

Demonstration Instructions for

Visual MODFLOW

The standard software package for professional, three-dimensional groundwater flow and contaminant transport modeling.

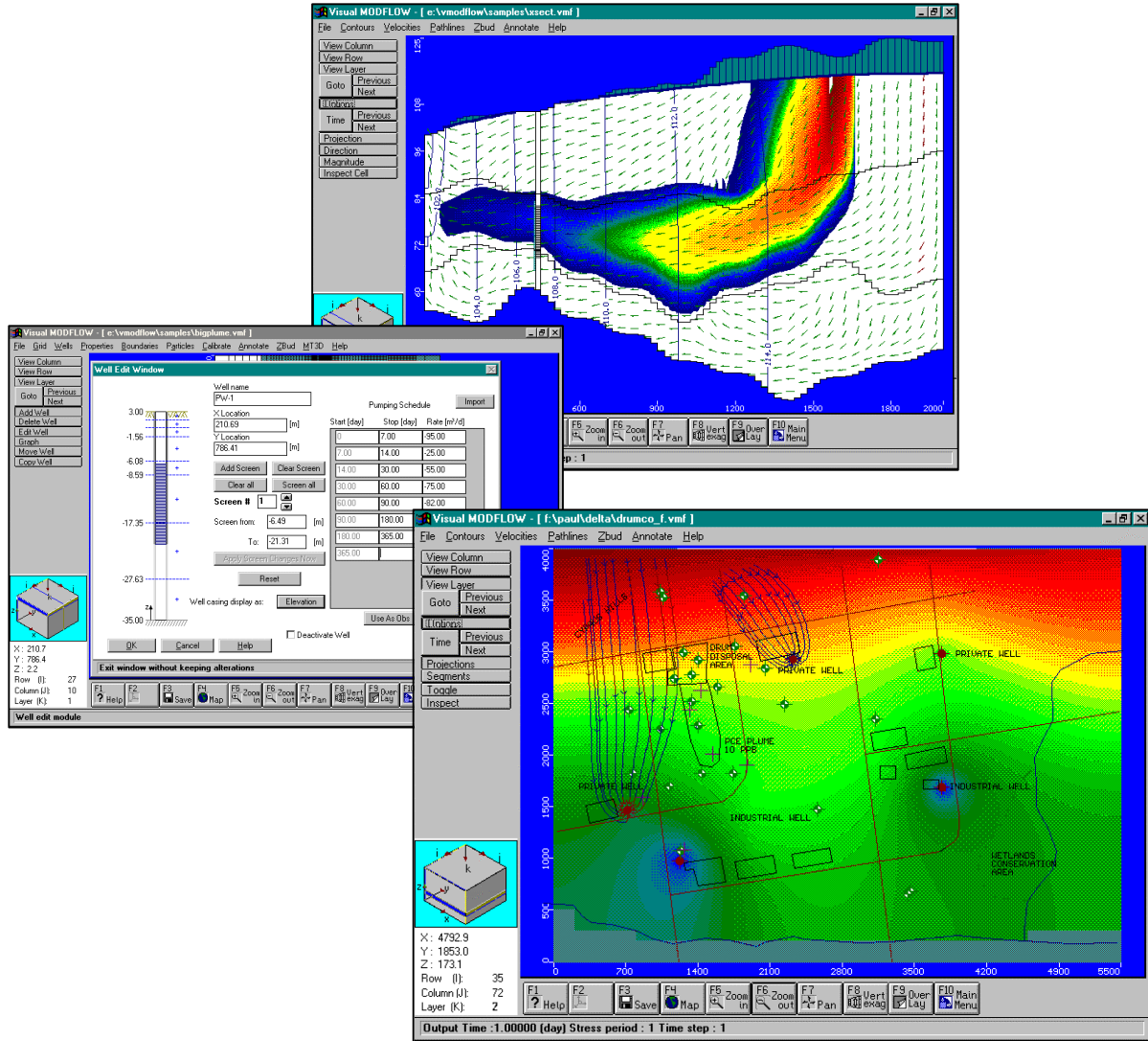


TABLE OF CONTENTS

INTRODUCTION TO VISUAL MODFLOW	3
INSTALLATION.....	4
ABOUT THE VISUAL MODFLOW INTERFACE	4
DEMONSTRATION PROGRAM DESCRIPTION.....	5
DESCRIPTION OF THE EXAMPLE MODEL.....	6
TERMS AND NOTATIONS.....	8
STARTING VISUAL MODFLOW FOR WINDOWS	8
OPENING A FILE	8
MODULE I: MODEL INPUT	10
SECTION 1: Refining the Model Grid	10
SECTION 2: Assigning Pumping Wells	12
SECTION 3: Assigning Model Properties	14
SECTION 4: Assigning Model Boundary Conditions.....	16
SECTION 5: Assigning Particle Tracking Locations	18
SECTION 6: Assigning MT3D Input Parameters.....	19
SECTION 7: Contaminant Transport Calibration.....	20
MODULE II: RUNNING VISUAL MODFLOW	21
MODULE III: OUTPUT VISUALIZATION	25
SECTION 8: Equipotentials and Contouring Options.....	27
SECTION 9: Velocity Vectors and Options.....	29
SECTION 10: Pathlines and Options	32
SECTION 11: MT3D Concentration Contours	35

Please note that it is not necessary to follow the model-input section of this tutorial to proceed to the output display of results. If you are pressed for time you can skip any section of the tutorial and proceed to another section.

INTRODUCTION TO VISUAL MODFLOW

Visual MODFLOW is the most complete and easy-to-use graphical modeling environment for professional 3-D groundwater flow and contaminant transport simulations. This seamless package allows the user to graphically assign all necessary flow and transport parameters, run the simulation [MODFLOW, MODPATH and MT3D are included!], calibrate the model and visualize the results. Both the input and output can be visualized in plan view or full-screen cross-sections at any time. For practical applications in three-dimensional groundwater flow and contaminant transport modeling, Visual MODFLOW is the only software package you will ever need!

Visual MODFLOW was first released in August 1994 and is currently being used by over 3,000 consultants, regulators and educators world wide. It is used by both the USGS and the USEPA. In addition, Visual MODFLOW is *the* featured model in many continuing education courses around the world.

Contact Scientific Software Group for information
on these courses. E-mail: info@scisoftware.com.

When you purchase Visual MODFLOW, you are getting the established model of choice for practicing groundwater professionals and environmental engineers. In addition, you also get free technical support from our team of programmers and qualified modeling professionals.

To obtain more information, or to order the program, please contact us.

INSTALLATION

The Visual MODFLOW for Windows demonstration program is distributed on three 3.5" floppy diskettes. The diskettes are formatted for standard IBM PCs.

If you have any difficulties while installing the demo or running through the tutorial, please contact Scientific Software Group technical support staff at (703) 620-9214 or email us at tech@gwsoftware.com.

Note: You will need at least 8 Mb of extended RAM (in excess of the operating system requirements) and 15 Mb of hard disk space available to run this example. Make sure your machine meets these requirements before proceeding.

Windows 95/Windows NT:

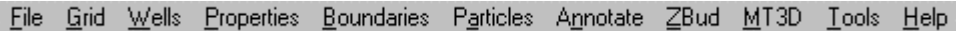
- 1** Insert disk #1 of **Visual MODFLOW Demo Version 2.7** in your floppy disk drive.
 - ☞ **[Start]** (bottom left corner of the screen), then
 - ☞ **[Run]**, and Type "A:\setup32.exe"
 - ☞ **[OK]** to start the installation process
- 2** Enter the path for the **Visual MODFLOW** demo installation, or click **[OK]** to start copying files to the specified default path.
- 3** Once all three diskettes have been loaded, click **[OK]** to complete the installation procedure.

You should now see a program group called "WHI Software" on your screen with the **Visual MODFLOW** demo icon inside this window. To start the Visual MODFLOW demo, double-click the Visual MODFLOW icon.

ABOUT THE VISUAL MODFLOW INTERFACE

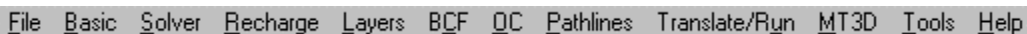
The Visual MODFLOW interface has been specifically designed to increase modeling productivity and decrease the complexities typically associated with building three-dimensional groundwater flow and contaminant transport models. The interface is divided into three separate modules, the Input Module, the 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, 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 (shown below) represent the basic 'building blocks' for developing a model data set for MODFLOW, MODPATH and MT3D. These menus are displayed in a logical order to guide the user through the steps required to design a groundwater model.



A screenshot of a menu bar with the following items: File, Grid, Wells, Properties, Boundaries, Particles, Annotate, ZBud, MT3D, Tools, Help.

The Run Module allows the user to modify the various MODFLOW, MODPATH and MT3D parameters which are run-specific, such as selecting initial head estimates, setting solver parameters, activating the re-wetting package, specifying the output controls, etc. Each of these menu selections (as shown below) have default settings which will run most simulations.



A screenshot of a menu bar with the following items: File, Basic, Solver, Recharge, Layers, BCF, QC, Pathlines, Translate/Run, MT3D, Tools, Help.

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



A screenshot of a menu bar with the following items: File, Contours, Velocities, Pathlines, Zbud, Annotate, Tools, Help.

Each screen of the Visual MODFLOW interface has a set of shortcut buttons located along the bottom of the screen as shown below. These buttons allow the user to quickly access some of the more common graphical functions such as zooming in or zooming out, panning the display in any direction, adding or removing overlay displays, saving the existing data file, and returning to the Main Menu.

DEMONSTRATION PROGRAM DESCRIPTION

The demonstration program is fully functional and provides you with all of the tools necessary for pre-processing and post-processing of MODFLOW, MODPATH and MT3D. The demo allows you to design, modify and save any kind of model input file you like. However, the code necessary to translate the Visual MODFLOW files to

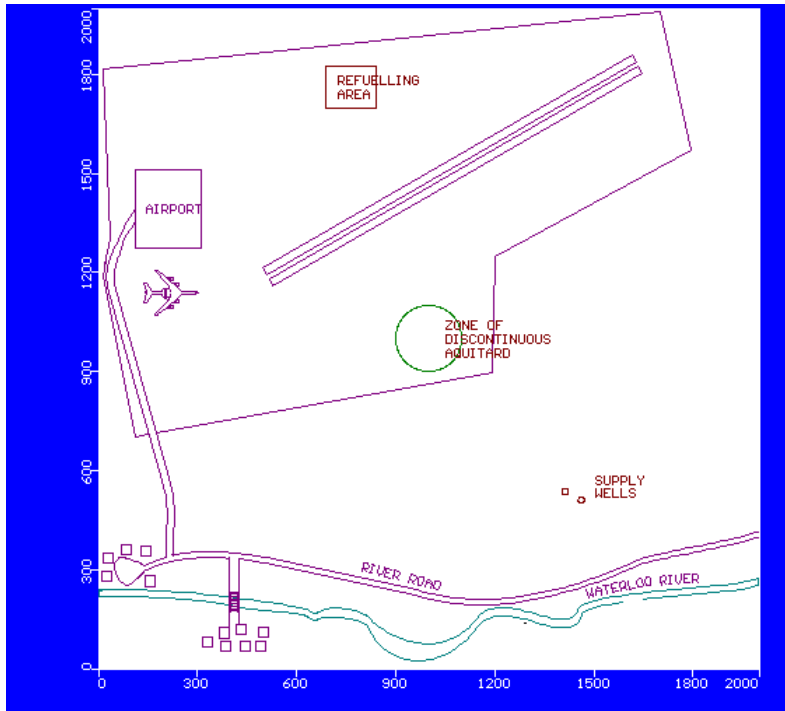
MODFLOW, MODPATH and MT3D readable format has been disabled and the code required to execute the numerical engines has been removed.

This demo also provides you with a complete set of input and output files for an example model (Output) to allow you to examine the post-processing features and capabilities of Visual MODFLOW. The numerical simulations (MODFLOW, MODPATH and MT3D) for this problem have already been completed to allow you to evaluate the output visualization features for the sample model results.

This demonstration guides you through some of the steps necessary to design and run a model, and visualize the results. The instructions for the demonstration are provided in a step-wise format that allows you to choose the features that you are interested in examining without having to complete the entire exercise.

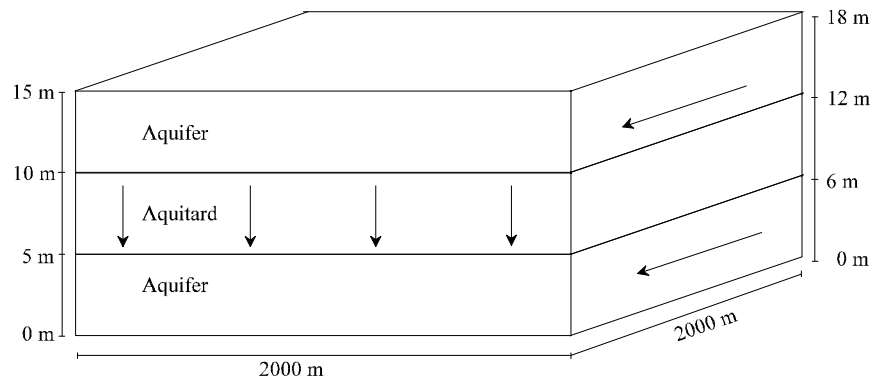
DESCRIPTION OF THE EXAMPLE MODEL

The site is located near an airport just outside of Waterloo. The surficial geology at the site consists of an upper sand and gravel aquifer, a lower sand and gravel aquifer, and a clay and silt aquitard that separates the upper and lower aquifers. The relevant site features consist of a plane refuelling area, a municipal water supply well field, and a discontinuous aquitard zone. These features are illustrated in the following figure. The Municipal Well Field consists of two wells. The East Well pumps at a constant rate of 400 m³/day, while the West Well pumps at a constant rate of 550 m³/day. Over the past ten years, aviation fuel has been periodically spilled in the refuelling area. Infiltration of this fuel has produced a plume of contamination in the upper aquifer. This tutorial will guide you through the steps necessary to build a groundwater flow and contaminant transport model for this site. This model will demonstrate the potential impact of the fuel contamination on the municipal water supply wells.



When discussing the site, in plan view, the top of the site will be designated as north. Groundwater flow is from north to south (top to bottom) in a three-layer hydrogeologic system consisting of an upper unconfined aquifer, an intervening middle aquitard, and a lower confined aquifer as illustrated in the figure below. The upper aquifer and lower aquifers have hydraulic conductivities of 2×10^{-4} m/sec and the aquitard has a hydraulic conductivity of 1×10^{-10} m/sec.

Recharge = 10 cm / year



TERMS AND NOTATIONS

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

- type - type in the given word or value
- select - click the left mouse button where indicated
- ↵ - press the <Enter> key
- ↔ - press the tab key to toggle between input boxes
- ☞ - click the left mouse button where indicated
- ☞☞ - double-click the left mouse button where indicated

STARTING VISUAL MODFLOW FOR WINDOWS

In any Windows environment, the VMDEMO icon will be displayed in the WHI Software program group. Simply double-click the icon to start Visual MODFLOW for Windows.

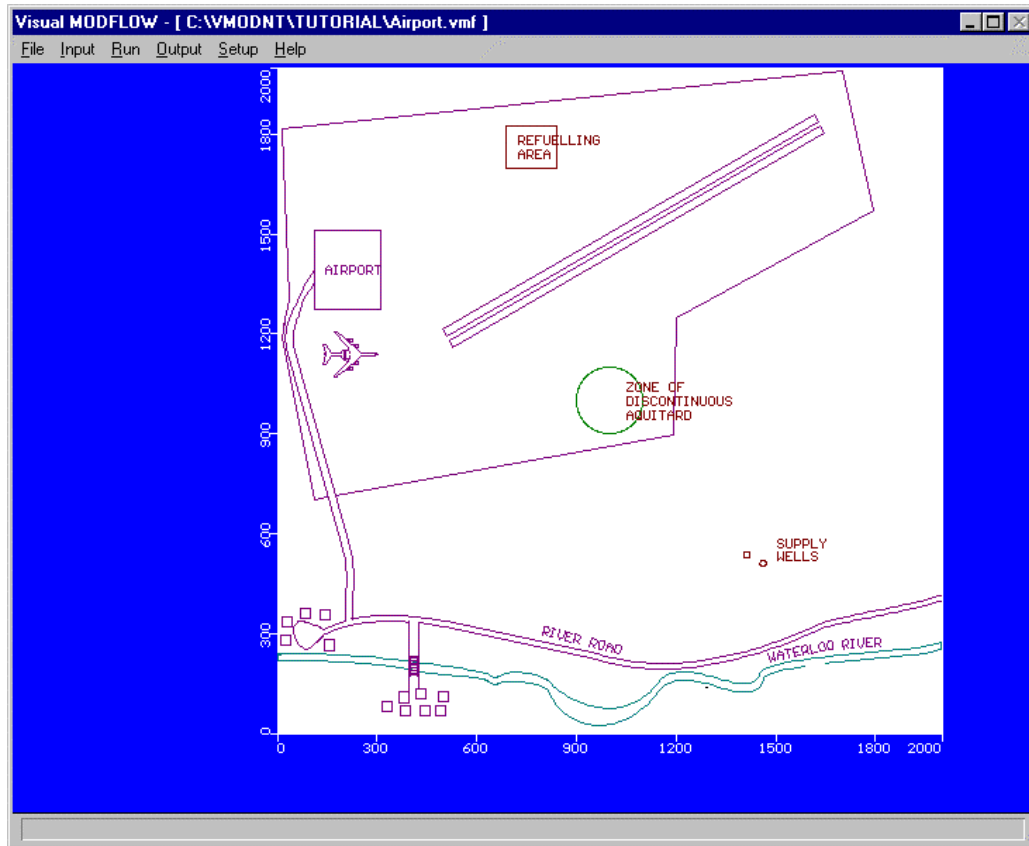
You are now at the opening screen of Visual MODFLOW for Windows.

- ☞ [OK] to proceed to the Main Menu.

OPENING A FILE

- ☞ **F**ile from the top menu bar and a drop-down menu will prompt you to make a selection from a list of standard file options.
- ☞ **O**pen and a file selection window will appear
- ☞ **demo.vmf** from the list of files
- ☞ [**O**pen] to open the file

After a few seconds the Main Menu will appear with a model grid and site map displayed, as shown in the following figure.



This section of the demonstration will introduce the Visual MODFLOW input menus and briefly describe the functionality and main features of each one.

MODULE I: MODEL INPUT

SECTION 1: Refining the Model Grid

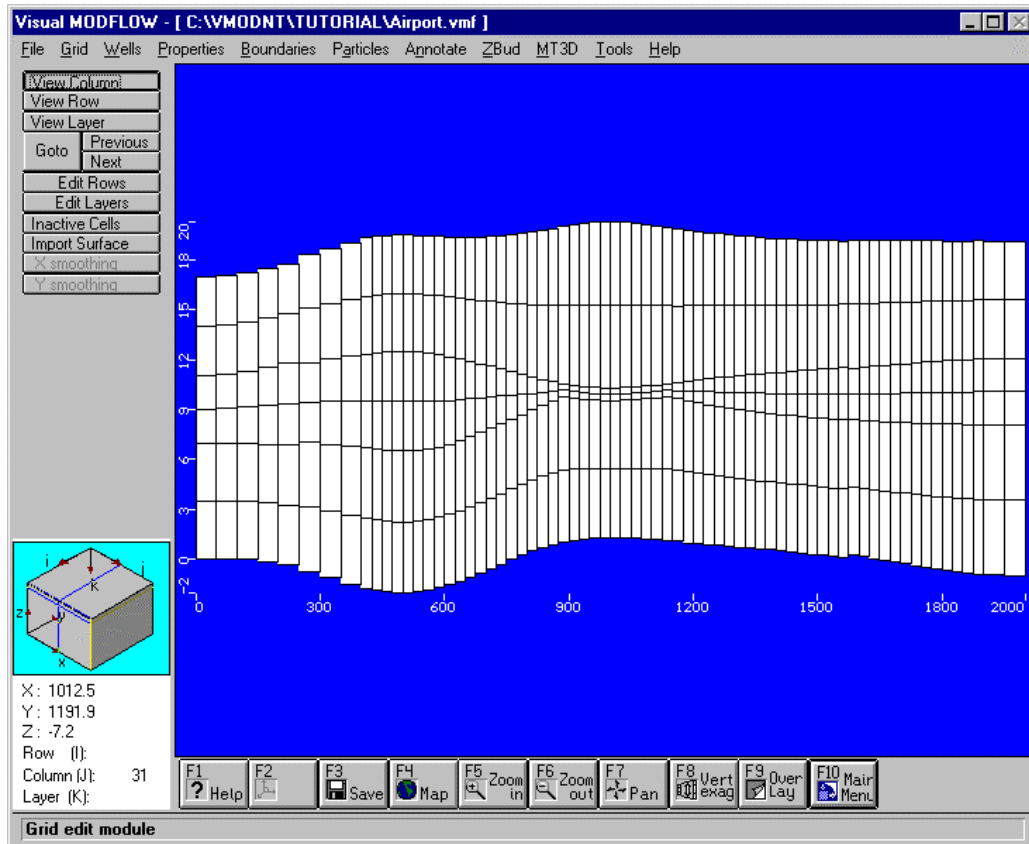
☞ **[Input]** from the top menu bar

You will then be transferred to the Input Module where the “building blocks” for a model are organized as a menu across the top. Visual MODFLOW for Windows loads the **[Grid]** input screen when you first enter the Input Module. The top four buttons on the left side of the screen (**[View Column]**, **[View Row]**, **[View Layer]** and **[Go to]**) appear on every screen and allow you to change the model display from plan view to cross-section at any time. The remaining buttons along the left side of the screen describe the various functions that can be performed to modify the model grid.

The first thing to do is to switch from the current plan view of the model to a cross-sectional view.

☞ **[View Column]** from the left menu

Then move the mouse into the model grid. The grid columns corresponding to the mouse location will be highlighted. To see one of the columns in cross-section, click the left mouse button on one of the columns. A cross-sectional view of the model grid will be displayed as shown in the following figure.



Use the **[Next]** and **[Previous]** buttons along the left menu to advance through the various column cross-sections of the model. Use the navigator cube along the bottom left side of the screen to determine your relative location within the model grid as you advance along.

Now return to the plan view display of the model.

☞ **[View Layer]** from the left menu bar

Move the mouse pointer into the model domain and, once the top layer is highlighted red, click the left mouse button once.

Now try adding a row.

☞ **[Edit Rows]** from the left menu bar

A "Rows" window will appear prompting you to choose one of the editing features. The Add Row option is the default setting. Once you move the mouse into the model domain, a horizontal red line will appear indicating the location of any new row you wish to add. Click on the location you wish to add the first row. A thin green horizontal line will appear within the model domain. This line represents the location of the new grid row. Once you move the mouse, the new grid line will turn blue, and the red line will appear for adding additional rows. To add a row at an exact known location, click the RIGHT MOUSE BUTTON and an Add Horizontal Line window will appear. Examine the options available to you and then select the [OK] button. Experiment with the remaining [Edit Row] options or click [Close] to exit the [Edit Row] options.

The [Edit Column] option functions similarly to the [Edit Row]. The [Import Surface] button allows you to import an irregular layer surface from either a SURFER .GRD file or, an x, y, z ASCII file. If you have time, try some of the other buttons to get a feel for the powerful tools that Visual MODFLOW for Windows provides for building and modifying model grids.

The [X smoothing] and the [Y smoothing] options allow you to evaluate the grid design and automatically modify the grid spacing to achieve a smoother transition from large grid spacing to smaller grid spacing. This will decrease the solution convergence times and can often result in convergence of non-convergent problems. This is a more advanced option and is not discussed in this demo. However, it is fully functional with on-line help if you wish to examine the applications of grid smoothing.

SECTION 2: Assigning Pumping Wells

Next you will examine the unique well-input system of Visual MODFLOW. First we will turn off the grid overlay to facilitate viewing of the map.

☞ [F9 - Overlay] button from the bottom menu bar.

An Overlay Control window will appear with an alphabetical listing of all of the available overlays that you may turn on or off. Scan down the listing and locate the **Grid Overlay**. Click **Grid Overlay** to highlight it, and then click on the button labelled [ON] to toggle it to [OFF]. This should delete the asterisk beside the **Grid Overlay**, indicating that it has been deactivated. Select the [OK] button to continue.

☞ Wells from the top menu bar.

☞ Pumping Wells

You will then be prompted to save you data.

☞ [Yes]

☞ [Add Well] from the side menu.

Move the cursor to the east supply well and click on it. A well edit window will appear prompting you to enter the specific well information.

Start [day]	Stop [day]	Rate [m³/d]
0	7300.00	-400.00

In the box labeled **Well Name:**

type: Supply Well 1

In the boxes labeled **X Location** and **Y Location type :**

X Location: 1461 ↔

Y Location: 511 ↔

Next click the [Add Screen] button. Move the mouse pointer into the well borehole at the left of the window and click once at an elevation of **5.0 m** to anchor the top elevation of the screen, and then click again at **0.3 m** to close the screen interval.

Alternately you can assign exact values for the well screen by entering values in the boxes provided.

To enter the well pumping schedule, move the mouse pointer inside the box under the column labelled **Stop**, click once and enter the following information.

Stop [day]: 7300 ↔

Rate [m3/d]: -400 ↔

☞ **[OK]** to accept this well information.

You can use a short cut to assign the well parameters for a second pumping well by using **[Copy Well]** which allows you to copy the characteristics from one well to another location.

☞ **[Copy Well]** (from the side menu bar)

Click the well you wish to duplicate and then click the location you wish to place the new well. You can then use the **[Edit Well]** option to edit the new well settings.

SECTION 3: Assigning Model Properties

Next you will examine the input system of Visual MODFLOW for Windows for layers with contrasting hydraulic conductivities.

☞ **Properties** from the top menu bar.

☞ **Conductivity**

You will then be prompted to save your data.

☞ **[Yes]**

You will then be transferred to the Properties input screen with the graphical input tools displayed along the left menu bar.

☞ **[F9 – Overlay]**

☞ ☞ **Grid Overlay** (to turn the grid back on)

If you were creating a new model a Default K Property window would appear prompting you to enter initial values for hydraulic conductivity, storage, specific yield and porosity. These values would then be assigned to each cell in the model.

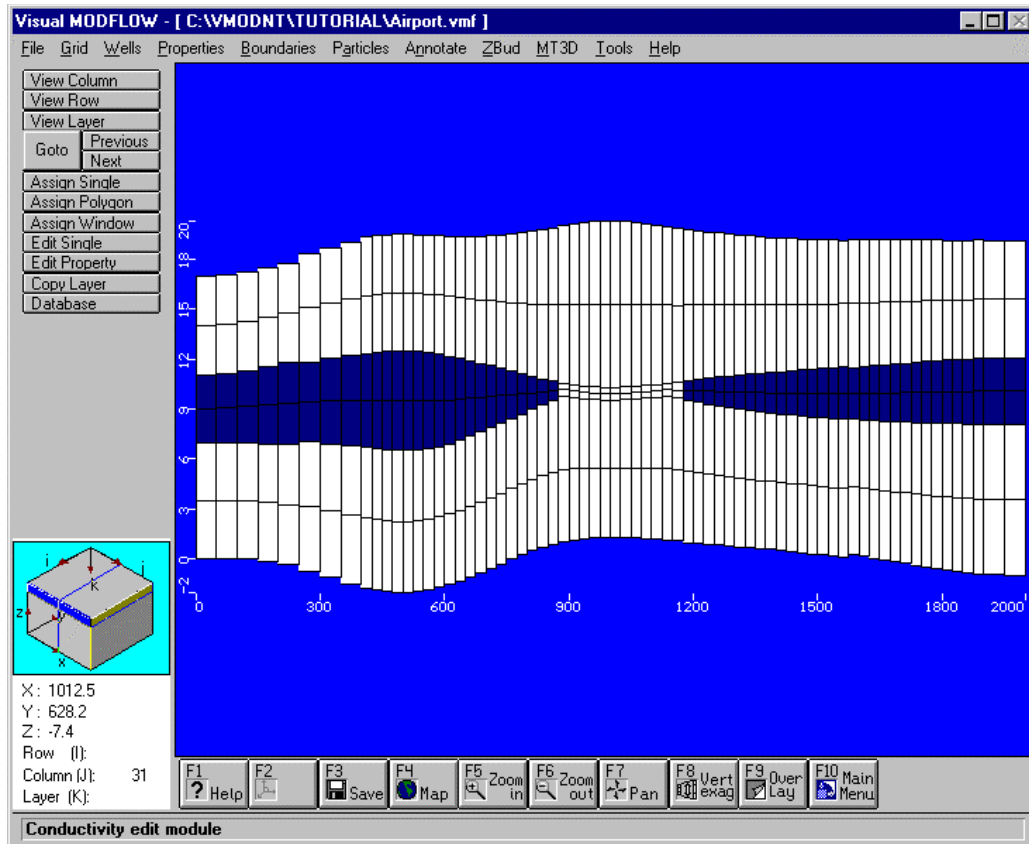
Areas or layers containing different hydraulic conductivities can be entered into the model by using the **[Assign Window]**, **[Assign Polygon]** or **[Assign Single]** graphical tools.

The **[Assign Window]** function allows you to assign a different hydraulic conductivity inside of a rectangular window. First select **[Assign Window]** from the menu on the left of the screen, then move the mouse pointer to the upper left corner of the area you want to select and click the left mouse button. Now move the cursor to the lower right hand corner of the area selected and click the left mouse button.

A dialogue box will appear where you can assign the current property number or you can enter a new value by clicking **[New]**. Enter a new value or click **[OK]** or **[Cancel]** to close the dialogue box.

☞ **[F6 Zoom Out]** to refresh the screen

☞ **[View Column]** to look at the layers in cross section. Move the mouse pointer (a highlighted bar) towards the circle in the middle of the .dxf background map (the discontinuous aquitard zone). Select the desired column in this location by clicking the left mouse button. A cross-section will be displayed as shown in the following figure.



Select [**D**atabase] (from the left menu bar) to view the numerical values of the different hydraulic conductivities.

☞ **[OK]**

Return to the plan view display by selecting the [**V**iew **L**ayer] button from the left menu and then clicking Layer 1 in the model cross-section.

SECTION 4: Assigning Model Boundary Conditions

Next we will have a look at some of the steps required to assign a constant head boundary.

☞ **B**oundaries from the top menu bar.

☞ **Constant Head**

You will then be prompted to save your data.

☞ **[Yes]**

You will then be transferred to the Boundaries input screen with the available graphical tools indicated along the left menu bar. The River boundary condition has already been assigned along the south section of the model.

Next, you will assign a constant head boundary condition will be assigned to the upper unconfined aquifer along the northern boundary of the model domain.

☞ **[Assign Line]** from the left menu bar

Move the mouse to the northwest corner of the map and click on the center of the cell. With the right mouse button, click on the center of the cell in the northeast corner of the map. A horizontal line of cells will be highlighted and an **Assign Constant Head** dialogue box will appear as follows.

Start Time [day]	Stop Time [day]	Constant Head [m]
0	7300.00	19.00
End Pt.		19.00

Enter the following values:

Code #: **1**

☞ **Assign to appropriate layer**

☞ in **Stop time** box

Stop time: **7300** ↔

Start point: **19** ↔

End point: 19 ↔

☞ [**OK**] to accept these values

The pink line will now turn to a dark red indicating that a constant head boundary value has been assigned. Now we will assign this value to the second layer.

☞ [**Copy Layer**] from the left menu

The Copy window will appear with the default setting '**Copy only code #:**' and a value of '1' in the adjacent box.

☞ **Layer 2** (this will highlight it)

☞ [**OK**] to copy Constant Head Code #1 to Layer 2.

Note that a river boundary (blue line) in the southern portion of our model domain has been pre-defined. If you wish to see the values assigned to the river, click on Boundaries (from the top menu) then Rivers. Click on Edit Single and then click on any of the cells highlighted blue. An **Edit River** dialogue box will appear displaying the River boundary input values for that cell. Click [**Cancel**] to exit.

SECTION 5: Assigning Particle Tracking Locations

In this section you will be guided through the steps necessary to assign forward tracking particles to determine the preferred contaminant exposure pathways.

☞ **Particles** from the top menu bar.

☞ [**Yes**] to save your data

You will then be transferred to the Particles input screen with the available graphical tools indicated along the left menu bar.

☞ [**Add Line**]

Move the mouse pointer to the left of the refueling area, north of the landing strip, click the left mouse button and stretch a line to the right of the Refueling Area. Then click again. An **Add Particles** dialogue box will appear.

☞ [**OK**] to assign the line of forward tracking particles near the Refueling Area

SECTION 6: Assigning MT3D Input Parameters

Now we will go over the input procedure for MT3D

☞ **MT3D** from the top menu

☞ [**Yes**] to save your data.

You will then be transferred to the MT3D input module where the menus across the top of the screen indicate the available MT3D input parameters. The default MT3D input screen is the Initial Concentration input.

The source of contamination at the Refueling Area will be designated as a Recharge Concentration, which serves as a source of contamination to infiltrating precipitation.

☞ **Boundaries** from the top menu bar

☞ **Recharge Conc.**

A precipitation recharge area (shaded rectangle) has been pre-defined. Check to ensure that you are viewing Layer 1 by referring to the navigator cube on the lower left side of the screen. If Layer 1 is not displayed, use the [**Previous**] or [**Next**] buttons on the left menu bar to advance to Layer 1.

Then assign a concentration to the natural recharge entering the model at the refueling area:

☞ [**Assign Polygon**] from the left menu bar.

Move the mouse pointer around the perimeter of the refueling area clicking at each corner of the area with the left mouse button, then click the right mouse button and the area will become shaded pink. The **Assign Recharge Concentration** dialogue box will appear prompting you to enter the following:

Code #: **1** ↔

Stop Time [days]: **7300** ↔

Recharge Conc. [mg/l]: **5000** ↔

☞ [**OK**] to accept these Recharge Concentration values.

SECTION 7: Contaminant Transport Calibration

Finally, you will add an observation well to the model to monitor the concentration at a discrete location in the model domain.

☞ **Calibrate** from the top menu bar

☞ **[Yes]** to save your data.

You will then be transferred from the Initial Concentrations input screen to the MT3D Calibrate input screen where you can assign monitoring points for calibrating the contaminant transport simulation to observed field data.

☞ **[Add Obs.]**

Move the mouse pointer to anywhere in the model domain and click the left mouse button. An **Edit Observation Point** dialogue box will prompt you for the following information:

Edit Observation Point

Observation name: OW 1

X Location: 760.00 [m]

Y Location: 1667.00 [m]

Z location of observation point: 2.40

Set obs. point

Z-axis displayed as: Elevation

Time [day]	Observed Conc. [mg/L]
7300.00	375

OK Cancel Help

Observation Name: OW1 ↔

X coordinate: 760 ↔

Y coordinate: 1667 ↔

Z coordinate: **2.4** ↔

Now enter the observed concentrations.

☞ ☞ **Time:** **7300** ↔

Observed Conc.: **375** ↔

☞ [**OK**] to accept these values.

Select **F10** or [**Main Menu**] from the bottom menu bar to return to the main menu.

☞ [**Y**es] to save your data.

This completes the MODULE I: MODEL INPUT. Please continue to explore the many powerful graphical features available for building a groundwater flow and contaminant transport model using Visual MODFLOW.

MODULE II: RUNNING VISUAL MODFLOW

The following guides you through the selection of some of the MODFLOW, MODPATH and MT3D run options which are available with any version of Visual MODFLOW.

If you have just completed the Input Section, then you may skip the instructions inside this box.

☞ **F**ile

☞ **O**pen

A File Selection window will appear.

☞ **demo.vmf**

The file will be opened and you will be presented with the Visual MODFLOW Main Menu.

☞ [**R**un] from the Main Menu options

You will then be transferred to the Run options screen and a window will prompt you to select either a **Steady-state** or **Transient** simulation.

☞ [**OK**] to accept the default steady state simulation

The Run options in Visual MODFLOW are divided into two separate sections; run options for flow simulations, and run options for MT3D simulations. When entering the Run option screen, Visual MODFLOW will default to the run options for flow simulations.

☞ **B**asic from the top menu bar

☞ **I**nitial Heads

The default initial condition for a new model is **Constant by layer**. Visual MODFLOW determines an initial guess for each layer based on the elevations and the boundaries in each layer. For simple problems this usually leads to a convergent solution. However, a better initial guess may significantly decrease the number of iterations required for convergence to occur. This initial heads guess can be important from either an ASCII (x,y,z) file, a SURFER .GRD file or a previous Visual MODFLOW run.

☞ **C**ancel to close this window.

Next you will examine the selection of numerical solvers provided by Visual MODFLOW.

☞ **S**olver

The default solver is the WHS Solver, a proprietary solver developed by Waterloo Hydrogeologic. It is the fastest and most stable MODFLOW solver available. For this example, the WHS Solver was used to calculate the flow solution.

☞ **[O**K] to accept the WHS Solver

A **WHS Solver Parameters** dialogue box will appear showing all of the default solver settings.

☞ **[O**K] to accept the default solver settings

There are still several other Run options for flow simulations. If you have time, investigate the options that were not mentioned in this tutorial.

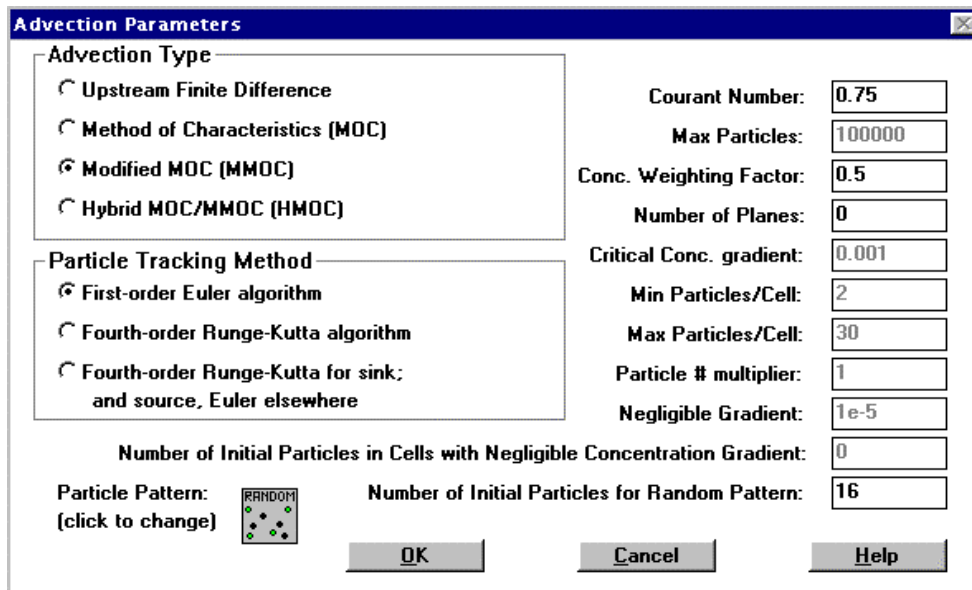
Now you will examine the Run options for MT3D simulations.

☞ **M**T3D from the top menu bar

This will transfer you to the MT3D run options screen.

☞ **A**dvection from the top menu bar

An **Advection Parameters** dialogue box will appear as shown in the following figure.

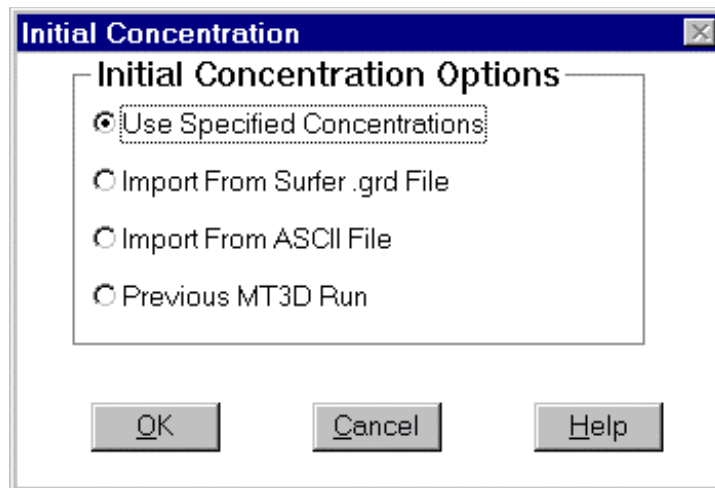


☞ **Modified MOC(MMOC)** from the Advection Type selections

☞ **[OK]** to accept the remaining default parameters

This will return you to the MT3D run options screen.

☞ **Initial concs.**



An **Initial Concentration** dialogue box will appear as shown above. You have the option of using the specified initial concentration, importing the concentrations from a SURFER .GRD file, an ASCII file or a previously MT3D run.

☞ **[OK]** to accept the default option of using the specified initial concentrations.

This concludes the MT3D run options section of the tutorial. If you have time, investigate some of the other options available.

Although this demo does not allow the numeric engines of MODFLOW, MODPATH, Zone Budget, or MT3D to be executed, to run the models you would select **Translate/Run** from the top menu bar and a Run Settings window would appear:



You would select the appropriate numerical engines for the analysis and Visual MODFLOW would translate the information provided in the Input section of the program into standard MODFLOW, MODPATH and MT3D input files with the appropriate extensions (.BAS, .WEL, .BCF etc.). Visual MODFLOW would then execute the simulation. When the simulation was completed you would be returned to the Visual MODFLOW Main Menu.

To continue with this demonstration, select the **[F10 - Main Menu]** button from the lower menu bar.

MODULE III: OUTPUT VISUALIZATION

Since this demo does not allow you to run the model analysis, you will be visualizing results from the **OUTPUT** model simulation. These are the results that you would have obtained had you been able to run the model simulation.

OPENING THE OUTPUT.VMF FILE

Please check to ensure that you are in the Visual MODFLOW Main Menu. The top menu bar should read **File, Input, Run, Output, Set-up, and Help**. If you are not at the Main Menu then press the [F10] key.

☞ **File** from the top menu

☞ **Open**

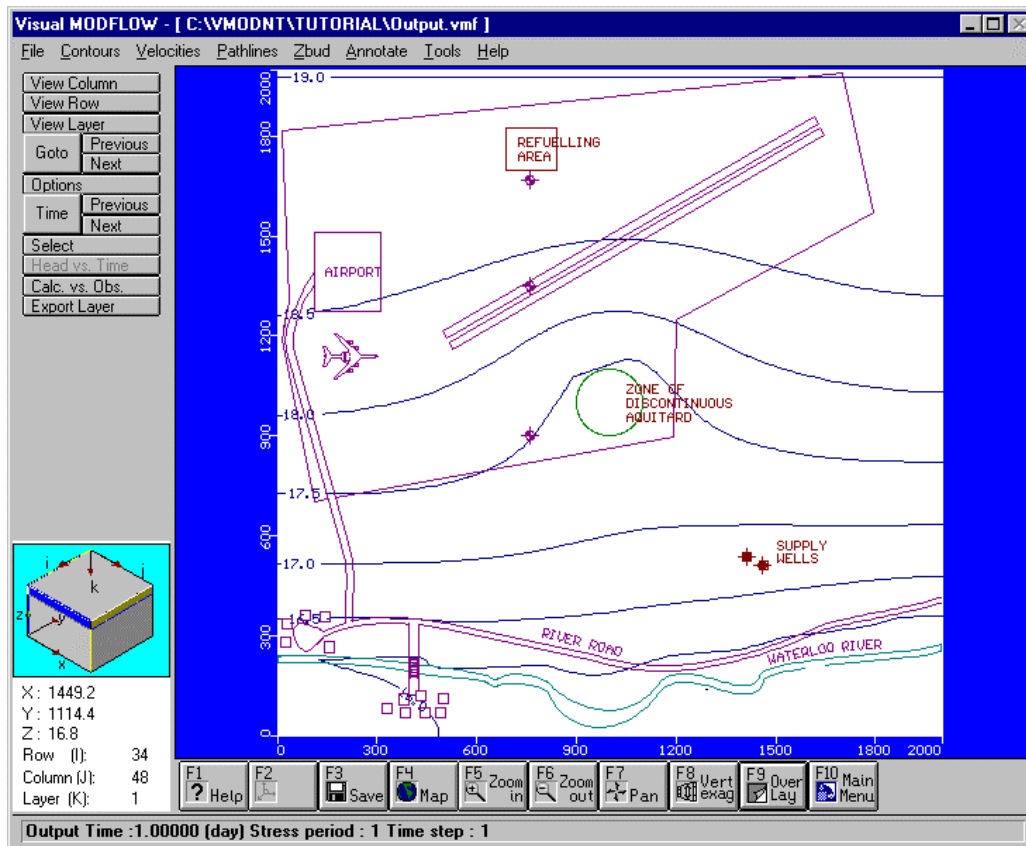
A file selection window will appear with a list of available Visual MODFLOW input files (*.vmf).

☞ **Output.vmf**

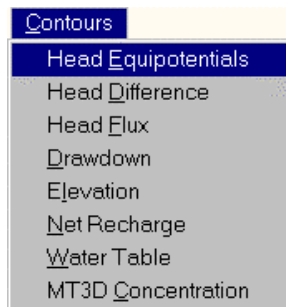
☞ **[Open]**

☞ **Output** from the top menu bar

You will then be transferred to the Visual MODFLOW Output options screen. The default display is a plot of the equipotential contours in layer 1 of the model.



To see the following list of contouring possibilities select **C**ontours from the top menu bar.



Select Head Equipotentials to return to the original display.

SECTION 8: Equipotentials and Contouring Options

To select contouring options for Equipotential Heads:

☞ [Options] from the left menu bar.

The Equipotential Overlay Contouring Options window will appear as shown in the following figure, except without the colour shade bar on the right.

The screenshot shows the 'Equipotential Overlay Contour Options' dialog box. It contains several sections for configuring contouring parameters:

- Automatic reset minimum, maximum and interval values:**
- Color shading:** . Includes input fields for Minimum (15.5 m), Maximum (19 m), and Interval (0.25 m). A 'Labels' field is set to 2 decimals. A 'Contour color' dropdown is set to blue.
- Automatic contour levels:** . Includes a 'Custom contour levels' checkbox (unchecked) and a list box containing '0.0000000000000000'.
- Ranges to color:** A table with columns for 'Max', 'Ranges to color', and 'Basic points'.

Max	Ranges to color	Basic points
19.000000	[Dark Red]	<input checked="" type="checkbox"/>
18.450981	[Red]	<input type="checkbox"/>
17.833334	[Yellow]	<input type="checkbox"/>
17.490196	[Light Green]	<input type="checkbox"/>
16.872549	[Green]	<input type="checkbox"/>
16.598040	[Teal]	<input type="checkbox"/>
16.049019	[Blue]	<input type="checkbox"/>
Min 15.500000	[Dark Blue]	<input checked="" type="checkbox"/>
- Cut off levels:** 'upper' and 'lower' fields, both set to 0.000000.
- Buttons:** 'Reset Min, max, interval', 'OK', and 'Cancel'.
- Contouring resolution/Speed:** A button labeled 'High/Slow'.

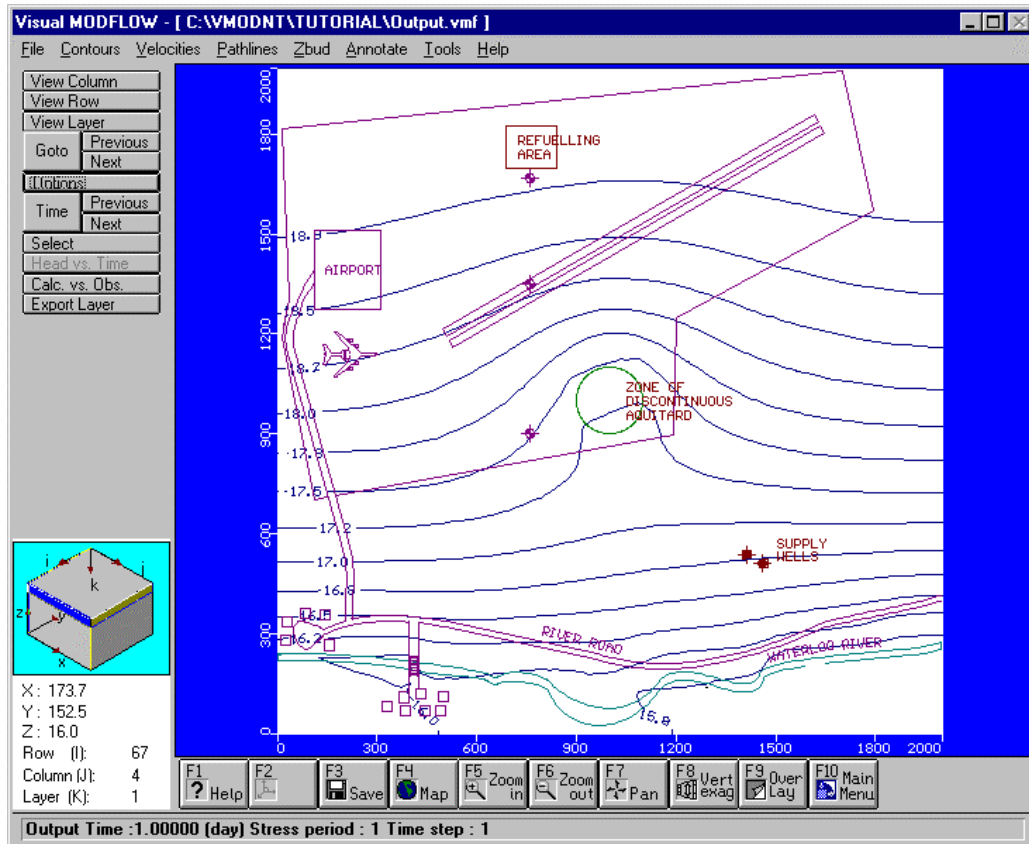
To activate colour shading:

Colour shading

In the box labelled **Interval**, change the value from **0.5** to **0.25**. In the box labelled **Labels**, change the number of decimals to 2.

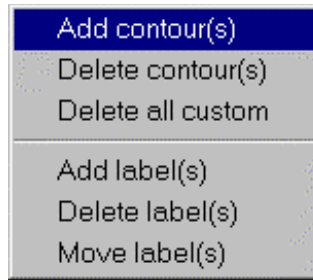
You can also change the speed of the contouring by clicking on the button labelled **Contouring resolution/speed**. Click this button once to increase the speed of the contouring by a factor of two. Click the button until it reads "**High/Slow**".

☞ [**OK**] to accept these contouring options.



The location of the contours should be very similar to the previous plot, however, the above plot does not show colour shading.

Pressing the RIGHT MOUSE BUTTON inside the model domain activates other contouring options that are not activated by the menu buttons. This will bring up a pop-up window (as shown below) with options for adding, deleting and moving contours and labels.



Select the **Add Contour(s)** option and then move the mouse to anywhere in the model domain and click the LEFT MOUSE BUTTON once. A contour line will be added at the location that you clicked the mouse. To add another contour line, simply click the LEFT MOUSE BUTTON again. To finish adding contour lines, click the RIGHT MOUSE BUTTON once. To retrieve the pop-up contour options window, simply press the RIGHT MOUSE BUTTON again. This time select the **Move Label** option and move the mouse to the location of a contour label that you wish to move. Press and hold the mouse on the label and then drag the label to the desired location along the contour line and release the mouse button to set the new label location. When you are finished moving contour labels, press the RIGHT MOUSE BUTTON.

Next, take a look at the site in cross-section. Select the [**View Column**] button from the left menu bar and then move the mouse into the model domain. Choose a column near the middle of the model domain by clicking the left mouse button.

To remove the custom contours, click the RIGHT MOUSE BUTTON in the cross-section and the contouring options pop-up window will appear.

☞ **Delete all custom**

☞ **View Layer** to return to the plan view of the airport site.

Before proceeding to the next section the colour shading should be turned off.

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

☞ **Colour shading (to remove the check mark)**

☞ [**OK**]

SECTION 9: Velocity Vectors and Options

☞ **Velocities** to view the velocity vectors

You will automatically be transferred to the Velocity Vectors output options screen as shown in the following figure. The velocity vectors will be plotted according to the default settings, which plot the vectors in relative sizes according to the magnitude of the flow velocity.

To make the velocity vectors independent of magnitude:

☞ **[Direction]** from the left menu

All of the arrows will then be plotted as the same size and simply act as flow direction arrows.

☞ **[Options]** from the left menu

Change the number of vectors displayed by typing:

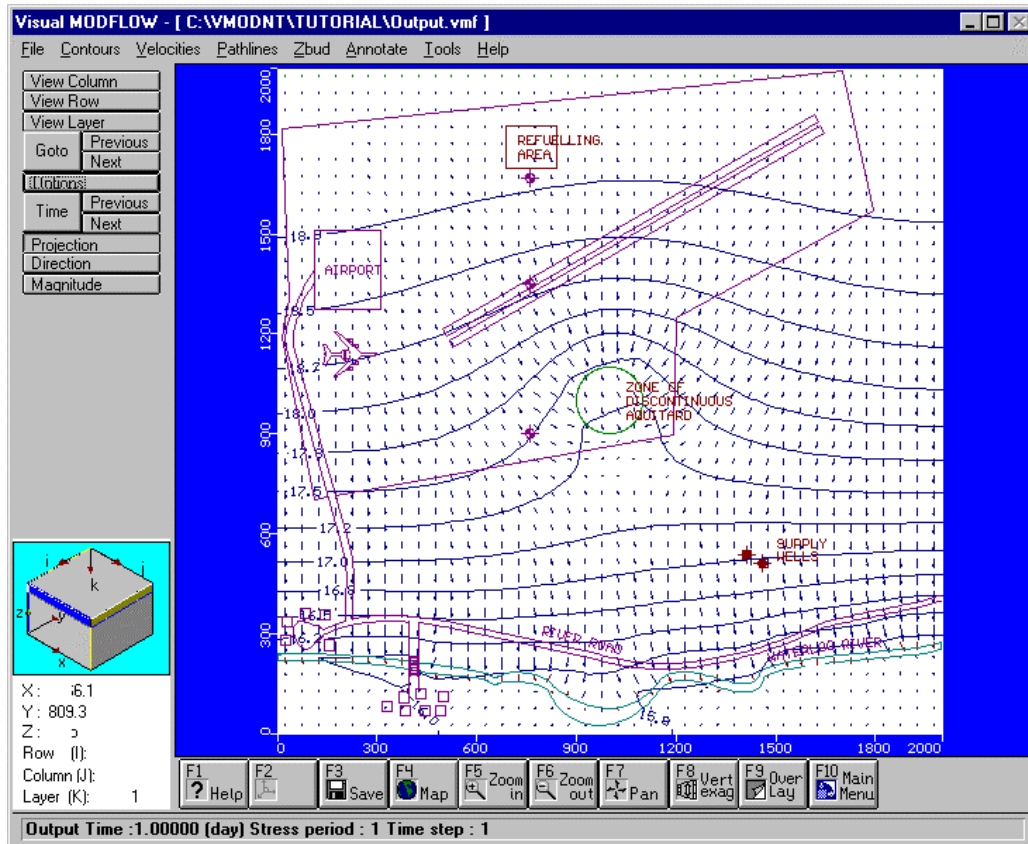
Vectors: 40

This specifies the number of velocity vectors along a single row.

Autoscale

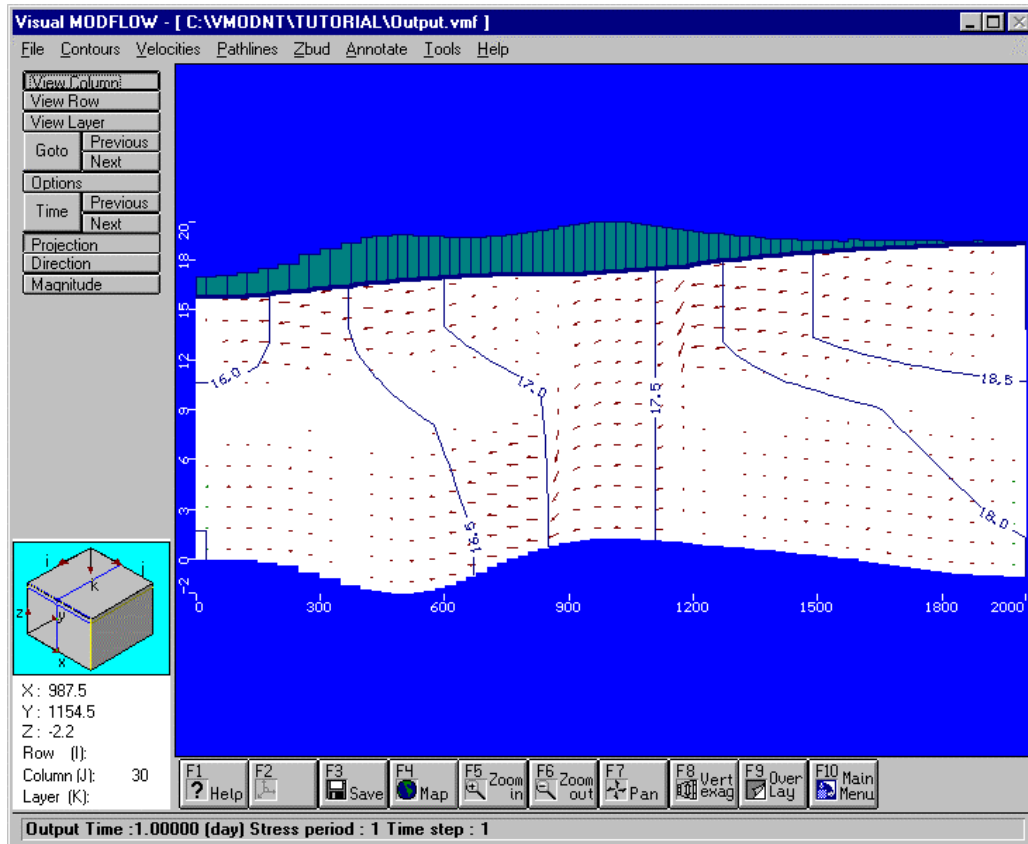
Variable Scale

☞ **[OK]** to accept the velocity vector settings



Notice the colour scheme: red is outward (i.e. up when viewing by layer), blue is inward (i.e. down when viewing by layer), and green is parallel to the plane (i.e. horizontal when viewing by layer).

To see the site in cross-section, select the [**View Column**] button from the left menu and then move the mouse pointer into the model domain and click once to select any column. A display similar to the following figure will appear showing both the equipotentials and velocity vectors in cross-section.

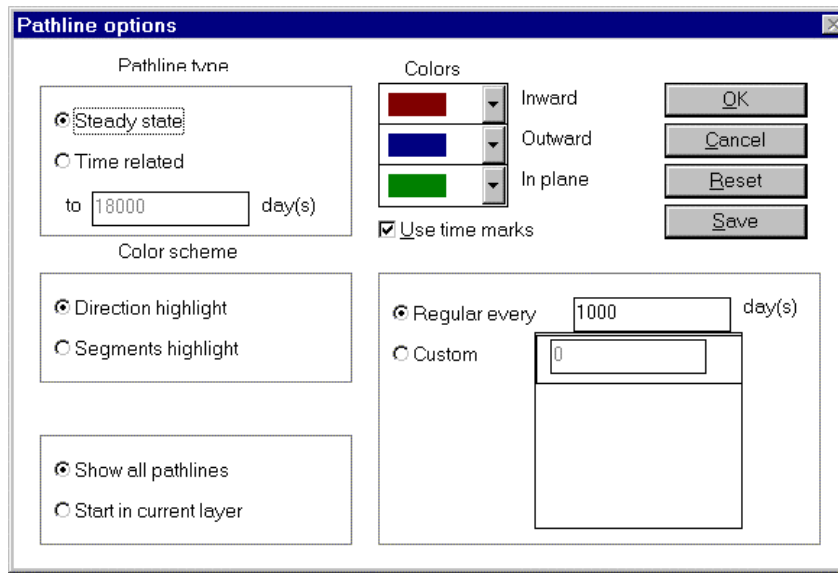


Return to the plan view display of the model by selecting the **[View Layer]** button from the left menu and then clicking on Layer 1 in the model cross-section.

SECTION 10: Pathlines and Options

To remove the velocity vectors from the screen display select the **[F9 - Overlay]** button from the bottom menu bar. An **Overlay Control** dialogue box will appear with an alphabetical listing all of the available overlays that you may turn on or off. Scan down the listing and locate the **Velocity Overlay**. Click on **Velocity Overlay** to highlight it, and then click on the button labelled **[ON]** to toggle it to **[OFF]**. This should remove the asterisk beside the **Velocity Overlay**, indicating that it has been deactivated.

particle reaches a certain destination. To determine the time interval for each time marker, select the **[Options]** button from the left menu bar and a **Pathlines Options** dialogue box will appear as shown in the following figure:



Notice that: **Use time marks** option is active. The time marker interval is indicated in the box labelled **Regular every 1000 days**

To see how far the pathlines would go in 10,000 days;

Under Pathline type:

☞ **Time Related** and enter a value of 10,000 in the box.

☞ **[OK]** to view the time-related pathlines up to a time of 10,000 days

Now view the pathlines in cross-section.

☞ **[View Column]**

Move the mouse pointer across the screen until the highlight bar is in the vicinity of the zone of discontinuous aquitard and click on it.

Now we will revert back to the plan view display.

☞ **[View Layer]**

Move the mouse pointer across the screen until the first layer is highlighted and click on it.

To remove the pathlines from the display, select the **[F9 - Overlay]** button from the left menu bar and scroll down the list of overlays until you get to **Particles Overlay** and **Pathlines Overlay**. De-activate both of these overlays by double clicking on them to remove the asterisk.

☞ **[OK]**

SECTION 11: MT3D Concentration Contours

☞ **C**ontours from the top menu bar

☞ **M**T3D Concentration

A plot of concentration contours will appear which plots the concentrations for the first output time of 1825 days.

You can customize the contour display as follows:

☞ **[Options]**

Choose the following settings if not already set:

- Automatic contour levels**
- Custom contour levels**
- Colour shading**

☞ **[OK]**

There were three output times specified for this MT3D simulation, 1825, 3650 days, and 7300 days. Select the **[Time]** button on the left menu bar to see a list of the output times.

☞ **3650** to highlight it

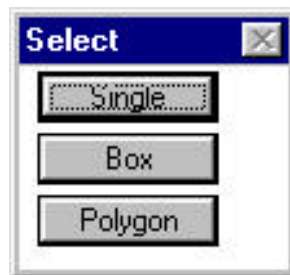
☞ **[OK]** to plot the concentration contours after 3650 days.

The concentration contours for the last output time can be displayed by selecting the **[Next]** button located next to the **[Time]** button. This will display the concentration contours after 7300 days.

Concentration breakthrough curves can also be plotted for individual or multiple observation points.

☞ **[Select]** from the left menu bar to select the observation wells

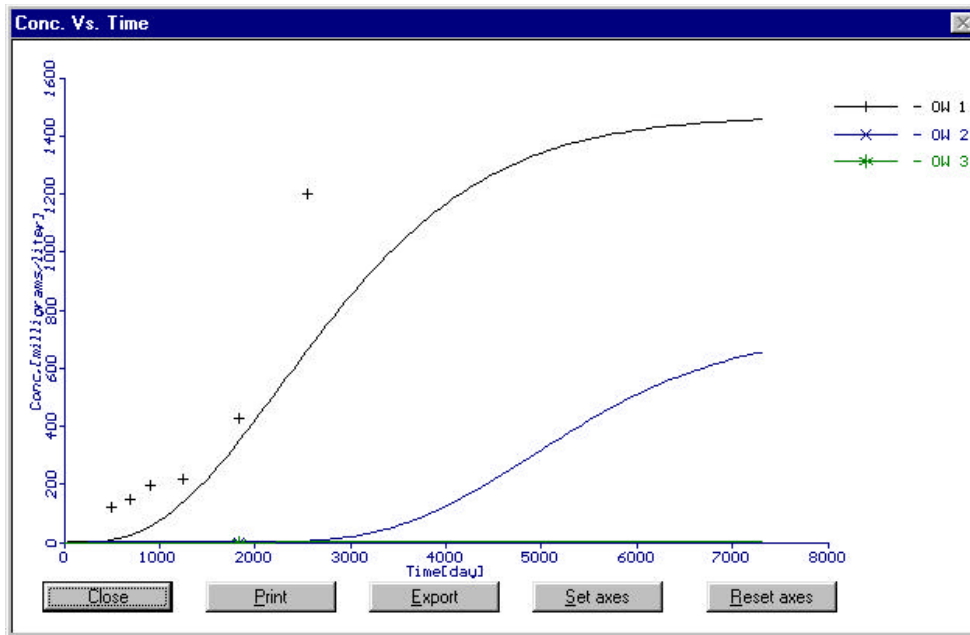
A pop-up window will prompt you to select the method by which you will select the observation wells (**[Single]**, **[Box]**, or **[Poly]**)



☞ **[Box]**

Stretch a box around the three observation wells south of the refuelling area by clicking the left mouse button once to anchor the box and again to close the box. The wells will turn red to indicate that they have been selected.

☞ **[Conc. vs. Time]** from the left menu



A graph of concentration versus time for each observation well will be displayed.

☞ **[Close]**

This concludes the Demonstration Tutorial. These instructions have provided you with an introduction to the main features and capabilities of Visual MODFLOW. If you have time, we would encourage you to go back and examine some of the other powerful features and analysis capabilities which were not covered by this tutorial. If you have any questions about the functionality, capabilities or features of this software, please do not hesitate to contact us at tech@gwsoftware.com.

☞ **[F10-Main Menu]**

☞ **File**

☞ **eXit (to exit Visual MODFLOW)**