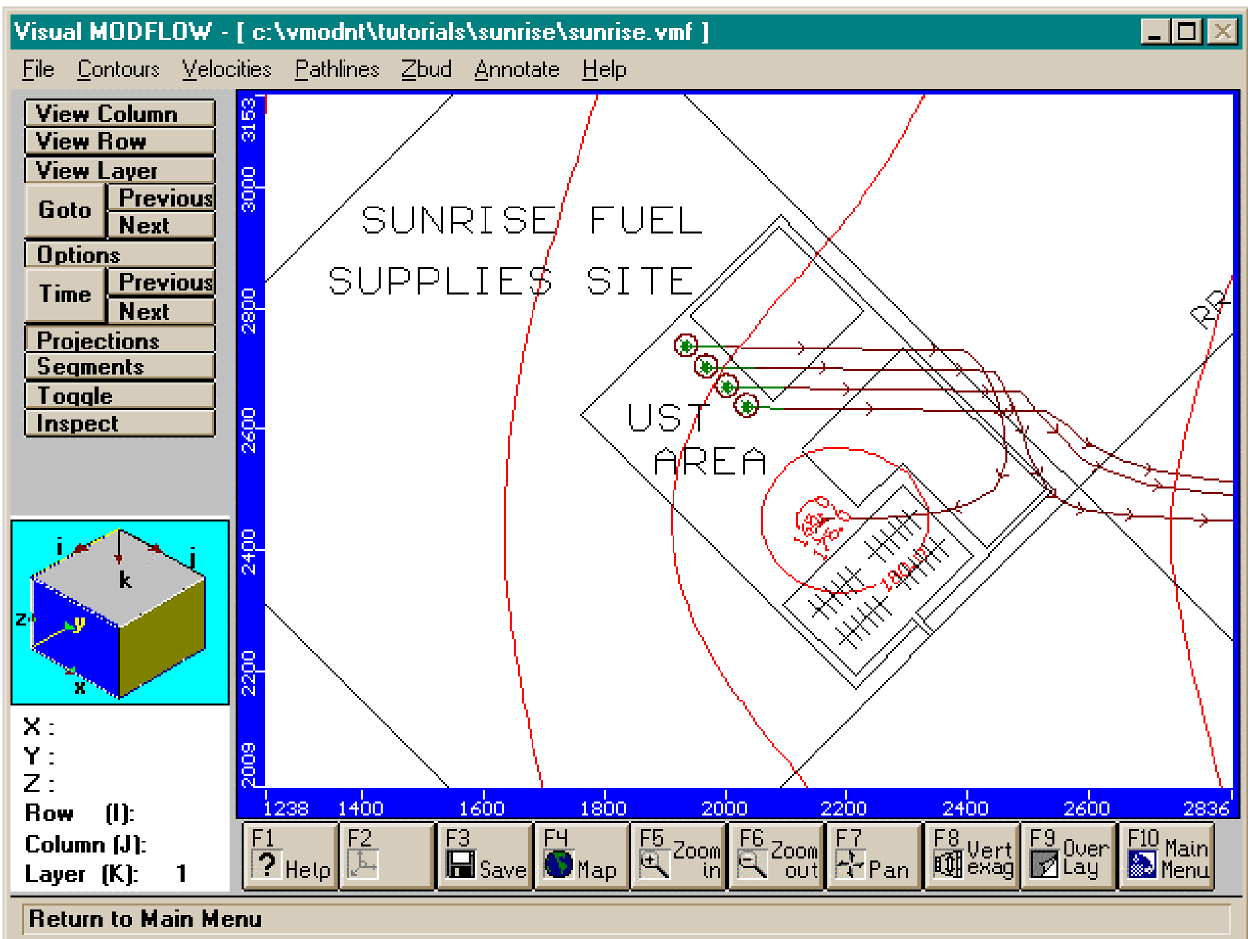
☞ **7300**

☞ **[OK]**

The displayed heads should look similar to those in the following figure, with the exception that 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.

☞ **Pathlines**

The pathlines display should appear similar to the following figure. These results indicate that the grid refinement did not have a noticeable impact on the flow pathlines.



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

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

Move the mouse cursor to the location of the well (Row 20, column 16) and check the Cell Inspector window for the calculated head in the cell. The calculated head in the cell should be approximately 161.6 ft (this may vary slightly depending on the grid spacing).

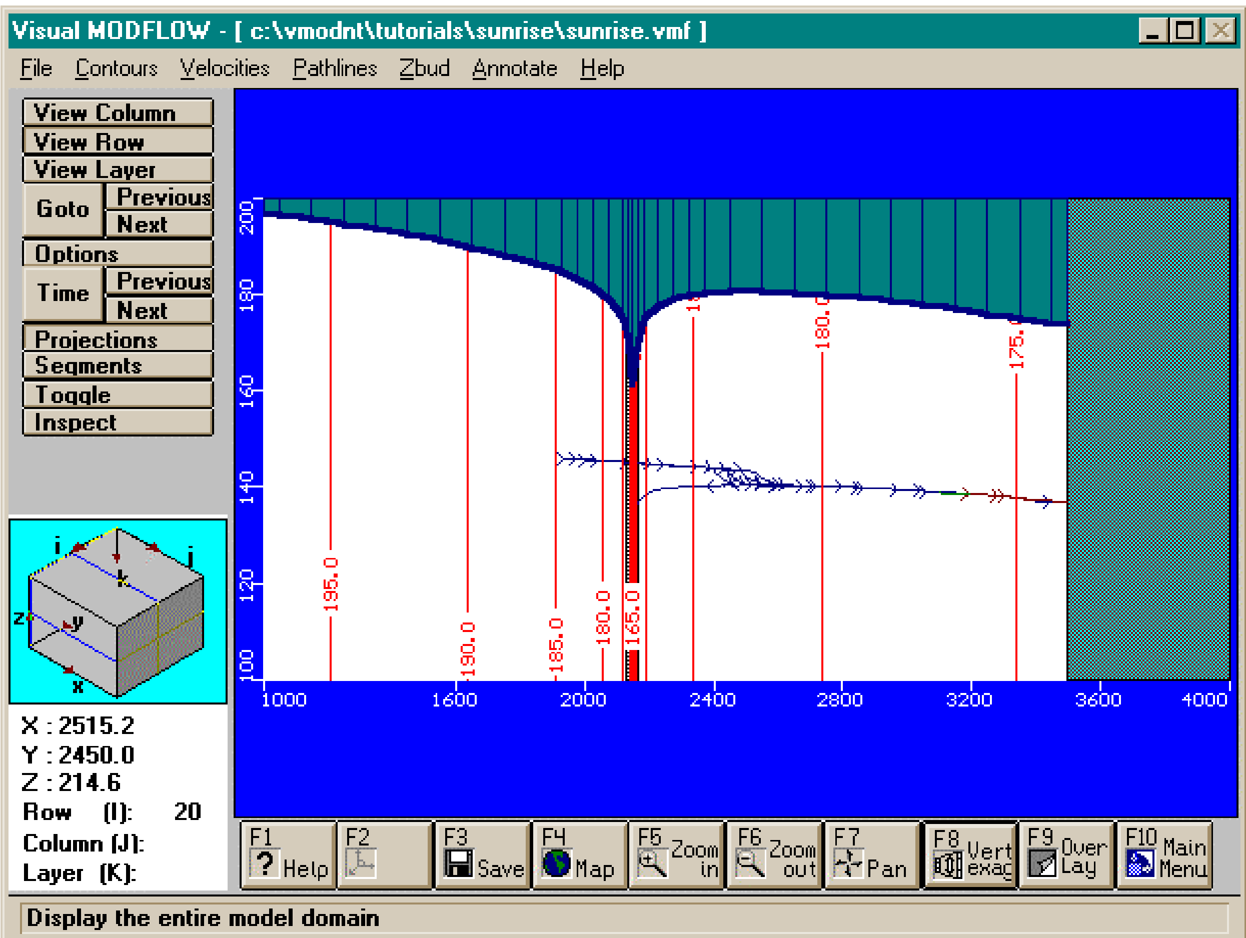
Compare this head value to the head value calculated in the first simulation before the grid refinements (173.25 ft). Clearly, the grid refinements have a significant impact on the heads calculated by the model, particularly in areas where there are steep gradients.

This is further illustrated by viewing a cross-section through the well.

☞ **[Close]** (to close the cell inspector window)

☞ **[View Row]**

Move the mouse 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.



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

☞ **[View Layer]**

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

Section 4: Increasing the Pumping Rate

In the final section of this exercise, you will increase the well pumping rate to 115 US gpm to capture and contain the off-site migration of the groundwater plume.

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

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 window will appear displaying the existing well data. Change the pumping rate in the well from -100 US gpm to -115 US gpm, by double-clicking on

-100.

- ☞ **[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 Run Type)
- ☞ **Run**
- ☞ **[OK]** (to run MODFLOW and MODPATH)

Visual MODFLOW will then begin translating the Visual MODFLOW files and will activate the Win 32 MODFLOW Suite (see page 16 for details). 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 then display a plot of the head contours for the first time step of the first stress period (time = 547.5 days). Notice that the head contours are still displayed in red.

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

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

Move the cursor to a location northwest of the site area and click the mouse button once to anchor the starting location of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window.

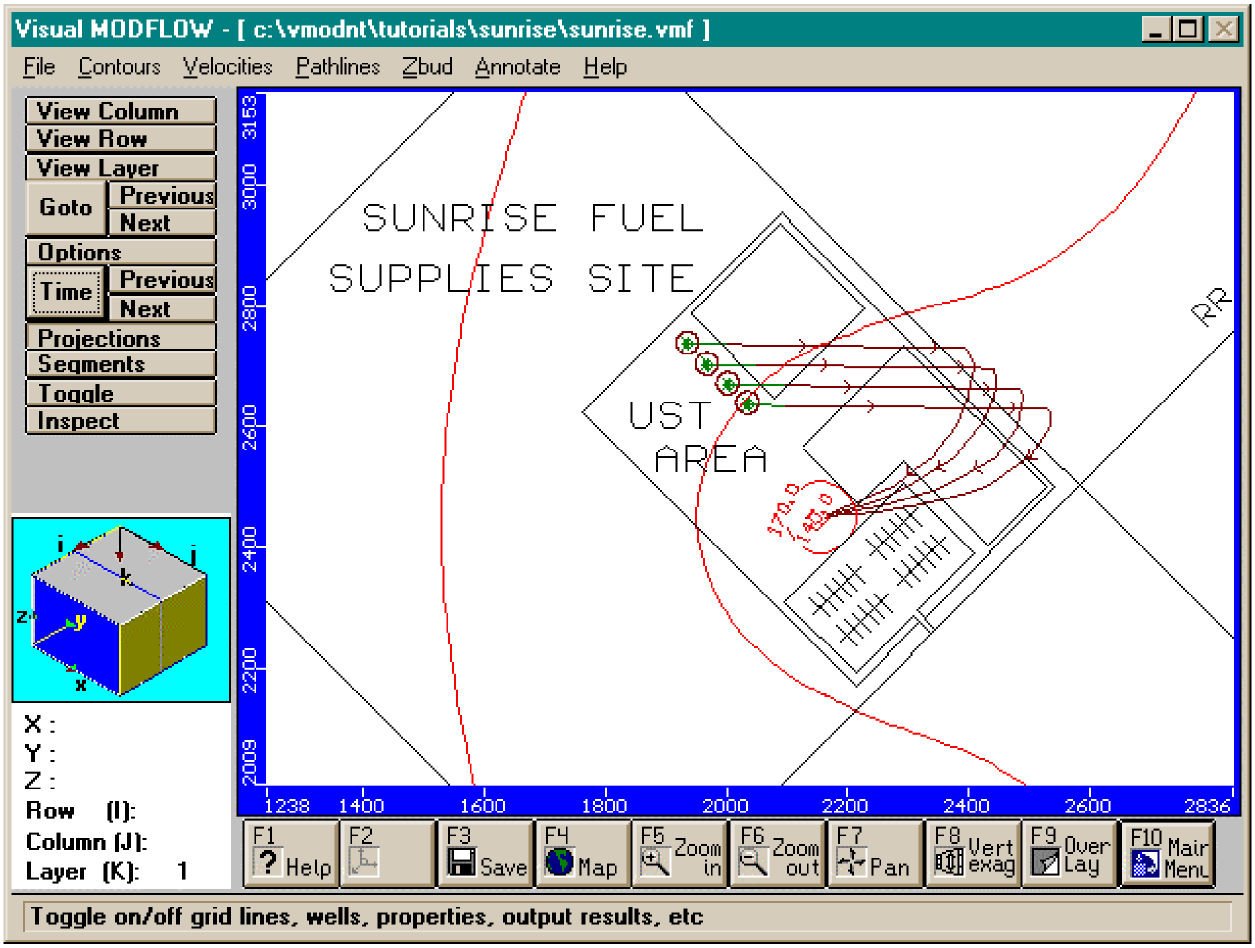
☞ **[Time]**

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

☞ [OK]

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

The results should indicate that the increased pumping rate of 115 US gpm successfully captures the particles that migrated off-site previously. This figure may be slightly different depending on how the river was defined. If you do not see a complete capture, you may have to increase the pumping rate slightly.



This concludes the Part A of the Sunrise exercise. 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.

Part B

LEARNING OBJECTIVES

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

This portion of the exercise is a continuation of the Sunrise Tutorial - Part A. If you have not completed Part A, you may not be able to successfully complete Part B.

In this exercise you will not have to actually calibrate the model, since most of the model parameters have been provided for you in Part A. However, this laboratory will guide you through the steps of modifying the various input properties to examine the influence each of them has on the simulation results.

If you are still in Visual MODFLOW and have Part A displayed, rename your data set to sunrs_b.vmf. If you are not in Visual MODFLOW, load up Part A (sunrise.vmf) and then rename your data set to sunrs_b.vmf.

☞ [**F10-Main Menu]**

☞ **File** (from the top menu bar)

☞ **Save As**

type: **sunrs_b**

☞ **[OK]**

This will copy all of the **sunrise** model files to **sunrs_b** files. In this exercise you will be changing many of the model input parameters to examine the influence that these changes have on the calibrated results. The first section of this exercise will be used to demonstrate the non-uniqueness of a model that is only calibrated to steady-state, non-pumping conditions.

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.

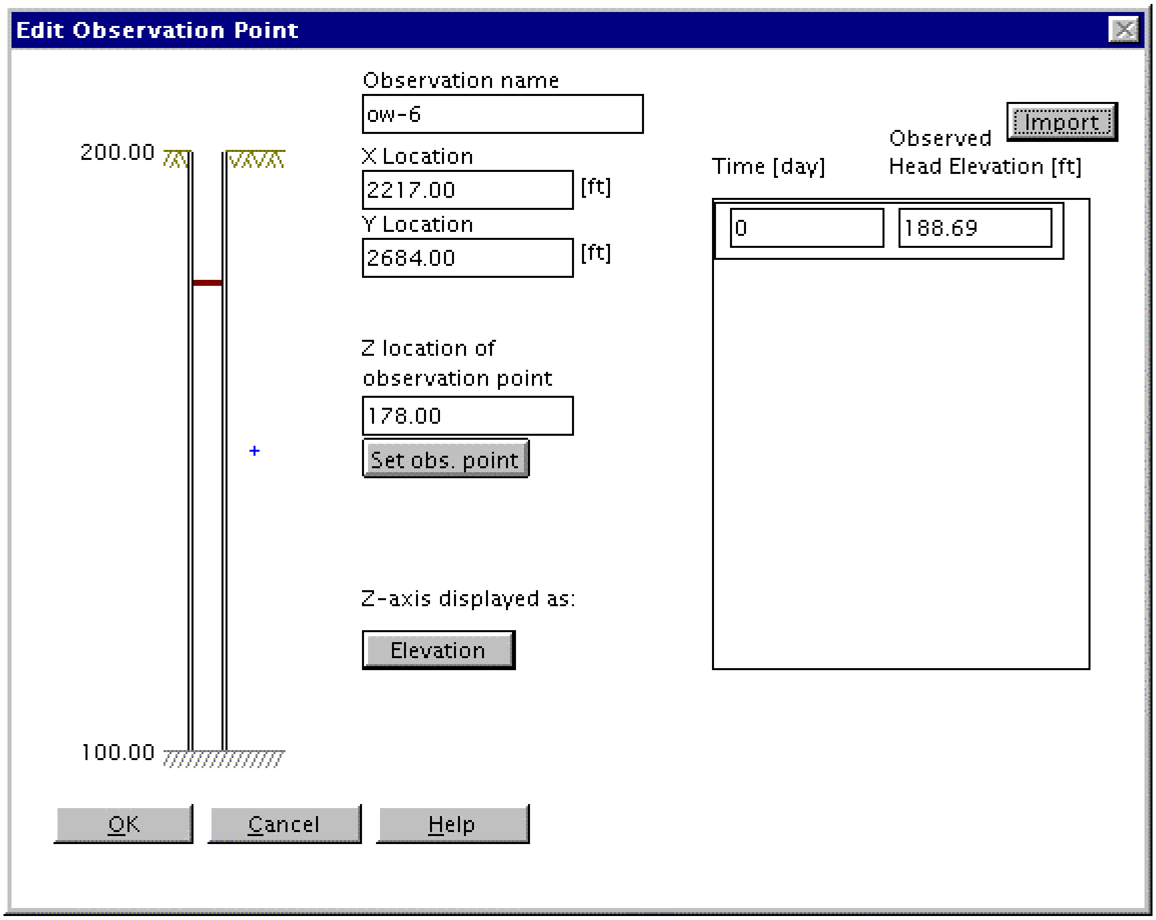
☞ **Input** (from the top menu bar)

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

You will be transferred to the Calibration 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-hand menu bar)

Move the cursor into the model domain and click the left mouse button to add an observation well at any location. An Edit Observation Point window will appear, as shown in the figure below, which will allow you to enter the observation well information.



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

Well Name: **OW-6** ↔

X-location: **2217** ↔

Y-location: **2684** ↔

Z location **178** ↔

Time: **0**

Observed Head: **188.69**

☞ **[OK]** (to accept these values)

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

Since there are 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 very similar to Table 1 (Well-ID, X-Loc., Y-Loc., Z-Loc., Time, Head)

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

An Import Observations window will appear on the screen prompting you to select the .txt file containing the observation well information.

☞ **'sun_obs1.txt'**

☞ **[OK]**

All of the observation well data will be imported into the model and the observation well locations will be displayed on the screen (as indicated by green well symbols).

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.

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

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	188.69
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

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

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

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

☞ **Run** (from the top menu bar)

☞ **⦿ Steady State**

☞ **[OK]**

At this point, you may be wondering whether the pumping well that you assigned in Part A 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 115 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)

☞ **[OK]** (to execute MODFLOW and MODPATH)

Visual MODFLOW will then translate the input files from Visual MODFLOW format to create standard USGS MODFLOW and MODPATH input data files. The program will then activate the Win32 MODFLOW Suite and run MODFLOW and MODPATH. When the MODFLOW and MODPATH calculations have been completed press the **Exit** button to close the Win32 MODFLOW Suite.

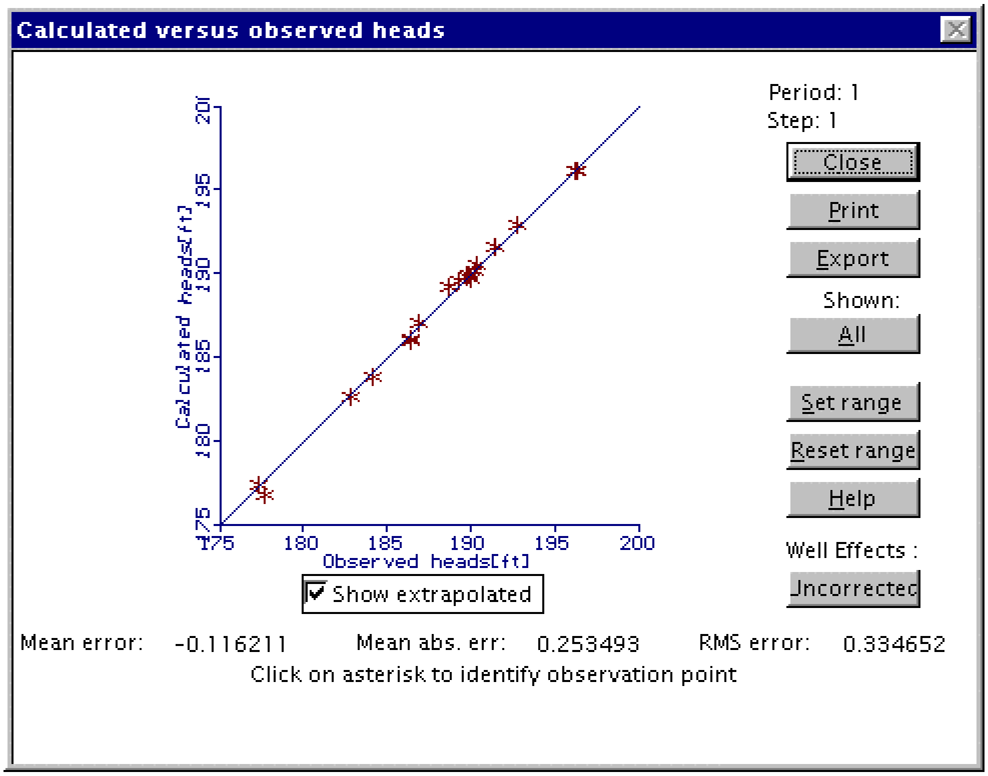
Section 3: Visualizing the Model Results

☞ **Output** (from the top menu bar)

A contour plot of the steady-state heads will be displayed. To check the calibration of the model you must compare the calculated heads against the observed heads.

☞ **[Calc vs Obs]** (from the left-hand menu bar)

A Calculated Heads vs. Observed Heads graph will be displayed as shown in the following figure with calibration statistics provided below the graph. The results for your model may be slightly different depending on how the river was specified.



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 red asterisks represent the observation 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.

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 change the K values and Recharge by an order of magnitude and still get the same results.

☞	**[Close]**	(to close the heads calibration window)
☞	**[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-hand menu bar)

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

Kx (ft/day) **1** ↔

Ky (ft/day) **1** ↔

Kz (ft/day) **0.1** ↔

☞ **[OK]**

The hydraulic conductivity value for the aquifer has just been reduced by an order of magnitude. Intuitively, this will raise the heads in the aquifer since the water entering the system through recharge will encounter more resistance to flow before it discharges to the river. Therefore, to offset the increased water table elevation, you can reduce the amount of recharge.

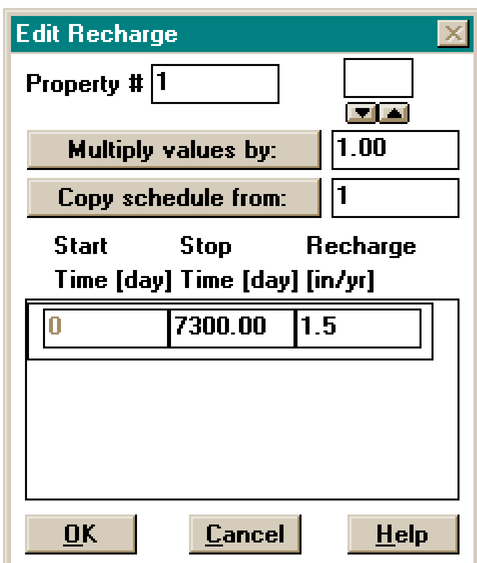
☞ **Boundaries** (from the top menu bar)

☞ **Recharge**

You will then be transferred to the Recharge Input screen.

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

An Edit Recharge window containing the recharge input values will appear as shown in the figure below.



Click in the box labelled **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)

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

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

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

☞ **Run** (from the Main Menu)

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

☞ **Run** (from the top menu bar)

☞ **[OK]** (to execute MODFLOW and MODPATH)

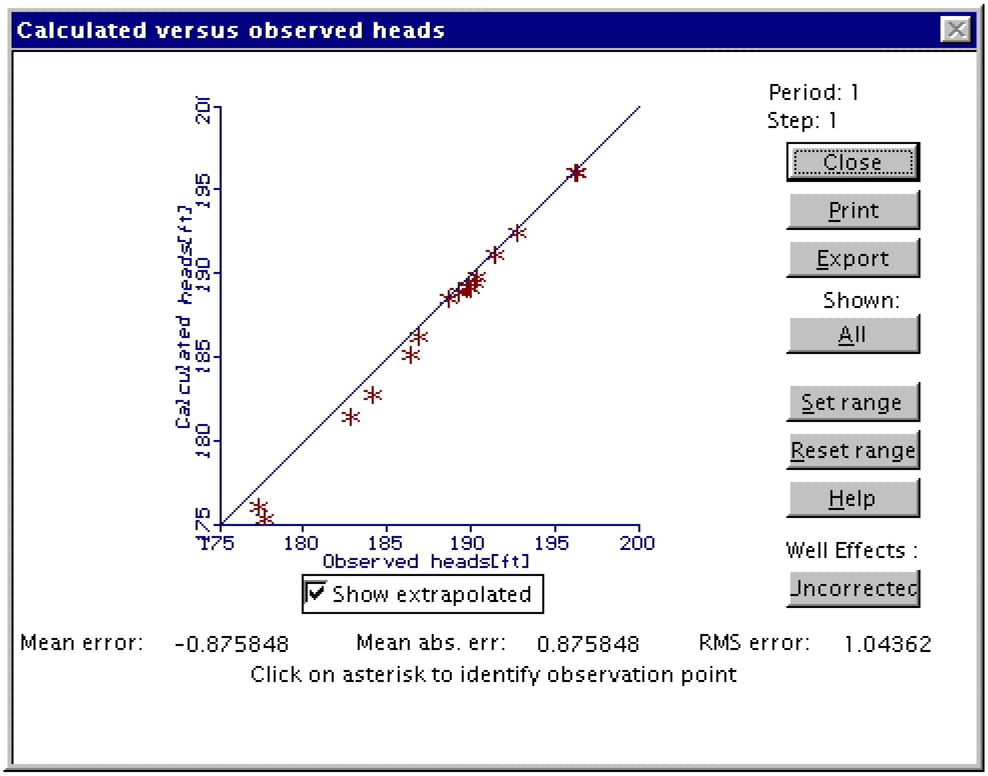
Visual MODFLOW will translate the input files from Visual MODFLOW format to create standard USGS MODFLOW and MODPATH input data files. The program will then activate the Win32 MODFLOW Suite to execute the MODFLOW and MODPATH. Once this is completed, press **Exit** to close the Win32 MODFLOW Suite and return to the Visual MODFLOW Main Menu.

☞ **Output**

A contour plot of the steady-state heads will be displayed. To check the calibration of the model you must compare the calculated heads against the observed heads.

☞ **[Calc vs Obs]** (from the left-hand menu bar)

A Calculated Heads vs. Observed Heads graph will be displayed. 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).



☞ **[Close]**

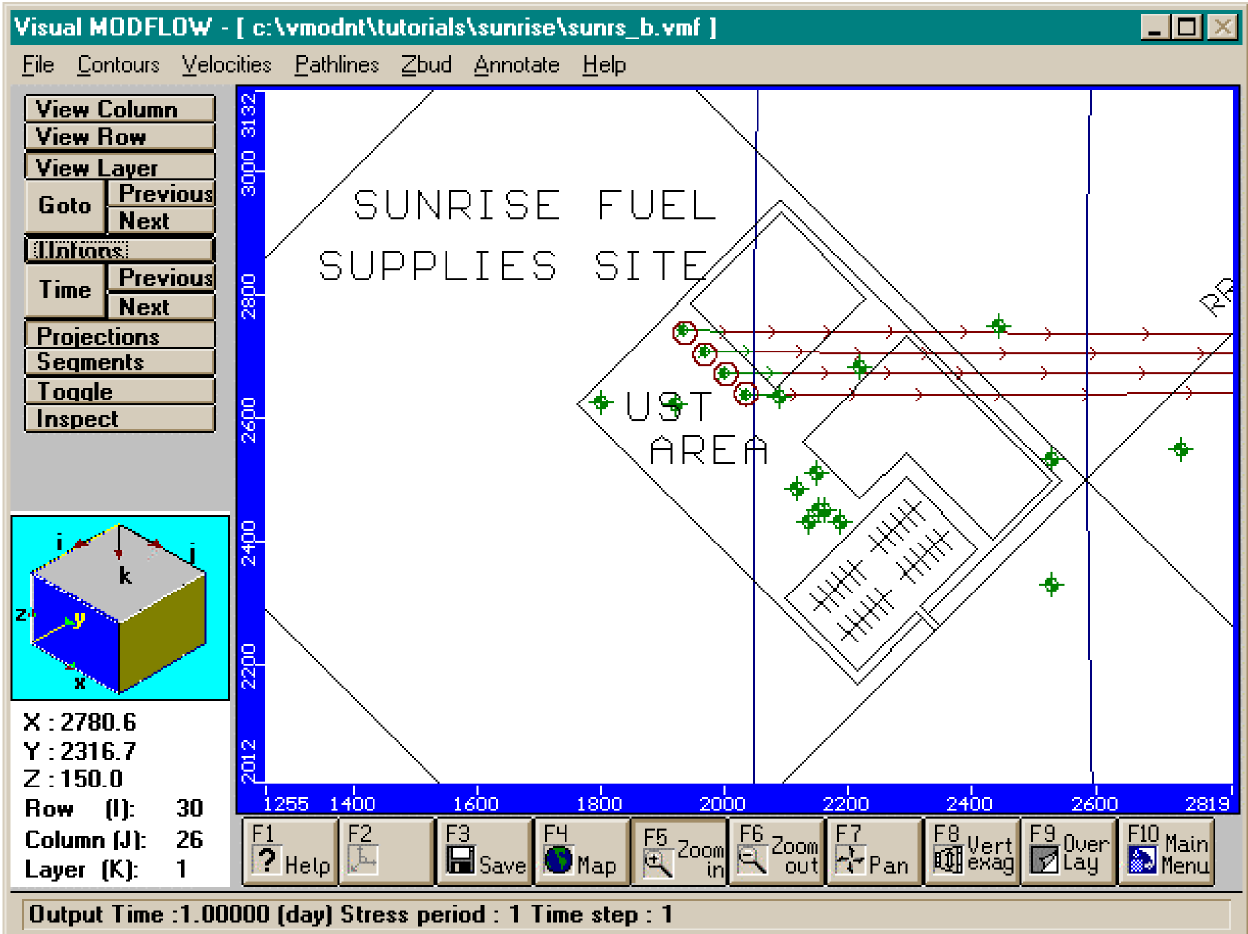
While the changes to the hydraulic conductivity and recharge did not significantly affect the calculated heads, it does have a strong influence on pathline time markers and time-related capture zones.

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

Now zoom in to the area around the Sunrise Fuel Supplies site.

☞ **[F5 - Zoom In]**

Move the mouse to a location northwest of the site area and click the mouse button once to anchor the top left corner of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window. The new screen display should resemble the following figure.



To determine the time interval of the pathline time markers,

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

A Pathline options window will appear showing the default settings for the pathline display options. Refer to the box in the bottom right corner labelled ⦿ **Regular every '2000' 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 with the previous simulation (about 10 times longer!).

The travel times of the pathlines are also very sensitive to the effective porosity of the soil material(s). For the purpose of calculating the particle travel times, the most important property is the effective porosity which is defined as the fraction of the total porosity through which flow actually occurs (i.e. neglecting dead-end pores). It is a common mistake for modellers 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 that the proposed pumping well was originally an industrial water supply well which has been operating continuously for the past 5 years at an average pumping rate of 35 gpm. 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:

Kx = 10 ft/day

Ky = 10 ft/day

Kz = 1 ft/day

Rch = 15 in/yr

☞ **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-hand menu bar)

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

Kx (ft/day) **10** ↔

Ky (ft/day) **10** ↔

Kz (ft/day) **1.0** ↔

☞ **[OK]**

Now change the recharge values back to 15 inches/year.

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

☞ **Recharge**

You will then be transferred to the Recharge Input screen.

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

Click in the box labelled **Recharge [in/yr]** and enter the following information.

Recharge: **15**

☞ **[OK]**

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

☞ **Wells** (from the top menu bar)

☞ **[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]**

Move the mouse to a location northwest of the site area and click the mouse button once to anchor the top left corner of the zoom window. Then stretch a window across the site area to a location southeast of the site area and click the mouse button again to close the zoom window.

☞ **[Edit Well]**

Move the cursor over top of the existing pumping well (Row 20, Column 16) and press the left mouse button. An Edit Well window will appear with the existing well information for pumping well PW-1 (from Part A).

Click the mouse in the second line of the pumping schedule and press the **<Delete>** key to delete the second stress period in the pumping schedule. Change the remaining well information as follows:

Stop Time (days): **7300** ↔

Pumping Rate (US gpm): **-35**

Note that 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.

Directly beneath the Pumping schedule is a button labelled **[Use As Obs]** which allows you to use the pumping well location as an observation well to monitor the head in the pumping well.

☞ **[Use As Obs]**

An Observation Point Edit window will appear allowing you to enter the applicable information for the observation well. Click the mouse in the box labelled **'Observation Name'** and enter the following information:

Observation Name: OW-PW

Observed Head: 178.34

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

This will return you to the Edit Well window.

Directly below the **'Use As Obs'** button is a box labelled **'Well Radius'**. This box allows you to enter the radius of the pumping well, which is then used to calculate the 'corrected' head in the well based on the size of the grid cell.

Click in the **'Well Radius'** box and enter a value of **'0.35'**.

☞ **[OK]**

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

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

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

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

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

☞ **[OK]**

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

Table 2: Observed Heads - Steady State, Pumping Conditions				
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	182.3
OW-3	2134	2433	182	183.4
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	196.2
OW-13	1138	2150	191	196.4
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	173	175.19

Now run the model simulation.

☞ **[F10 - Main Menu]**

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

☞ **Run** (from the top menu bar)

☞ **[OK]** (to accept steady state)

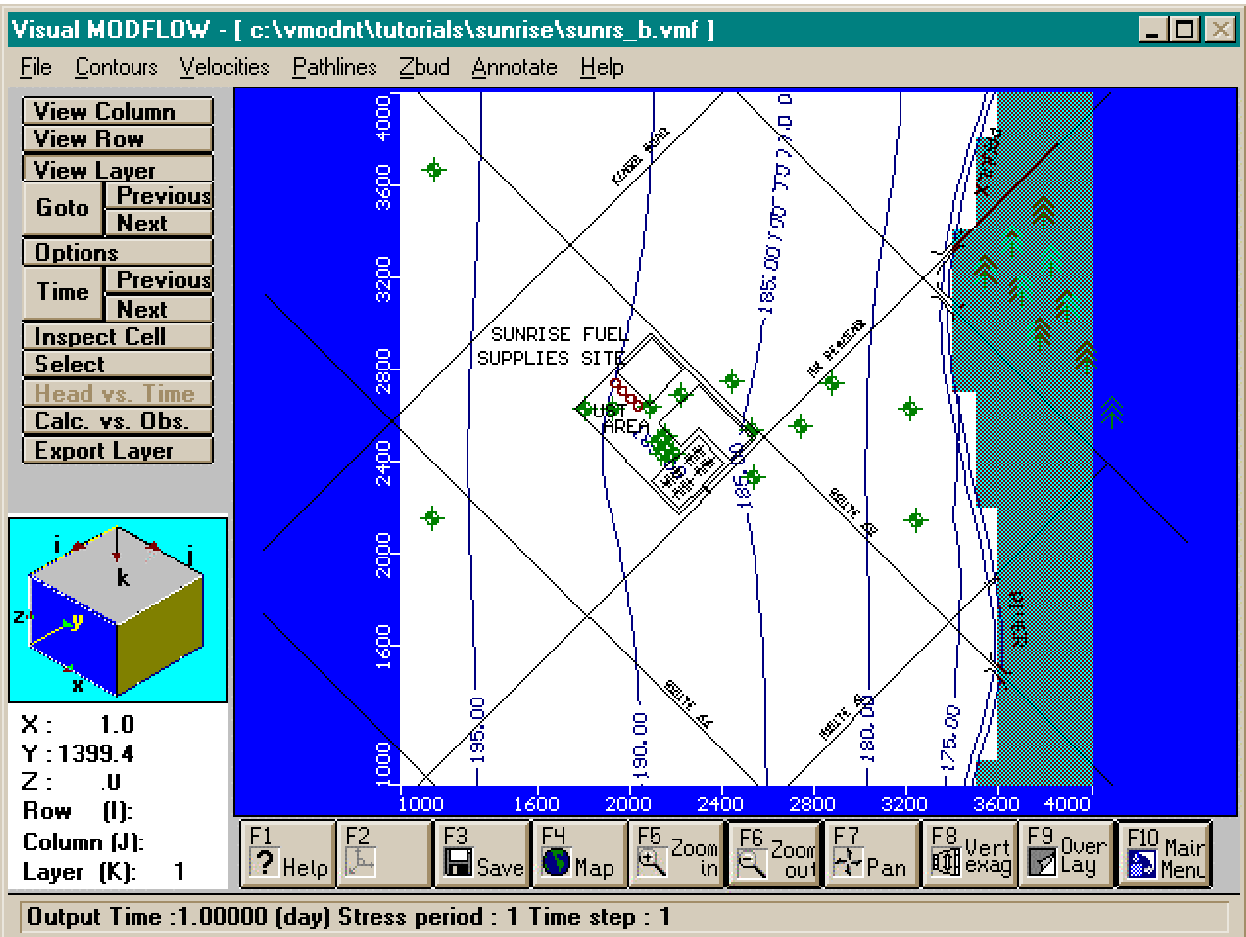
☞ **Run** (from the top menu bar)

☞ **[OK]** (to execute MODFLOW and MODPATH)

Visual MODFLOW will translate the input files from Visual MODFLOW format to create standard USGS MODFLOW and MODPATH input data files. The program will then activate the Win32 MODFLOW Suite to execute the 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)

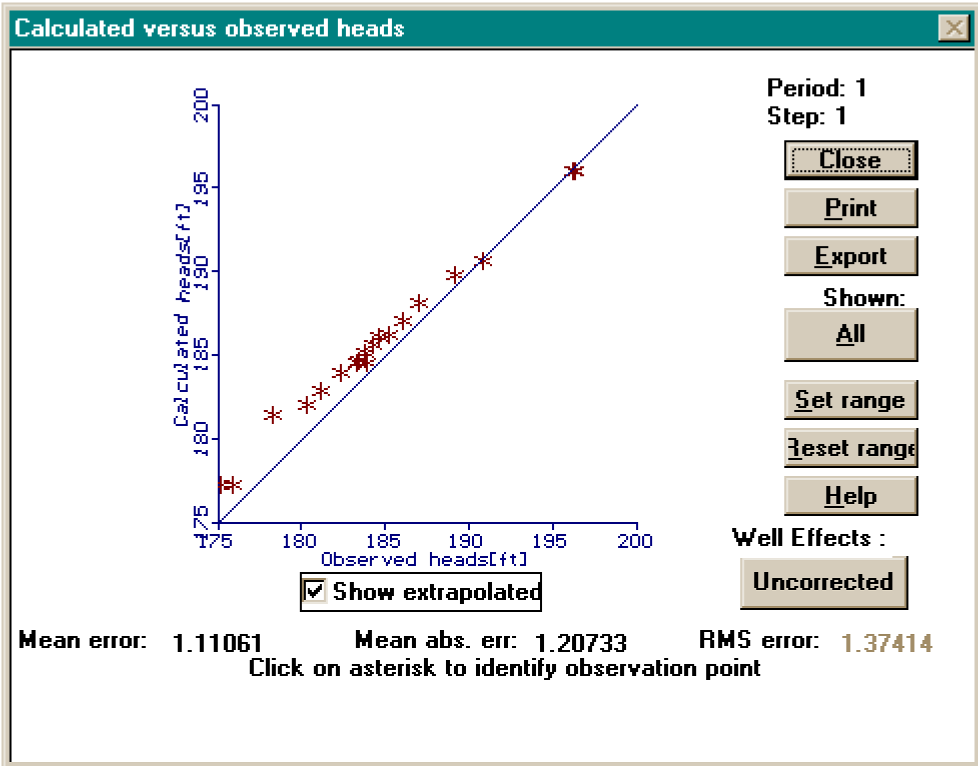
A contour plot of the steady state heads will be displayed as shown in the following figure.



To check the calibration of the model you must compare the calculated heads against the observed heads.

☞ **[Calc vs Obs]** (from the left-hand menu bar)

A Calculated Heads vs. Observed Heads graph will be displayed which shows an exceptionally good fit between the observed and calculated heads (see the following figure). **NOTE: The results that you obtain may be slightly different than the results shown below depending on how you refined the model grid and defined the river in Part A.**



The lone outlying data point occurs at a calculated head of approximately 180 ft. For this data point the model has over-estimated the head at the observation point. This observation point can be identified by using the mouse to click on the red asterisk. The asterisk will turn blue and the observation point information will be displayed below the calibration statistics.

This observation point is OW-PW, which monitors the head in the pumping well. This type of discrepancy is common when you are using MODFLOW to predict the head in a cell which acts as a pumping well because MODFLOW does not consider well effects when calculating the head in that cell. The actual head in the pumping well is usually significantly less than the head that MODFLOW calculates for the cell. Part of the reason for this is the cell dimensions are usually much larger than the actual size of the pumping well. However, Visual MODFLOW allows you to correct for well effects within the cell by considering the size of the cell used to simulate the well and the actual radius of the pumping well.

There is a button near the bottom left-hand corner of the pop-up window which is labelled **[Uncorrected]**, which indicates that the calculated heads displayed in the graph have not been corrected for well effects. Click the left mouse button on the **[Uncorrected]** button to toggle the it to read **[Corrected]**. Notice that the calculated head for OW-PW has been corrected for the well effects and now provides a better fit to the observed data.

☞ **[Close]** (to return to the Heads 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** (from the top menu bar)

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

☞ **Conductivity**

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

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

Kx (ft/day) **5** ↔

Kx (ft/day) 5 ↔

Kz (ft/day) **0.5** ↔

☞ **[OK]** (to accept these K value changes)

Intuitively, a reduced K will create a larger water table drawdown near the well as steeper hydraulic gradients are required to get the same amount of water into the well.

Since reducing K will lower the water table, then the calculated heads will be lower than the observed heads, and the model will not have a good fit between the observed and calculated data. Therefore, you will need to bring the water table back up to achieve a good fit. Since reducing the recharge to the system (as you did for the steady-state, non-pumping condition) will cause the water table to drop even further, you must increase the recharge to the system for this particular situation.

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

☞ **Recharge**

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

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

Recharge [in/yr]: **20**

☞ **[OK]**

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

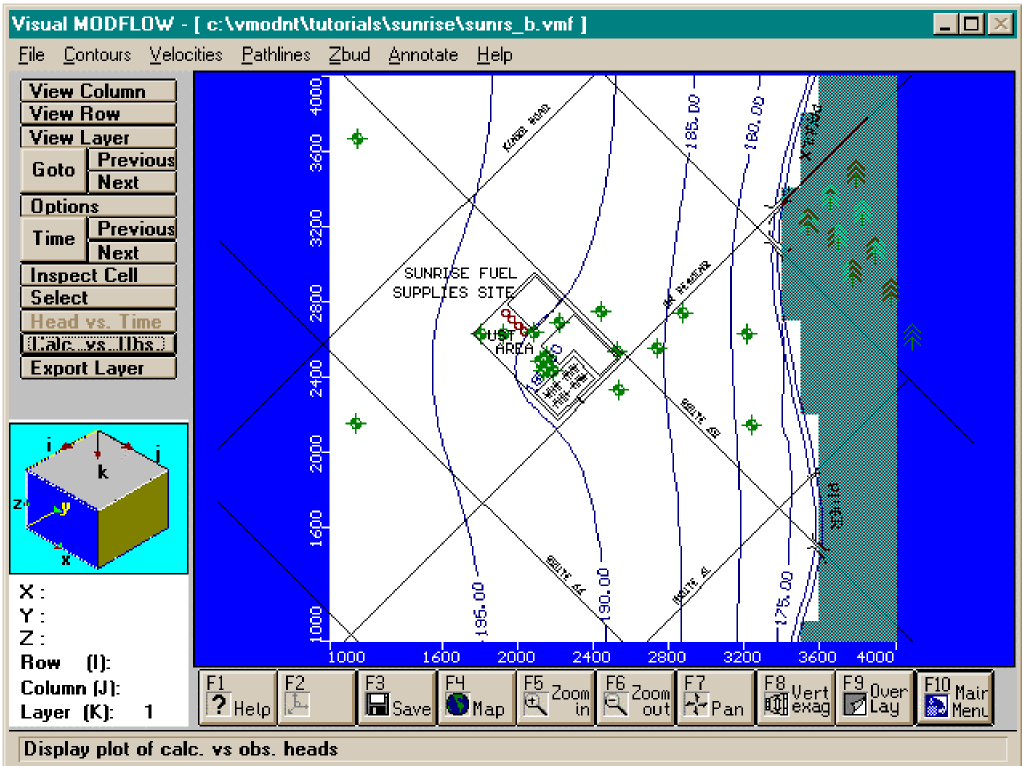
☞	**[F10 - Main Menu]**	(from the bottom button bar)
☞	**[Yes]**	(to save the boundary data)

☞	**Run**	(from the top menu bar)
☞	**[OK]**	(to accept steady state)
☞	**Run**	(from the top menu bar)
☞	**[OK]**	(to execute MODFLOW and MODPATH)

Visual MODFLOW will translate the input files from Visual MODFLOW format to create standard USGS MODFLOW and MODPATH input data files. The program will then activate the Win32 MODFLOW Suite to execute the 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)

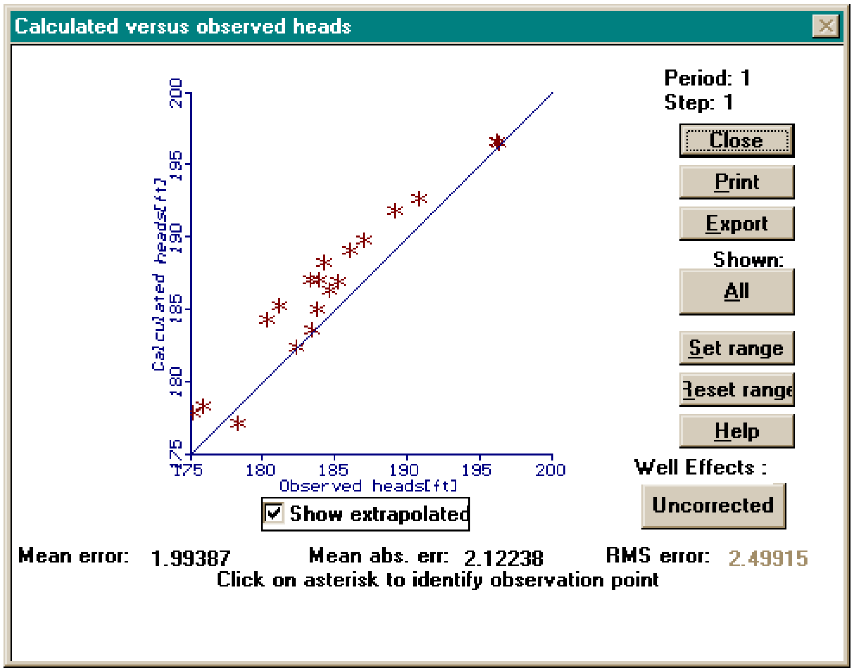
A contour plot of the steady-state heads will be displayed as shown in the following figure.



Compare the calculated vs. observed heads for this scenario.

☞ **[Calc vs Obs]** (from the left-hand menu)

Notice that the majority of the calculated heads are greater than the observed heads, while only a few data points produce a close match (see the figure below). If you look more closely at the graph you will see that the data points which fit are the ones nearest the pumping well, or nearest to the boundaries.



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.

☞ **[Close]**

First zoom in to the area around the pumping well.

☞ **[Select**] (from the left-hand 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 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).

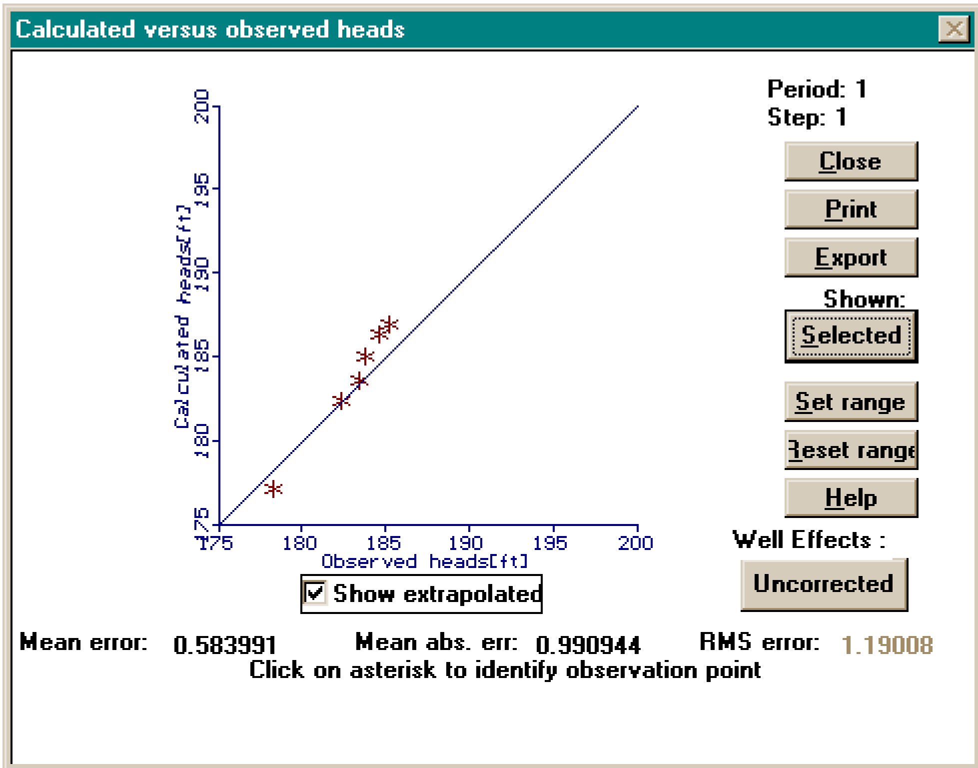
The observation wells located within the box will be highlighted blue, indicating that they have been selected.

☞ **[Calc vs Obs]** (from the left-hand menu bar)

The Calculated vs. Observed Heads graphs will be displayed again and will appear the same as that shown in the figure below. To display the selected observation wells only, click the button along the right-hand side of screen labelled **[All]** to toggle it to **[Selected]**. The display will then show the calibration data for the selected observation wells only, as shown in the last firgure. 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.

☞ **[Close]** (to return to the display of heads)

This example clearly demonstrates the importance of recognizing that steady-state models do not have unique solutions. In addition, it also shows that models which have been calibrated to pumping conditions are generally more reliable since there is less flexibility in the input parameters and boundary conditions to produce a calibrated model.



This concludes the exercise problem. If you have time, you should continue to experiment with the Visual MODFLOW interface and test some of the tools, which have not been covered.

To exit out of Visual MODFLOW,

☞ **File**

☞ **Main Menu**

Then in the Main menu,

☞ **File**

☞ **Exit**