

Groundwater Vistas Tutorial

Thank you for taking the time to try out our groundwater modeling software. Hopefully you will see how easy it is to create groundwater models in Windows using Groundwater Vistas. We recognize, however, that there is a lot of competition in the environmental industry for such software and we also know that there are other factors involved in your purchasing decision. Here are some factors we think may be important to you that set us apart from our competition:

Windows! We are committed to Microsoft Windows at every level. We are members of the Microsoft Developer Network and our software supports all versions of Windows from 3.1 through the new NT 4.0. For those of you still clinging to DOS, you should get used to the idea of Windows because Microsoft will no longer support DOS as you know it after the release of NT 4.0 this fall.

Support! One of the most important aspects of your purchase is the need for solid technical support from the software developer and excellent free and unlimited technical support is provided. Not only will we answer your questions related to the mechanics of using the software, but we will also help you with your project. Most environmental software companies limit support to running the software and will not venture into consulting. We are groundwater modeling consultants as well as software developers and will try to help you no matter what the question.

Updates! Most updates from are free and are distributed via Internet. These updates can be sent to you via email, Internet ftp, or through the mail. If you find a bug in the software, we will fix it immediately (usually the same day) and make it available to you on Internet as soon as the problem is fixed. If you do not have Internet access, we will send the repaired code to you via Federal Express at our expense.

Price! Our software is less expensive than most of our competition. We are also extremely competitive for volume discounts. Call 703-620-9214 to negotiate a multi-user license!

Training! We provide numerous on-site training courses to government and private industry each year. These seminars are hands-on courses in applied groundwater modeling using this software. On-site training is a very cost-effective way of providing professional development opportunities for your staff. For the cost of sending one or two staff to an outside short course, you can train a dozen or more staff in-house. Call for more information and a proposal to provide your organization with training in groundwater modeling.

I hope that we will have an opportunity to work with you on your groundwater modeling projects. If you have any questions regarding this software, support, consulting, or training, please do not hesitate to call or email to:

Scientific Software Group
P.O. Box 23041
Washington, DC 20026-3041
Phone: (703) 620-9214
Fax: (703) 620-6793

info@scisoftware.com

Welcome to the Groundwater Software Demos!

Thank you for your interest in Groundwater Vistas. This guide shows you how to install, explore, and uninstall demonstration versions of the following products:

Groundwater Vistas , a comprehensive modeling environment for Windows,	Page 4
MODFLOW^{win32} , version of the popular MODFLOW3D groundwater flow model,	Page 15
WinFlow , a 2D analytic element groundwater flow model, and	Page 20
WinTran , a 2D analytic element flow model coupled with a finite-element transport model.	Page 25

All of the demos are installed from one set of disks, as explained in the following section. After installing any or all demos, turn to the page listed above for a particular demo and follow the instructions to find out more about the software. Each demo is a fully functioning version of the software except that files cannot be saved and printing has been disabled. The last page of this guide contains an order form if you wish to purchase any of the software packages. If you have any questions as you are working through this guide, please do not hesitate to call us.

Contents and Installation

The demos come with a sophisticated installation program that is similar to other Windows products. You simply run “install” from the Program Manager **File** menu and follow the directions as the installation proceeds. Start by placing the first demo disk in drive A: (or B:). Now, select **Run** from the **File** menu in Program Manager (the Start menu in Windows 95) and enter the following:

```
a:install.exe <
```

You will first be prompted for the directory that will contain all of the demo software. The default is c:\esidemos. Next, select the Program Manager group that will contain the ESI icons. The default selection is “ESI Demos”. Finally, you will be given the option of installing each demo, starting with MODFLOW^{win32}. (**NOTE: you must install MODFLOW^{win32} if you intend to use Groundwater Vistas!**) When the installation program prompts for a particular demo, simply select Yes to install that demo or No to move to the next software package. Select CANCEL at any time to terminate the installation process. The installation program will now copy all of the MODFLOW^{win32} files into the directory you entered above. Next, the readme text file will be displayed to alert you to new information that was not available when the manual was created. Select OK when you are done reading the information. Although MODFLOW^{win32} was used in this explanation, all of the demos are installed with the same sequence of prompts.

If you have installed any of the demos, a dialog box will ask you to modify the *autoexec.bat* file on your computer. ESI Demos require that the environment variable **TMP** be set so that temporary files can be properly maintained. Modifying the *autoexec.bat* file here will properly set the **TMP** variable. Select YES to automatically make this change now or NO to change it yourself. If you do not set the **TMP** environment variable, the demos will not run. In this case, double-clicking on any of the demo icons displays a dialog asking you to set the **TMP** variable before the program can be executed. To set the **TMP** environment variable yourself, add the following line to your *autoexec.bat* file

```
set TMP=c:\tmp
```

(**Note: you will receive an error message that says “The DOS environment variable TMP must be set to a valid directory” if you do not place the above line in your autoexec.bat file**) Finally, a dialog box appears telling you to restart your computer to complete the installation. Please note, however, that this dialog only appears when you have changed the *autoexec.bat* file as discussed above. It will also not appear if you install Win32s below.

Installing Win32s

Groundwater Vistas, MODFLOW^{win32} and WinTran demonstration programs also need Microsoft’s Win32s software. This allows Windows V3.1 and V3.11 to run 32-bit software like MODFLOW^{win32}. If you are running Windows 95 or Windows NT, skip this step.

Installing Win32s will not harm your system in any way! In fact, it is only used if you run 32-bit applications. After you are done with the demonstration versions of ESI Software, you may delete it from your system, as described in the next section. In order to make it easier to get rid of Win32s, we suggest that you make a backup copy of your SYSTEM.INI file, located in your c:\windows directory. If you plan to purchase either Groundwater Vistas, MODFLOW^{win32} or WinTran, you may want to just leave Win32s on your system, as you will need it for these applications.

The Win32s installation is straightforward. If you have installed demos for Groundwater Vistas, MODFLOW^{win32} or WinTran, the installation program will ask you to install Win32s. Simply select Yes to install Win32s. Again, if you are running Windows 95 or NT, the installation program will not prompt you to install Win32s. If you are running Windows 95 or NT and you receive this prompt, then you made an error in selecting the operating system at the beginning of the installation. You should reinstall the software before continuing with the demonstrations.

Follow the prompts to install the Win32s software. After the installation is complete, you will be asked to restart Windows. You must do this in order to run the demos. **After restarting Windows, you must exit from Windows and reboot your computer so that the TMP environment variable will take effect.**

Uninstalling Demos

These demos come with an uninstallation program to make it easy to remove the software from your computer when you are done with your evaluation. To do so, simply double-click on the uninstall icon for the demo you wish to uninstall. You have 3 choices: (1) select *automatic* to quickly remove all of the demo files, (2) select *custom* to determine which files will be deleted, or (3) select *cancel* to end the uninstall program.

The uninstall program will not harm any program or file in your Windows directory or any other directory. Only files placed on your computer related to ESI demos will be removed. If you do not wish to use the uninstall program, simply delete the contents of the directory containing the demo software and delete the “ESI Demos” Program Manager Group in Windows.

Uninstalling Win32s

Unfortunately, Microsoft does not provide an uninstall program for Win32s; however, the process is fairly simple. This is especially true if you made a backup copy of your SYSTEM.INI file in the C:\WINDOWS directory. Follow these steps to remove Win32s from your computer:

If you made a backup copy of SYSTEM.INI and you have not installed any other software since installing the GV, MODFLOW^{win32} or WinTran demos, copy the old version of SYSTEM.INI back to your Windows directory (usually C:\WINDOWS) and skip to step 3 below.

(1) Remove the following line from the [386Enh] section in the SYSTEM.INI file:

```
device=<WINDOWS>\<SYSTEM>\win32s\w32s.386
```

where <Windows> and <System> are where the Windows and System directories are, respectively. This is usually C:\WINDOWS\SYSTEM.

(2) Modify the following line from the [BOOT] section in the SYSTEM.INI file:

```
drivers=mmsystem.dll winmm16.dll
```

to the following (remove winmm16.dll):

```
drivers=mmsystem.dll
```

(3) Delete the following files from the <WINDOWS>\<SYSTEM> subdirectory or from the SYSTEM directory in network installations:

W32SYS.DLL

WIN32S16.DLL

WIN32S.INI

(4) Delete all the files in the <WINDOWS>\<SYSTEM>\WIN32S subdirectory or the <SYSTEM>\WIN32S subdirectory in network installations. Then, delete the WIN32S subdirectory itself.

(5) Restart Windows

Groundwater Vistas (GV) Demo!

Groundwater Vistas is Environmental Simulations's new groundwater modeling environment for Microsoft Windows. Groundwater Vistas, or GV, is compatible with ModelCad³⁸⁶ from Geraghty & Miller because both programs were written by the same author. However, GV goes beyond the capabilities of ModelCad or any other model preprocessor. GV is designed to be a cost-effective alternative to more expensive programs, such as ModelCad, Visual MODFLOW, MODIME, PMWIN and GMS. The following are the unique features of GV that set it apart from other preprocessors:

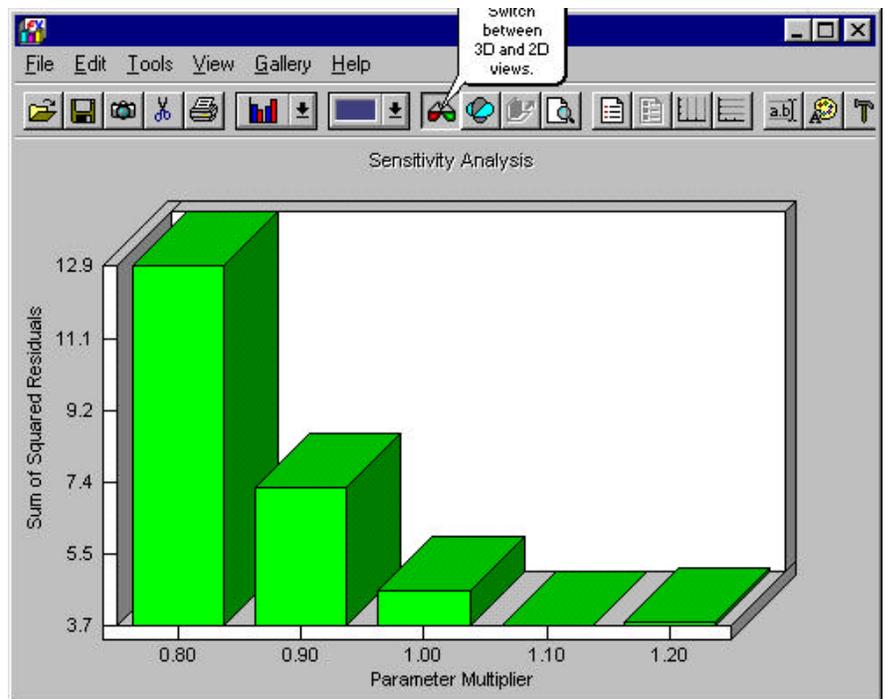
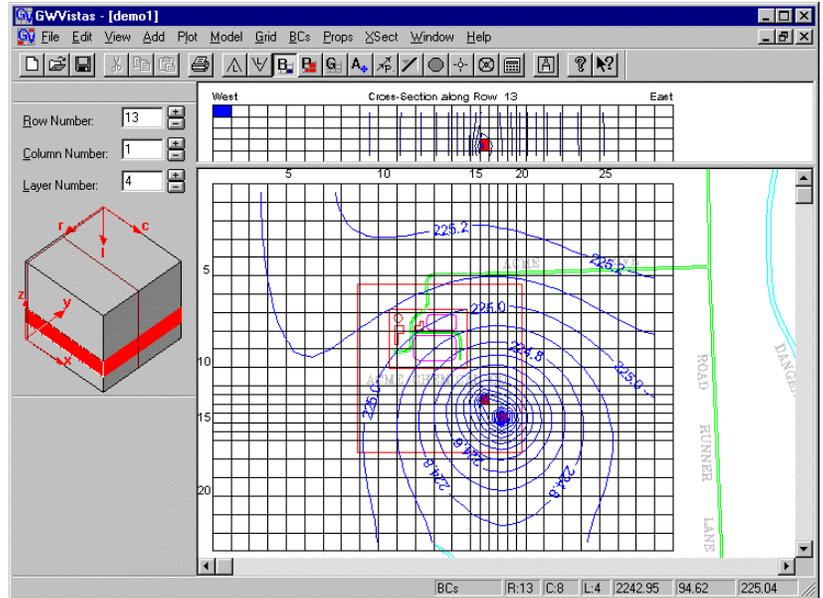
(1) Groundwater Vistas is designed specifically for Windows 3.1, Windows 95, and Windows NT (including the new Windows NT V4.0). The user interface follows Microsoft's conventions making the software easy to learn. The entire User's Manual is included as on-line, context-sensitive help.

(2) Groundwater Vistas is model-independent, supporting several different models including

- MODFLOW,
 - MT3D,
 - PATH3D,
 - MODPATH,
 - MODFLOWT (a new transport mode),
 - MODFLOW-SURFACT (a new version of MODFLOW), and
 - PEST, the model-independent calibration software.
- FTWORK, MODFLOWP, SUTRA and BIOPLUME II are also planned for the next release.

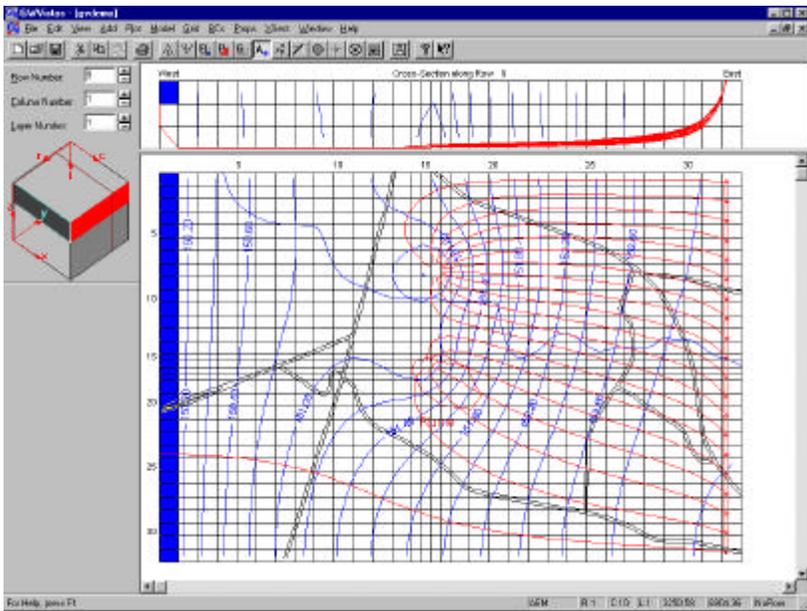
(3) Groundwater Vistas provides visualization of output from all supported models. This includes contouring, color flood contouring, velocity vectors, particle-trace display, and a detailed mass-balance analyzer (not found in the other competitors). All graphics are displayed in plan view and in cross-section at the same time using a split window. GV also plots head, drawdown, and concentration versus time at observation points, parameter sensitivity graphs, and profiles of head, concentration, drawdown, or flux along a cross-section.

(4) Boundary conditions may be specified either on a cell-by-cell basis or using grid-independent locations. The latter allows you to modify the grid without influencing the location of boundaries.



Groundwater Vistas

Advanced Model Design & Analysis for Windows!



Groundwater Vistas (GV) is a unique groundwater modeling environment for Microsoft Windows that couples a powerful model design system with comprehensive graphical analysis tools. GV, **developed by the author of ModelCad™**, is a model-independent graphical design system for MODFLOW and related models. In addition, GV supports the use of the PEST model-independent calibration software. The combination of PEST (a trial version is included with GV) and GV's automatic sensitivity analysis make GV a powerful calibration tool!

GV displays the model design in both plan and cross-sectional views using a split window (both views are visible at the same time). Model results are presented using contours, shaded contours, velocity vectors, and detailed analysis of mass balance. MODPATH particle traces are also displayed in both plan and cross-sectional views. Another unique aspect of GV is its use of grid independent boundary conditions. Grid-

independent boundaries do not change position as the grid is modified. This allows the modeler to make major changes to the mesh without wasting time repairing the location of boundaries. The following list summarizes more of GV's unique capabilities:

- ☞ Supports MODFLOW, MT3D, MODPATH, PATH3D, MODFLOWT, and MODFLOW-SURFACT
- ☞ Displays both plan and cross-sectional views
- ☞ **Run all supported models from within GV**
- ☞ Public domain versions of **MODPATH and MT3D included**
- ☞ Supports PEST model-independent calibration software
- ☞ Automated parameter sensitivity analysis
- ☞ **Imports existing MODFLOW & ModelCad files**
- ☞ Imports data from SURFER and ASCII files
- ☞ Export results to SURFER, Spyglass, DXF, and ASCII files
- ☞ Overlay maps in DXF and SURFER BLN formats
- ☞ Full on-line context-sensitive help
- ☞ WYSIWYG printing with print preview
- ☞ All Windows Versions supported! (3.1, 3.11, 95, and NT)

Types of Graphical Displays:

- ☞ Head, Drawdown, Concentration, Flux contours
- ☞ Head, Drawdown, Concentration, Flux color floods
- ☞ Velocity Vectors
- ☞ Pathline and travel times from MODPATH/PATH3D
- ☞ Local Mass Balance Bar Charts
- ☞ Plot head, drawdown, concentration versus time
- ☞ Parameter Sensitivity Plots
- ☞ Head, Drawdown, Concentration, Flux Profiles along a cross-section
- ☞ Calibration target scatter plots
- ☞ Calibration target hydrographs
- ☞ Calibration statistics for head, concentration, flux

Exploring Groundwater Vistas

The GV exercise, described below, introduces you to most of the important features of this software in a step-by-step example. You will be given very specific instructions to show how to use GV to design finite-difference models for MODFLOW. In a graphical user environment such as Windows, it is difficult to tell you exactly what to do during each step because many of the steps involve using the mouse. This demonstration provides several snap-shots of the GV screens to show you what your screen should look like, however, in case you miss a step. Note that the screen snap-shots were produced in Windows 3.1. Your screen will look slightly different if you are running Windows 95 or Windows NT. This demonstration also assumes that you have installed MODFLOW^{win32} in the same directory as GV and that the directory is C:\ESIDEMO.

The exercise below tells you to select menu items and a short-hand method is used to indicate what menu items are to be selected. The menu items are listed in order of selection and are separated by a “->” symbol. For example, instead of writing the following:

*Select **File** from the main menu and then select **Open** from the dropdown menu*

the following shortened version is adopted in this manual:

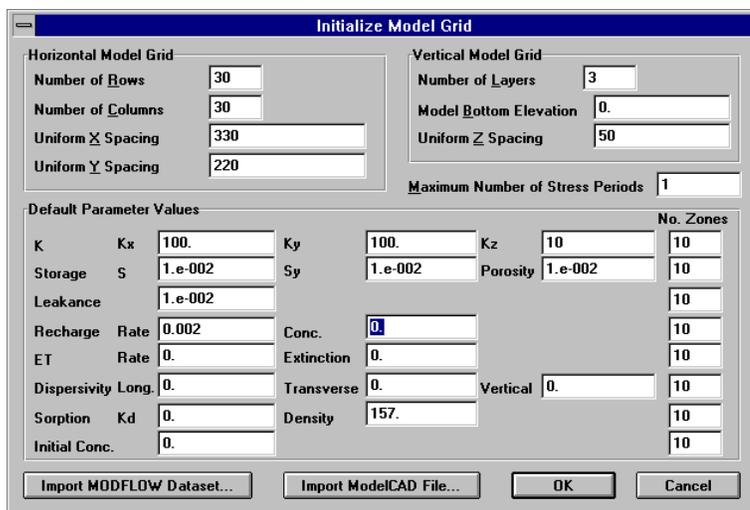
*Select **File->Open***

Some instructions for using the mouse should also be explained here, although most Windows users understand these concepts. The word “click” refers to pressing the left mouse button once and “double-click” means to press the mouse button twice in rapid succession. The word “drag” means to press the left mouse button and move the mouse to another location on the screen while keeping the button pressed down. Release the mouse button when the cursor is on the desired location.

Starting a New Model

We will start this exercise by showing you how to create a new model using GV. First, double-click on the GV icon to start the program. You will see a small menu over a blank model design window. Select **File->New** from the main menu or click the new

document button . You will see a rather large dialog on your screen that asks you for basic information describing your model. These data are used to construct the initial model, which will have uniform row and column spacings and uniform layer thicknesses. All aquifer properties (hydraulic conductivity, storage, etc.) will initially be uniform (homogeneous). For simple modeling studies, you only need to add boundary conditions in order to have a complete model ready to run with MODFLOW.



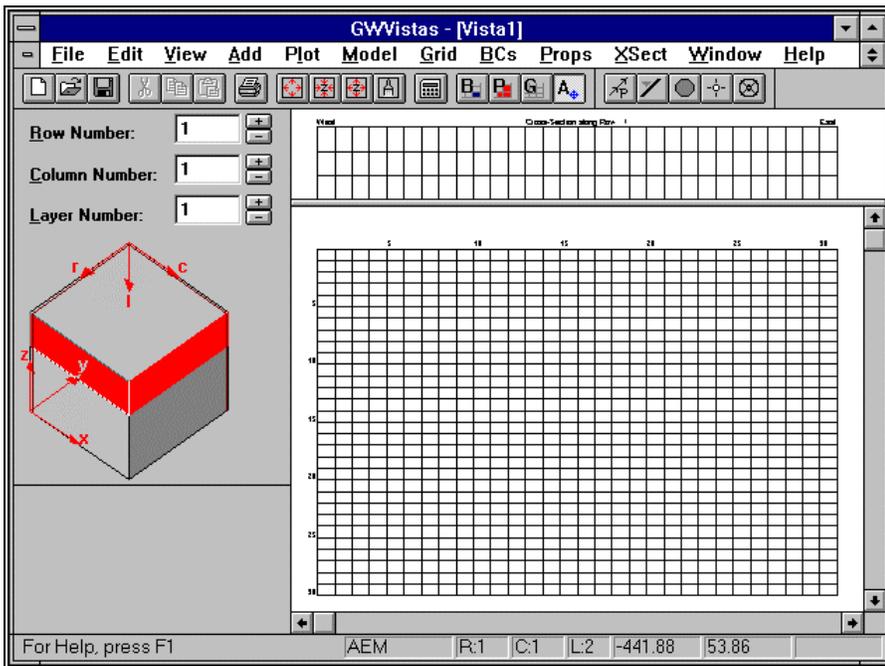
Default Parameter Values						No. Zones	
K	Kx	100.	Ky	100.	Kz	10	10
Storage	S	1.e-002	Sy	1.e-002	Porosity	1.e-002	10
Leakance		1.e-002					10
Recharge	Rate	0.002	Conc.	0.			10
ET	Rate	0.	Extinction	0.			10
Dispersivity	Long.	0.	Transverse	0.	Vertical	0.	10
Sorption	Kd	0.	Density	157.			10
Initial Conc.		0.					10

Now, fill in the dialog with the following information. When you are finished, press the OK button on the dialog. Before you choose OK, though, your screen should look like the one shown to the left.

Number of Rows	30
Number of Columns	30
X Spacing	330 ft
Y Spacing	220 ft
Number of Layers	3
Bottom Elevation	0.0 ft
Uniform Z Spacing	50.0 ft
Number of Stress Periods	1

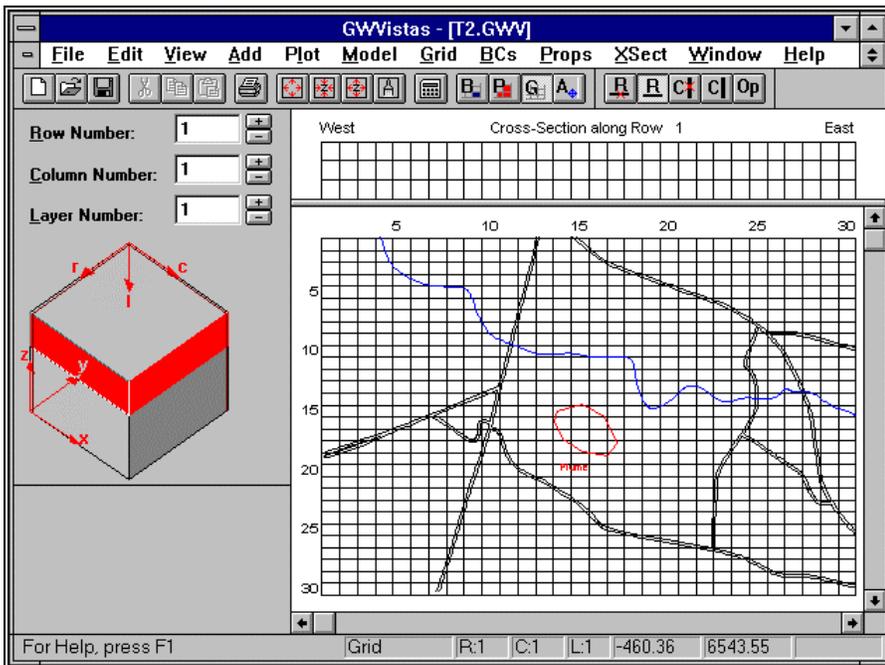
Change Kz to 10.0 ft/d
Change Recharge rate to 0.002 ft/d

After clicking the OK button, your screen should be similar to the one shown at the top of the next page. The model has uniform row and column spacings and the rows and columns are labeled.



Now, let's change the font size used for the row and column labels. All text used to annotate the GV model may be modified in terms of font style and size. To change the font for the row and column labels, select **Grid->Options** and click the font button. Change the font size to 8 points and click OK to return to the Grid Options dialog. Click OK again to return to the GV menu.

You will now import a base map to display with the model. Select **File->Map->GW Vistas**. A file open dialog will be displayed. Choose the map called "t2.map", which can be found in the demo directory (the default location is c:\esidemo). After importing the map, select **View->Full->Screen** to fit everything back in the GV window. Your screen should now look similar to the one shown below.



Adding Rows & Columns

GV has four different modes when designing the model. These include Analytic Elements, Grid, Boundary condition, and Property zones. The design operation that you may perform is determined on the Edit menu. Select Edit from the main menu. At the bottom of the pulldown menu you will see selections entitled Grid, Aquifer Properties, Boundary Conditions, and Analytic Elements. A check mark appears next to the option that is the current selection and the appropriate button is pushed down on the

toolbar. The  button represents Analytic Elements,  represents Boundary Conditions,  stands for Property Zones, and  represents Grid operations. The Grid option allows you to add, delete, and move rows,

columns, and layers. Aquifer Properties refers to assigning physical properties (e.g., hydraulic conductivity) to each cell in the model. Analytic Elements refers to the grid-independent boundary conditions in GV. You will see the buttons on the right side of the toolbar change depending upon which button is pressed down. This customization provides you with the most commonly used commands for each mode.

GV gives you the ability to insert, move, and delete rows, columns, and layers. In order to perform these operations, you must be in "grid" mode. This is accomplished by selecting **Edit->Grid** from the main menu or by pressing the  button on the toolbar. The word grid will appear in one of the panes of the status bar at the bottom of the GV window.

Once in grid mode, the cursor behaves differently than in other modes. When you are close to a row or column grid line, the cursor changes shape to either a left-right or up-down arrow. Pressing the left mouse button when this cursor appears allows you to slide the row or column line to a new position. You may not slide it beyond the adjacent row or column, however.

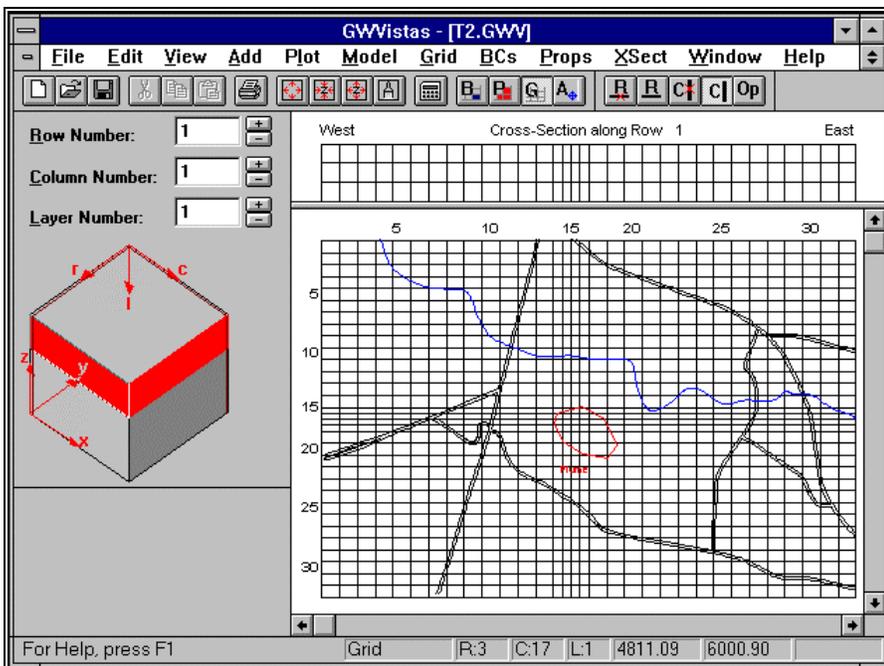
You may insert or delete rows, columns, and layers using the menu commands. These are fairly straight forward. Select the command (**Grid->Insert->Row** for example), move the cursor on the screen, and click the left mouse button. When deleting a row or column,

the row or column closest to the cursor is deleted. Layers may be added above or below the current layer (the current layer is displayed as L:1,2,3,... on the status bar).

The right mouse button has a special use in GV. When you are in Grid mode, the right mouse button inserts a row or column into the model or deletes the nearest row or column. The current grid operation (shown at the bottom of the Grid menu) determines what is added or deleted. To add rows or columns to the model, select **Grid->Insert Row** or **Grid->Insert Column** from the menu. To delete rows or columns from the model, select **Grid->Delete Row** or **Grid->Delete Column** from the menu. A check mark appears next to the type of action that GV will take when the right mouse button is clicked in Grid mode. The appropriate button is also pushed down on the toolbar ( to delete a row,  to insert a row,  to delete a column, and  to insert a column).

In this example, you will add two rows and two columns to the model. First, click the  button on the toolbar to enter Grid mode. Next, split row 15 into two new rows by placing the cursor anywhere within row 15 and click the right mouse button. Repeat this procedure for the next row to the south (Row 16 of the original model). When you insert a row or column, the default behavior is to split the current cell in half. You may change the way rows/columns are inserted by selecting **Grid->Options**.

Adding columns works the same way. Start by selecting **Grid->Insert Column** to place a check mark next to "Insert Column" on the Grid menu. Now split columns 14 and 15 just like you did for rows 15 and 16 above. Position the cursor within Column 15 and click the right mouse button. Repeat for the original column 16. Your screen should look like the one shown below:



Adding Boundary Conditions

You will now select Boundary Conditions as the current design mode so that you can add boundary condition cells to the model design. Select **Boundary Conditions** on the Edit pulldown menu. You may also click the toolbar icon .

In this example, you will add a column of constant heads along the left edge of the model in layer 1. You will then add two wells in the bottom layer (layer 3) of the model.

The easiest way to set a large number of boundary conditions is to use the Window command. Select **BCs->Insert->Window** from the main menu (or  from the toolbar). The cursor will change shape and appear like a mini-

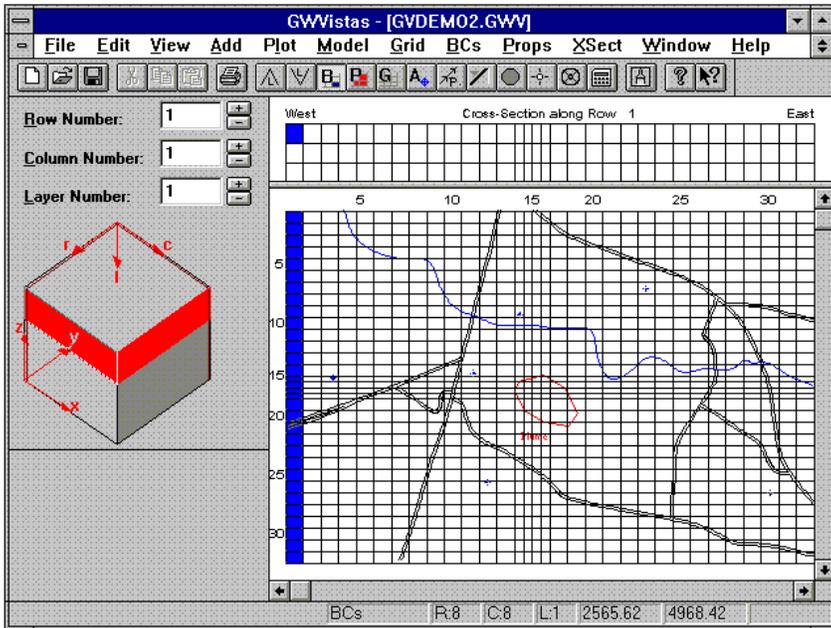
finite-difference grid. Move the cursor to the upper left corner of the model (row 1, column 1) and press the left mouse button. Hold the mouse button down and move the cursor to the lower left corner (row 30, column 1). Release the mouse button and a dialog appears to confirm the coordinates of the window that you just created. Simply press the OK button to accept these coordinates. Next, a constant head dialog appears. The only item that must be changed is the value of constant head. Change this value to 150 ft. Notice that by default the boundary condition is "steady-state". This means that the boundary cells will be active during the entire simulation. In MODFLOW, constant heads are active for the entire simulation and cannot be changed. Your screen should now look like the one shown on the next page.

Now, move to layer 3 of the model. The easiest way to change layers is by clicking the "+" button next to "Layer" on the 3D cube (called the Reference Cube) that is on the left side of the screen. Click the "+" button twice to get to Layer 3. The model will be redrawn and the constant head cells will disappear. This happens because these constant head cells were defined in layer 1 (the top layer) and we are now viewing the bottom layer of the model. You should still see one constant head in the upper left corner of the cross-section view however.

Select **BCs->Well** from the main menu. This places a check mark next to the word "Well" indicating that we are now editing wells. Next, select **BCs->Insert->Single Cell**. Move the cursor to row 16, column 15 and click the left mouse button. (You could also add a well by simply moving the cursor to row 16, column 15 and clicking the right mouse button.) You will notice that as you move the

cursor on the screen, the status bar at the bottom of the screen updates the current row and column. When you are at row 16, column 15 the status bar should read “R:16 C:15”. After clicking the mouse button, a well edit dialog will appear. (Note that in MODFLOW, negative flow rates are used for production and positive rates for injection). Enter a flow rate of -30000 ft³/d. GV assumes that units of length and time are consistent for all parameters. In this example, all length units are in feet and all time units in days. Therefore, hydraulic conductivity is in ft/d, well flow rates are in ft³/d, recharge is in ft/d, etc.

Add another well at row 8, column 16. This time, enter a flow rate of -40000. Use the same sequence of events that you used to insert the first well.



Creating MODFLOW Datasets

The example model is now complete. The aquifer properties have already been set from the initialization dialog because we are using a homogeneous system. You will now create a MODFLOW data set, run the simulation, and analyze the results.

The preprocessing of model-specific options is accomplished using the **Model** selection on the main menu. Select **Model** and you will see options including MODFLOW, MODPATH, MT3D, etc. Select MODFLOW from the dropdown menu to reveal another menu with numerous selections. You will select several of these options before creating the data files.

First, select **Model->MODFLOW->Packages**. The dialog displays all of the MODFLOW packages and

allows you to create only the ones necessary for the current model. The default selections do not normally need to be changed. The only change you need to make is the root file name. Change this parameter to “T2”. This means that the MODFLOW data files will be named T2.BAS for the BASIC package, T2.WEL for the well package, etc. Select OK when you are finished.

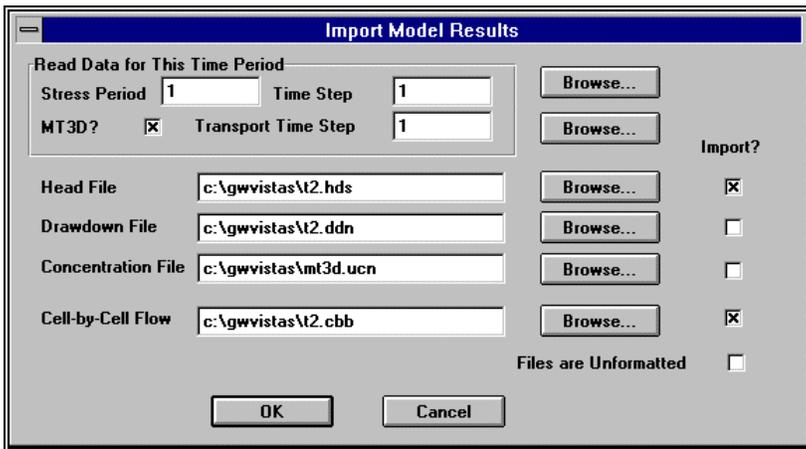
Choose **Model->MODFLOW->BCF Package** to edit the layer types in the Block-Centered-Flow (BCF) Package. Change layers 2 and 3 from type 3 to type 0 (confined) layers. Click OK when you are done. At this point, all of the model options have been selected. You will now run the model and view the results.

This is a good time to save your work. Choose **File->Save As** from the main menu. Use the file name T2.GWV for this first example. Click OK when you have entered the file name.

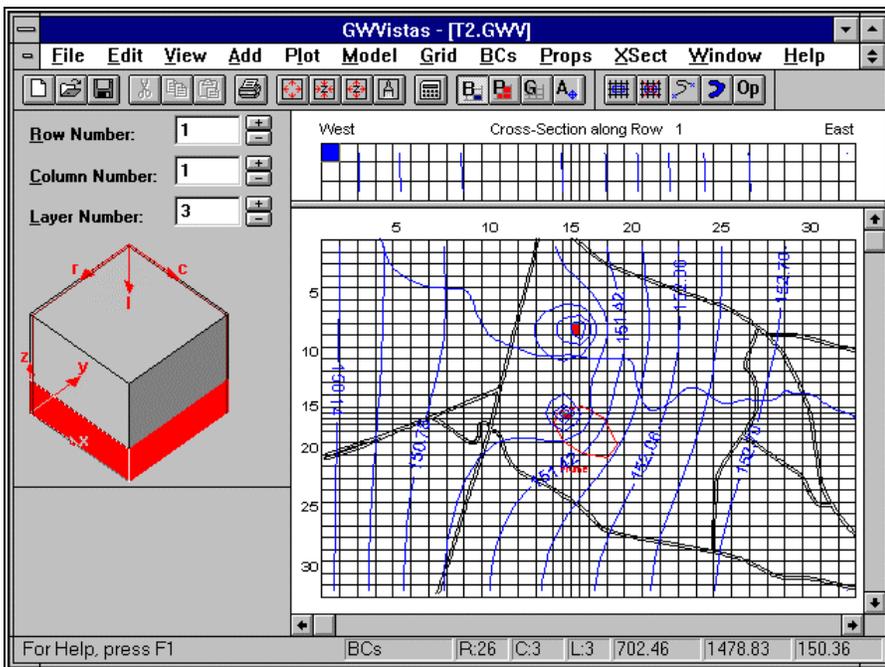
You are now ready to create data files for a MODFLOW simulation. The simplest way to create data sets and run the model is to click the calculator button on the toolbar. A progress dialog will be displayed briefly as the files are created. You will then be asked if you want to view the error file. Choose **Cancel** for this example as there should be no errors to view. After clicking Cancel, MODFLOW^{win32} will start to run.

NOTE: If MODFLOW^{win32} does not run, it means that GV could not find the proper model on your disk. To correct this problem, select **Model->Path to Models** from the main menu. Click the **BROWSE** button next to MODFLOW and use the file dialog to find the file called “for2cpp.dll”. For versions of MODFLOW other than ESI’s Windows version, find the executable model with the **Browse** button. If this does not solve the problem, then make sure that the MODFLOW^{win32} directory is in your **PATH** statement in the **AUTOEXEC.BAT** file on your hard disk.

When MODFLOW is finished, a dialog will be displayed notifying you that the simulation is done and asks if you want to process the results. Choose YES to start the post-processing session. You will next see a dialog that allows you to specify what model results to analyze (see the figure on the next page). The default settings allow you to read the head file from the MODFLOW run. We will also look at the mass balance results so place a check mark next to “CBC Files” as shown on the example dialog shown below. Click OK when you are done.

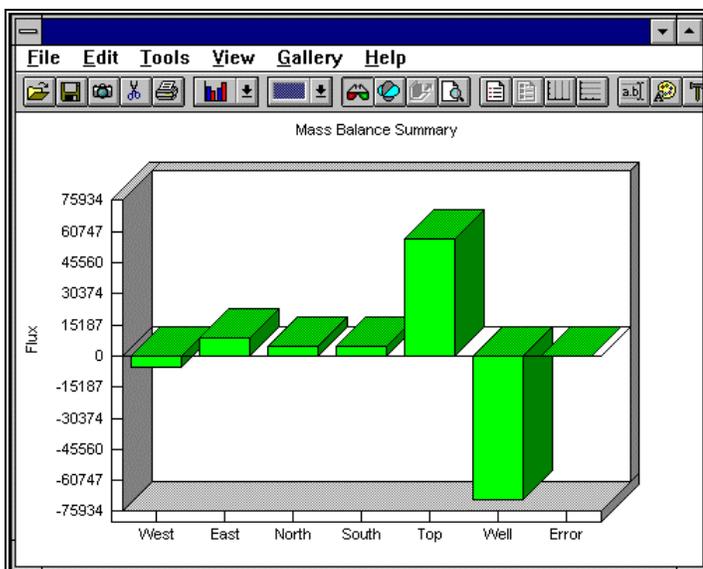


GV automatically contours the head results for the current layer and cross-section views. The resulting contours for layer 3 and for the cross-section along row 1 are shown below. Your screen should look similar, unless you have changed to another layer. You may contour any layer or cross-section by simply changing the settings on the reference cube. For example, if you click the “-” button next to Layer on the cube, the layer above the current layer will be contoured and displayed. Similarly, if you select a new cross-section, it will also be recontoured. You may plot velocity vectors on the map and cross-section by selecting **Plot->What to Display...** from the main menu and placing a check mark next to Vectors on the dialog. You may also produce a color flood map by placing a check next to “Color Flood”. You display all or none of these graphics using this dialog.



GV gives you full control over the contouring of MODFLOW results. You may change the contour interval, the font used to draw contours, the distance between contour labels, etc. These changes are made by selecting **Plot->Contour->Parameters (Plan)**. Change the starting contour level to 149.0 and the contour interval to 0.2 ft. Also, change the font size to 10 points. This is done by clicking the font button on the dialog. Click OK when you are done. A dialog will then tell you that the view should be recontoured. Click OK to proceed.

Other Types of Plots

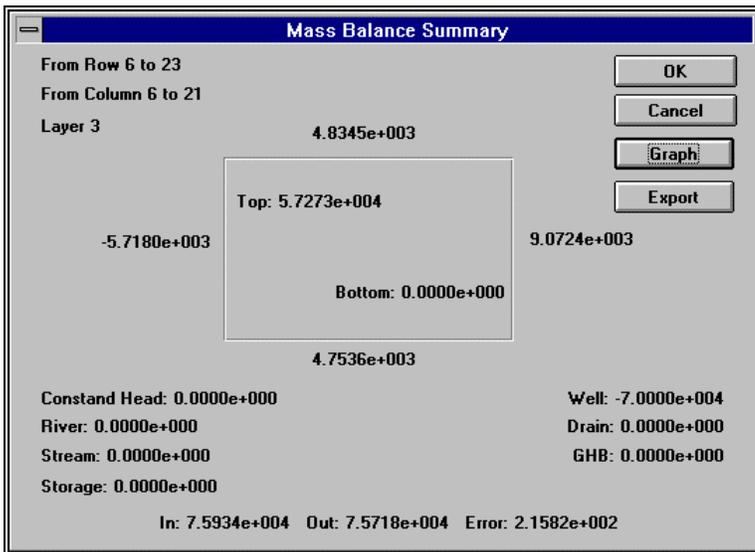


GV provides you with the tools to create many types of maps and graphs that are useful in analyzing model results. These are accessed from the **Plot** menu. We will explore a few of these plots now.

Select **Plot->Mass Balance->Window** from the main menu. Next drag a rectangular region on the screen. After you release the left mouse button, a dialog appears summarizing the flow of water into and out of the rectangular region you defined. The summary includes any boundary conditions that are within that rectangle in the current layer. An example is shown on the next page.

You may produce a bar chart of the mass balance results by simply clicking on the “Graph” button on the dialog. An example is shown to the left. Another useful mass balance feature of GV is that the flux rate of water into or out of a boundary cell is displayed on the status bar when the cursor moves over a boundary cell. Try this feature by moving the cursor over a constant in layer 1. The flow rate into the constant head for Row 1, Column 1 should be about -1561 ft³/d (NOTE: You must be in Boundary Condition mode to view fluxes; press the “B” button on

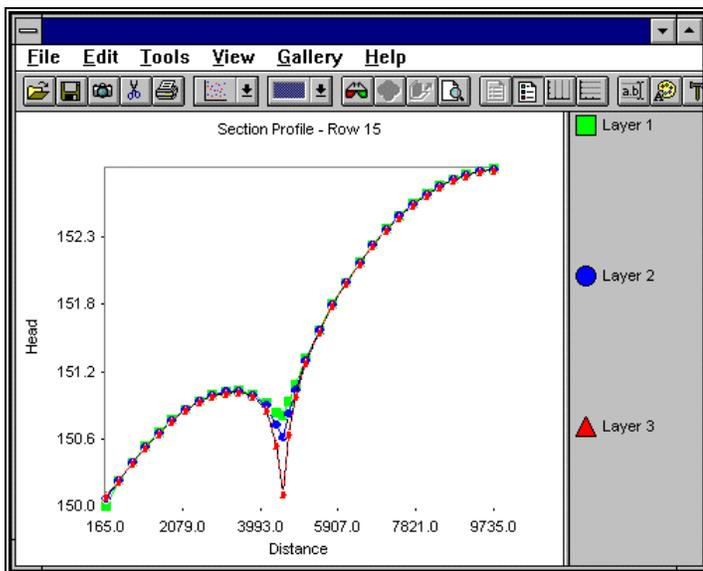
the tool bar to enter BC mode). The negative sign is the MODFLOW convention for water being removed from the aquifer.



Another graph that shows head relationships between different layers of the model is the “profile” plot. A profile is simply a graph of head, concentration, drawdown, or water flux plotted versus distance along the current cross-section. Select **Plot->Profile->Head** from the main menu to display a profile of head in your example model. An example profile plot through Row 15 is shown to the left. This plot shows the head relationships around a pumping well.

Particle-Tracking in GV

GV provides pre- and post-processing of particle tracking results for MODPATH (both the steady-state and transient versions) and for PATH3D. In the following example, we will use MODPATH. GV comes with two versions of MODPATH, the original USGS version that was limited to steady-state conditions and the newer version (Version 3.0) that works with transient models as well.



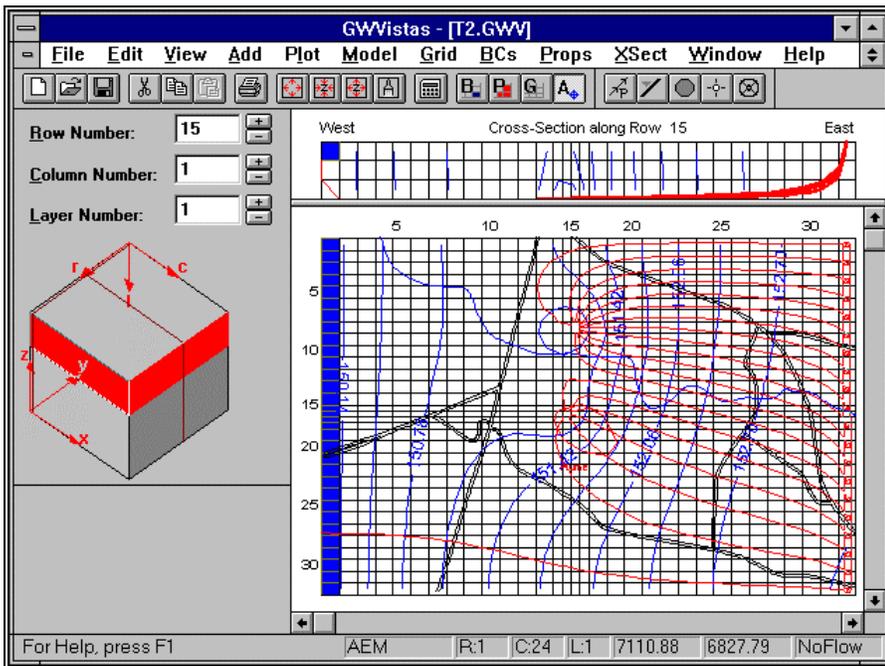
Particle tracking is a form of transport modeling in which only the bulk movement of groundwater is investigated. Particle tracking neglects the effects of chemical reactions, dispersion, and diffusion. The results of a particle tracking simulation are displayed by plotting pathlines through the aquifer system. Travel times are often labeled on the pathlines.

You start the particle tracking analysis by adding particle starting locations to the model. First, move to layer 1 by clicking the “-“ button next to layer on the cube until the number 1 appears in the Layer field. Select **Add->Particle->Line** from the GV main menu. Move the cursor to the upper right corner of the model (north-east corner). Click the left mouse button and drag a line to the lower right corner. Make sure the line of particles is within the bounds of the finite-difference grid. If the particles are outside the grid, they will not be tracked. Release the left mouse

button to display a dialog. Change the number of particles along the line to 20 and click OK.

Now, select **Model->Use MODPATH** to make MODPATH the current model. A check mark should appear next to “Use MODPATH” near the bottom of the Model menu. Click the calculator button on the toolbar and click the “Yes” button on the next dialog that asks to create MODPATH data sets. Select CANCEL from the next dialog that asks if you want to view the Error/Warning file. Now, MODPATH runs and produces results in a “pathline” file that GV will process. GV runs DOS and other console applications as minimized icons. You will see the MODPATH model as an icon when it is running and it will disappear when it is finished. Usually this does not take very long with MODPATH.

Finally, a dialog asks whether you would like to display MODPATH results. Select YES and then find the file called “pathline.ptl” in the subsequent “File Open” dialog. Click OK when this file is selected. GV will now read the file and display the pathlines in both plan and cross-section views. Your screen should look similar to the one shown on the next page.

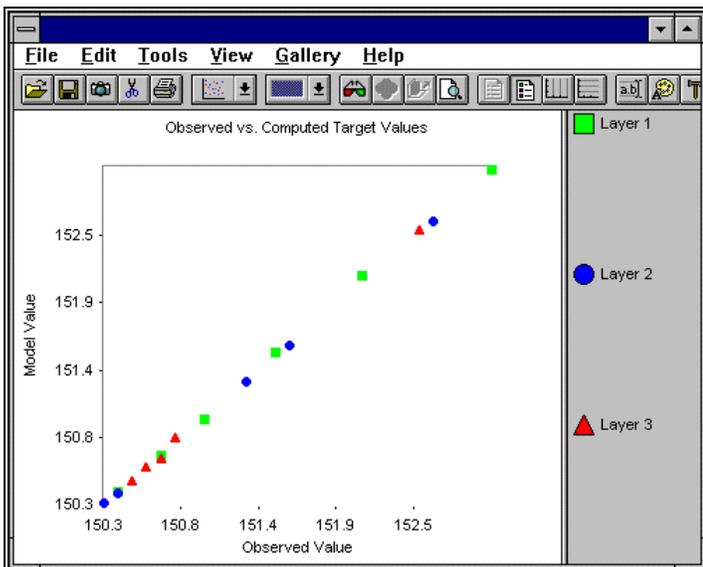


Model Calibration with GV

Calibration is one of the most complex parts of applying groundwater models. GV assists model calibration in three ways: (1) calculation of calibration statistics for head, drawdown, concentration, or flux, (2) automated parameter sensitivity analysis, and (3) support for the PEST model-independent calibration software. You start by adding calibration targets to the model. A calibration target is a point in the aquifer where a measurement of head, drawdown, concentration, or flux has been made. Calibration targets may be either steady-state or transient. When you run the model to compare against the target values, GV reads the model results and interpolates the model result in both space and time to compute an error or residual. Analysis of residual statistics is a powerful way of determining calibration quality and guiding further refinements to the model. The following exercise will illustrate the

calculation of calibration statistics and automatic sensitivity analysis. PEST support is not covered in this demo.

We will start by defining 16 head target locations in our example model. Rather than type in the data manually, you will import a text file containing the target data. GV provides many data import features for calibration targets, boundary conditions, aquifer properties, and base maps. Select **Plot->Calibration->Import Targets** from the main menu. Find the file called "targets.dat", which should be in the demo directory (default is c:\esidemos). Click OK when you have found it. GV will report the number of targets successfully imported. In this example, there should be 16. The targets will appear on the plan view as small blue dots. Targets are only displayed for the layer in which they are defined. You should see 6 in layer 1, 5 in layer 2, and 5 in layer 3. You may edit target information by double-clicking on a target symbol (You must be in Analytic Element mode, however; click the  button on the tool bar to edit targets in this manner).

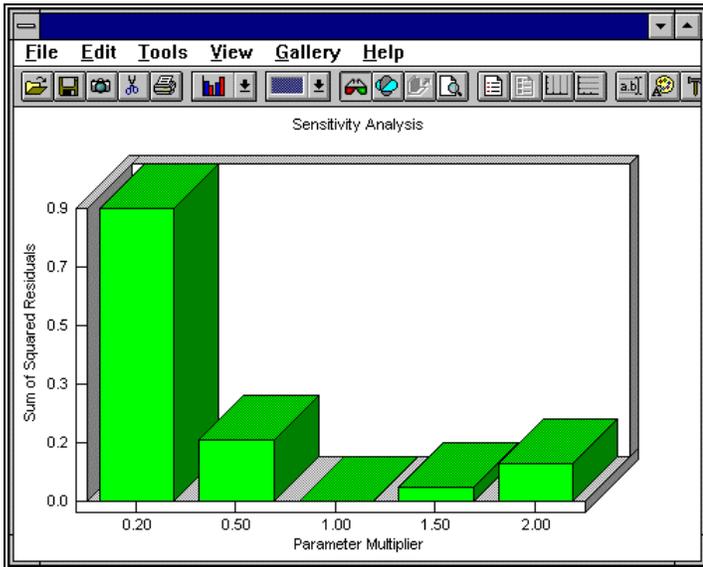


You will need to import model results again in order to compute the calibration statistics. Select **Plot->Import Results** and click OK on the dialog. All of the options should be set properly from the last run you made. You may view the calibration statistics by selecting **Plot->Calibration->Statistics/Plots...** A dialog is displayed that allows you to select the type of targets to use in the calculation (head, concentration, drawdown, or flux). You may also plot only selected ranges of layers. To view the statistics for this model, click the Statistics button. The residual sum of squares should be about $7.06E-03 \text{ ft}^2$. Click OK to leave this dialog. Now click the Plot button. A graph of observed vs. computed heads is displayed. Targets are color coded by layer. Your plot should be similar to the one shown to the left.

Another way to view the target residuals (errors) is to post them on the contour map. You do this by selecting **Plot->Calibration->Post Residuals**. Target residuals are posted when a check mark is displayed next to "Post Residuals" on this menu. In this example, the residuals are too small to read. You may change the font size by selecting **Plot->Calibration->Target Options**. Click the font button to change the size or font. Select **View->Refresh** to redraw the window with the new font.

Sensitivity analysis is an integral part of model calibration. Sensitivity analysis is the process whereby model parameters or boundary conditions are altered slightly and the effect on model calibration statistics is observed. By producing a series of simulations with different values for a single model parameter, you get a feel for how a parameter may be modified in order to achieve a better calibration. This is a tedious process because many simulations are required for each parameter and there are often many parameters

to analyze. GV provides you with an automated way of performing a sensitivity analysis that greatly improves the process. You simply choose a parameter type, the number of simulations, and a parameter multiplier for each simulation. GV then runs MODFLOW the desired number of times and produces a sensitivity plot. For each simulation, GV multiplies your initial parameter value by the multiplier you specify. After all of the simulations are finished, GV plots calibration statistics versus parameter multiplier to visually show the results of the analysis.



You start a sensitivity analysis by selecting **Model->Sensitivity Setup**. Select Kz as the parameter to vary. Keep the default of Zone number 1 and 5 simulations. Click on the “Multipliers” button. Enter the values 0.2, 0.5, 1.0, 1.5, and 2.0 for simulations 1 through 5, respectively. Click OK when you are done and click OK on the main dialog. Running the analysis is as simple as selecting **Models->Run Sensitivity** from the main menu. You will see MODFLOW flash on your screen five times and after the last simulation a dialog will ask whether you would like to see the results. Select “Yes” and you are asked to choose a variable to plot for each run. The choices include sum of squared residuals, residual mean, residual standard deviation, average drawdown, and total flux to a designated boundary condition reach. A plot of residual sum of squares vs. multiplier is displayed at the left for this example.

By producing a series of sensitivity plots, you can quickly determine the optimum parameter values to assign in your calibrated model. Of course, these parameters should be

reasonable values given your knowledge of the aquifer system.

Editing Aquifer Properties

Aquifer properties, such as hydraulic conductivity, are defined in GV using the “zone” concept (See the Help File topic entitled **Concepts** for a more elaborate discussion on zones). This means that you define a finite number of zones for each parameter and assign a zone number to each cell in the model. A zone number represents a fixed value for the parameter. When you first set up a model, every parameter is assumed to be homogeneous and every cell in the model is assigned a zone number of 1. For example, you entered an initial hydraulic conductivity value of 100 ft/d in the first GV dialog. GV assigns this value to zone 1 and then assigns zone 1 to each cell in the model.

GV displays zones using colors and fill patterns. To view and edit parameter zone values, select **Edit->Aquifer Properties** or click on the  button on the toolbar. Pull down the Props menu and you will see all of the available properties listed at the bottom of the menu. The property type with the check mark next to it is the one you are currently viewing and editing. Simply click on another property type to change the current property.

If the parameter zone is homogeneous, GV does not fill in the cells in your model. However, you may see the zone and property value assigned to cells by simply moving the cursor around the grid. The zone number and property value assigned to that number are displayed on the left side of the status bar at the bottom of the GV window. Do this now and you should see Zone:1 Kx=100.0. This means that the cells in you model are assigned a hydraulic conductivity (K) zone number of 1 and that zone 1 represents a K value of 100.0 ft/d.

We will now change the distribution of hydraulic conductivity in your model by first defining another zone value and then assigning this new zone to some cells in your model. The first step is to modify the database of zone numbers for hydraulic conductivity. Select

Props->Property Values->Database or press  on the toolbar. You will see a dialog that lists zone numbers on the left (NOTE: these are NOT layer numbers). There are three columns labeled Kx, Ky, and Kz. These are the three directional values of hydraulic conductivity. You should also see that zone 1 has been assigned values of 100.0, 100.0, and 10.0 for these three parameters, respectively. All other zone numbers in the database have values of zero. Now, change the Kx, Ky, and Kz values for zone 2 to 25.0, 25.0, and 2.5. Click OK to save these values.

By changing the value of the property assigned to zone 2, you have not changed the model at all because no cells are currently assigned zone 2! You have simply allowed for the possibility that a value of 25.0 ft/d may be assigned for hydraulic conductivity. To actually change certain cells to this new property value, select **Props->Set Zone Numbers->Window**. Now move the cursor to a location

within the model and drag a rectangle on the screen. Release the left mouse button and you will see a dialog asking for the zone number to assign to this region. Enter a value of 2 and click the OK button (or simply hit the Enter key). The screen will now be redrawn and you should see blue cells for the region covered by zone 1 and red for the region covered by zone 2. As you add more zone numbers, the colors change so that blue is assigned to zone 1 and red to the highest zone. A spectrum of colors is assigned between the two extremes. You may change the color and pattern assigned to each zone by selecting **Props->Property Values->Edit Zone Colors** and **Props->Property Values->Edit Zone Patterns**.

The Future of GV...

ESI is committed to making GV the most comprehensive and least expensive modeling platform in the industry. We have many enhancements planned for the near future. Many of these enhancements will be sent to GV users free of charge or distributed over Internet. The following are some of the new features that will be added over the coming months:

Telescopic Mesh Refinement (TMR)

TMR is the ability to create a new refined model covering a subregion of a larger model. The new model has smaller grid spacings and usually more nodes than the large-scale model. This technique is useful for performing transport modeling over a portion of the original model. The TMR technique applies boundary conditions from the regional model to the subregional model based upon the user's choice of constant head, constant flux, or head-dependent flux boundary conditions. These boundaries are interpolated from the larger-scale model automatically by GV. The small-scale model captures all of the complexity of the regional model and preserves the regional flow effects at the local scale. TMR is planned for release in GV by the end of September 1996.

3D Graphical Displays

In order to keep the price of GV as low as possible, ESI will not build three-dimensional visualization directly into GV. Rather, we intend to offer file export features in GV that will allow you to use whatever graphics program you wish. The supported visualization packages will include (from least to most expensive): *Slicer* by Fortner Research (formerly SpyGlass), *Environmental Workbench* by SESCO, and *EVS*. Many other packages can be supported using GV's generic 3D data export feature. These file export options are expected to be complete by the end of October 1996.

Stochastic Simulation

GV will be enhanced this fall to include pre- and post-processing capabilities for stochastic simulation using *Stochastic MODFLOW*. ESI, in cooperation with the author of *Stochastic MODFLOW*, will offer a free upgrade to GV for these stochastic enhancements and a fee for the purchase of *Stochastic MODFLOW for Windows*. You will be able to perform Monte Carlo simulations using both MODFLOW and MODPATH. These enhancements are planned for January 1997.

Automated Calibration Techniques

Groundwater Vistas already offers the most powerful set of calibration features of any modeling system. The most significant to applied modelers is the automatic sensitivity analysis as demonstrated above. The automatic sensitivity analysis will be taken one step further to include a nonlinear least-squares approach to model calibration. You will simply choose which parameters to estimate, set bounds for the estimated parameters, and GV will run MODFLOW, MODFLOWT, MODFLOW-SURFACT, or MT3D many times to perform the least squares analysis. While there are several such inverse models on the market (including PEST which comes with GV in a trial version), GV will be unique in providing you the ability to monitor the progress of the calibration and intervene during the process. Automated Calibration is planned for late 1996.

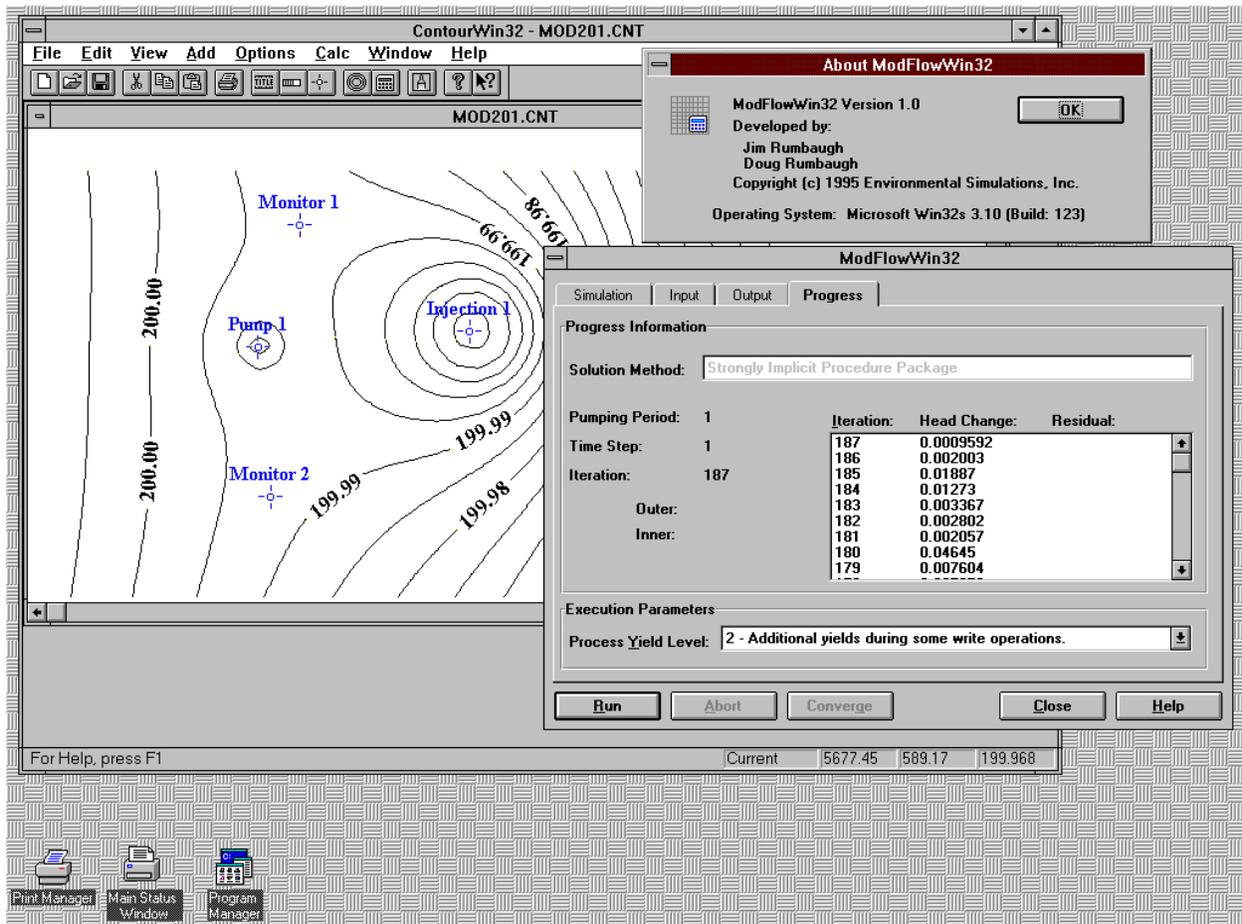
MODFLOW^{win32} Demo!

MODFLOW^{win32} is Environmental Simulation's version of the popular USGS groundwater flow model. It has all of the features of other MODFLOW versions, including the newest packages added over the years since MODFLOW's original release by the USGS. These new packages include the Stream Routing Package, Aquifer Compaction Package, Horizontal Flow Barrier Package, BCF2 and BCF3 Packages, and the new PCG2 solver. In addition, MODFLOW^{win32} will create files for use with MODPATH (a particle-tracking model) and MT3D (a solute transport model). MODFLOW^{win32}, as its name implies, is a 32-bit program designed to address all of the memory available to Windows. The software will run in all versions of Windows including Version 3.1, 3.11, Windows 95, and Windows NT.

MODFLOW^{win32} is a true Windows program; not just a DOS program that will run in Windows' DOS box. A comprehensive contouring program, called Contour^{win32}, is included with MODFLOW^{win32} to provide for post-processing of MODFLOW simulation results. Both head and drawdown may be contoured from any time-step, stress period, or layer. You may also overlay a base map in WinFlow, QuickFlow, ModelCad, and DXF formats. Particle-tracking results from MODPATH (not included with this MODFLOW^{win32}) may also be displayed on the contour maps. In addition, the finite-difference grid may be plotted over the contours.

MODFLOW^{win32} does not include any preprocessing software; however, you may purchase ESI's Groundwater Vistas preprocessing software at a special price when you also buy MODFLOW^{win32}. The cost of MODFLOW^{win32} bundled with Groundwater Vistas is half the cost of Visual MODFLOW and also much less expensive than PMWIN or ModelCad.

This demonstration version of MODFLOW^{win32} is fully functional with the following limitations: (1) MODFLOW has been limited to the size of the demo problem (about 2,700 nodes), and (2) Contour^{win32} will not print or save files. (Please note that you must be running Windows Version 3.1 or higher with at least 4 megabytes of RAM and 4 megabytes of disk space available.) We hope that this demo version will assist you in your decision to purchase MODFLOW^{win32}. If you have any questions, please feel free to give us a call.



MODFLOW Features

MODFLOW^{win32} is ESI's version of this popular groundwater flow model, originally developed by the USGS. MODFLOW is a 3D finite-difference groundwater flow model and there are many versions available. Unlike MODFLOW versions from other vendors, however, ESI's MODFLOW^{win32} was **developed specifically for Microsoft Windows (Win32s) and Windows NT**. Using a unique technology, MODFLOW^{win32} runs as a DLL which is controlled by a shell program or by ESI's Contour^{win32} which is provided with MODFLOW^{win32}.

MODFLOW^{win32} contains all of the popular add-on packages (BCF2, BCF3, PCG2, HFB, Compaction, Stream Routing) and is a 32-bit program utilizing all available memory. Contour^{win32} is a postprocessor for contouring MODFLOW results and displaying particle trace pathlines computed by MODPATH. The following are a list of capabilities for this unique software combination:

MODFLOW^{win32}:

- ☞ Runs 10 to 20 percent faster than other versions of MODFLOW for DOS!
- ☞ Contains the BCF2, BCF3, PCG2, STREAM Routing, and Aquifer Compaction Packages;
- ☞ User interface organizes file names and allows user to "abort" the simulation or force convergence when the solution oscillates;
- ☞ Uses all memory available to Windows, including virtual memory;
- ☞ Runs as a 32-bit DLL in Windows V3.1+, Windows NT, and Windows 95;
- ☞ Reads files created by all popular MODFLOW preprocessors;
- ☞ All MODFLOW manuals have been reformatted as Windows Help files;
- ☞ Provides several user-selectable levels of multi-tasking, even under Windows V3.1;
- ☞ Iteration data displayed as MODFLOW runs.

Contour^{win32}:

- ☞ Included with MODFLOW^{win32}!
- ☞ Displays contours of head and drawdown for any layer and for subregions within the model;
- ☞ Overlay base maps in DXF, ModelCad, or WinFlow format;
- ☞ Contour head and drawdown while MODFLOW^{win32} is running;
- ☞ WYSIWYG printing with print preview and user-defined margins to any Windows device;
- ☞ Export contour maps in Spyglass, Geosoft, SURFER, DXF, and Windows Metafile formats;
- ☞ Displays particle traces from MODPATH;
- ☞ Displays finite-difference grid;
- ☞ Annotate maps with scale bar, titles, arrows & travel times on particle traces;
- ☞ Computes calibration target statistics and post residuals (errors) on map.

Exploring MODFLOW^{win32}

The MODFLOW^{win32} demonstration, described below, introduces you to most of the important features of this software in a step-by-step example. You will be given very specific instructions to show how to use MODFLOW to solve real-world problems. In a graphical user environment such as Windows, it is difficult to tell you exactly what to do during each step, however, because many of the steps involve using the mouse. This demonstration provides several snap-shots of the MODFLOW^{win32} and Contour^{win32} screens to show you what your screen should look like.

The exercise below tells you to select menu items and a short-hand method is used to indicate what menu items are to be selected. The menu items are listed in order of selection and are separated by a “->” symbol. For example, instead of writing the following text:

*Select **File** from the main menu and then select **Open** from the pulldown menu*

the following shortened version is adopted in this manual:

Select File->Open

Some instructions for using the mouse should also be explained here, although most Windows users understand these concepts. The word “click” refers to pressing the left mouse button once and “double-click” means to press the mouse button twice in rapid succession. The word “drag” means to press the left mouse button and move the mouse to another location on the screen while keeping the button pressed down. Release the mouse button when the cursor is on the desired location.

Running a MODFLOW Model

We will start this demonstration by showing you MODFLOW^{win32} and Contour^{win32} separately. The final section will illustrate how the two programs work together. To start, double-click on the MODFLOW^{win32} icon now. You will see a tab dialog interface with the following sections or tabs:

- Simulation - reads in a MODFLOW^{win32} simulation input file (containing all data set names),
- Input - allows you to specify the names of MODFLOW input files (Basic Package etc.),
- Output - specify output file names (Head-save, cell-by-cell, etc.),
- Progress - displays iteration and error summaries as the model runs, and
- Head/Drawdown - communication with Contour^{win32}

You will select “**Input**” by clicking the mouse on the **Input** tab. Click the **Input File** button to start entering file names for a MODFLOW simulation. Select T1.BAS from the file list and click OK. This is the way you will start MODFLOW^{win32} using existing MODFLOW data sets. Now, click the **Run** button at the bottom of the MODFLOW^{win32} dialog. You will be asked to provide file names for packages and output files needed for the simulation. Simply select the only file name given (T1.*), in the order below:

T1.BCF	- Block-Centered Flow Package
T1.WEL	- Well Package
T1.RIV	- River Package
T1.RCH	- Recharge Package
T1.SIP	- SIP Solver
T1.OC	- Output Control

After the output control file, you must enter two additional files for computed head and drawdown. These files do not exist, so no file names appear in the dialog. Simply type the following names in the order shown below:

T1.HDS	You will type in this name for head output for contouring
T1.DDN	You will type in this name for drawdown output for contouring

After entering the drawdown file name (T1.DDN), click the Progress tab to see how the simulation progresses. This particular run consists of 2 stress periods with 20 time steps each. The model is relatively small so the simulation should not take long. When finished, your screen should look like the screen shown below:

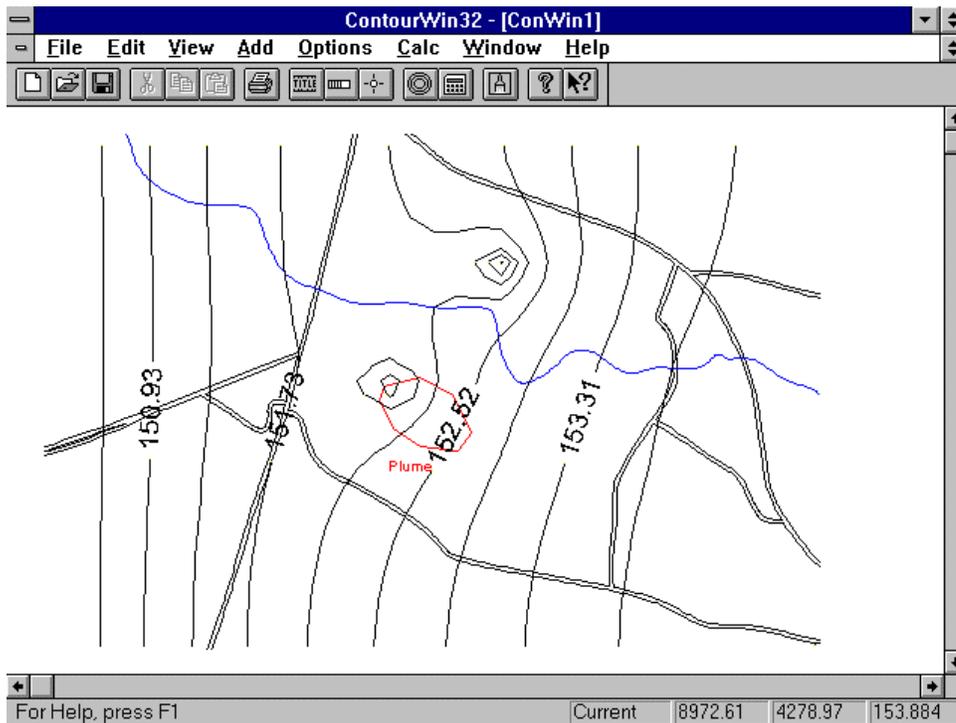
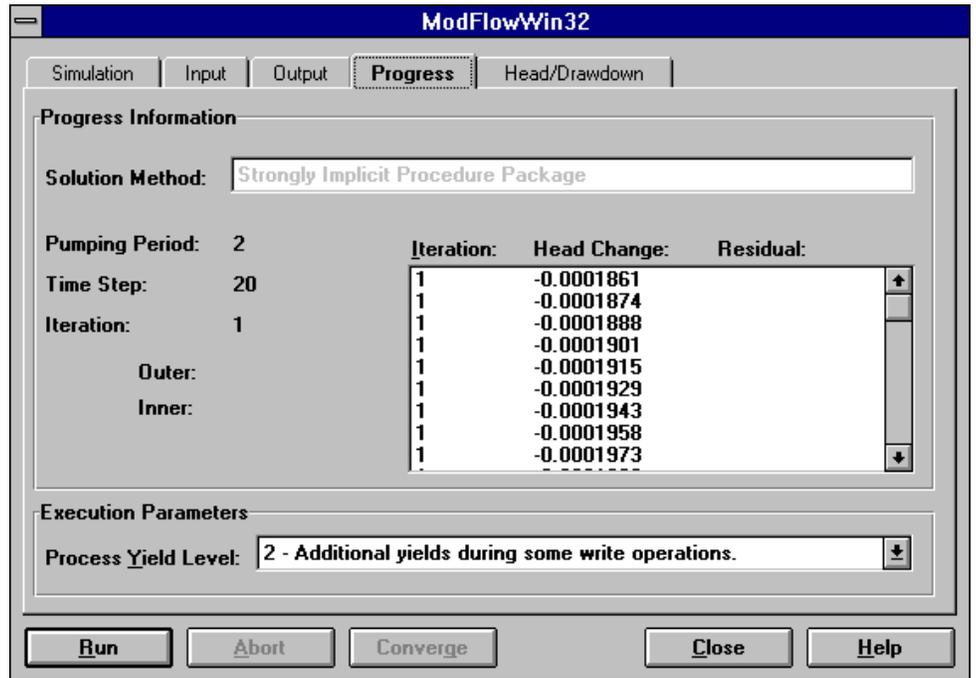
Now, you will save this simulation in a MODFLOW^{win32} file for processing by the contouring program. Click the **Simulation** tab and then click the **Save As** button. Enter a file name of T1.MFW. Click the **OK** button when finished. Finally, click the **Close** button at the bottom of the dialog to end the MODFLOW session.

Contouring the Results

We will now show you the procedure for contouring MODFLOW^{win32} output. Double-click the Contour^{win32} icon. Start a new map by selecting **File->New** from the menu. You may also press the



button on the toolbar below the menu. To load a set of heads or drawdowns from the MODFLOW simulation, select **File->Model** and choose the T1.MFW file. Click OK when you are finished selecting the MODFLOW^{win32} file. After a few seconds, a list of heads and drawdowns are presented showing the variable type (Head or Drawdown), Stress Period number, Time Step number, and Layer number. Choose Heads in Stress Period 2, Time Step 20, Layer 3 and click the OK button. Head contours will be displayed on your screen for the selected time and layer.



may change the contouring parameters by selecting **Options->Contour->Parameters** from the menu. Change the minimum contour level to 150.0 ft and the contour interval to 0.2 ft. Click **OK** when you are finished. A message will then appear asking you if you want to recontour using these parameters. Select **Yes** to do this.

Overlay a digitized base map by selecting **File->Map->WinFlow** from the menu. Select T2.MAP and click **OK**. A further dialog is displayed asking for coordinate transformation details. Simply click **OK** to accept the defaults (If you are using ModelCad, you can enter the offsets and rotation to adjust the map to the model grid using this dialog). Your screen should look like the one shown at the left.

Contour^{win32} can plot the finite-difference grid on the contour map. Select **Options->Grid** from the menu. Click **Display Numbers** for both rows and columns. Click **Display Grid** to plot the grid on the screen. Finally, change the Interval to 5 so that row and column numbers are labelled every 5 cells. Click **OK** when you are done and your screen should look like the one at the top of the next page:

By default, Contour^{win32} selects a contour interval to show 10 contour levels. You

Although MODPATH is not part of the MODFLOW^{win32} package (a version is included with Groundwater Vistas, however), Contour^{win32} will display MODPATH results and MODFLOW^{win32} output files are compatible with most MODPATH versions from other vendors. We have provided a MODPATH output file with this demo to show you how to process particle-traces.

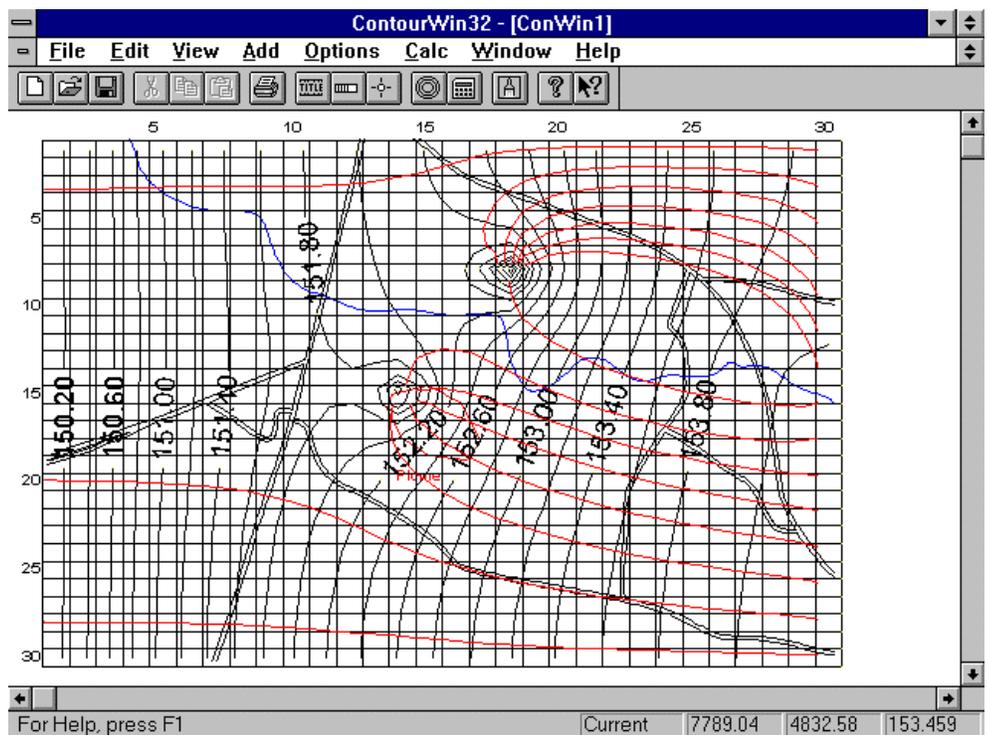
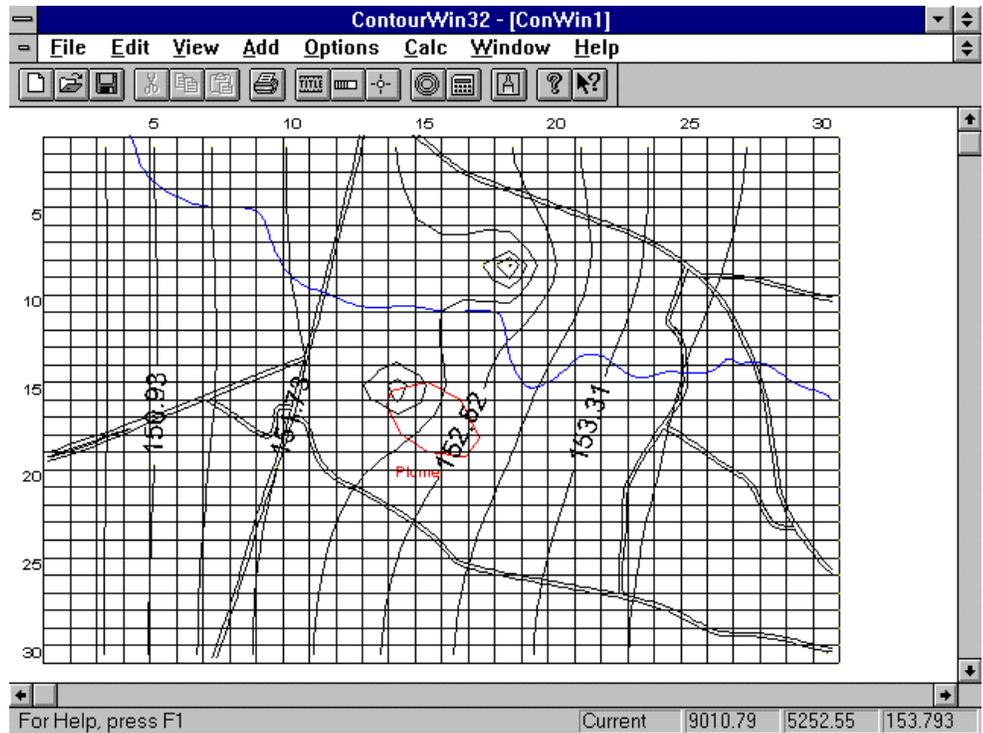
Select Options->Particle from the menu. Click the Browse button to show a list of files and select the file called PATHLINE.PTL. Also, click the "Display traces" button on the dialog after selecting the file. Click OK when you are finished. Your screen should look like the one shown below:

Particle traces may be annotated with travel time and arrows. Select Options->Arrow from the menu and click "Display Arrows" on the dialog. Click OK when you are done. This causes arrow-heads to be displayed on the traces. You can add time postings using Options->Time.

Contouring While MODFLOW is Running

We will now show you how MODFLOW^{win32} and Contour^{win32} can work together. Under all versions of Windows, the two programs (MODFLOW and Contour) are totally linked. MODFLOW^{win32} can start the Contour^{win32} program and vice versa.

Double-click the MODFLOW^{win32} icon to run the model again and click the Open button in the MODFLOW dialog. Select the T1.MFW file and click OK. Start the model simulation by clicking the Run button. Now click the **Head/Drawdown** tab to see heads and drawdown data written to the output files. When some appear on the screen, select one by clicking on the line containing the desired variable, stress period, time step, and layer number. Choose any one and click the Contour button. This causes MODFLOW^{win32} to run Contour^{win32} and pass the data to Contour^{win32}, which will open a new document and contour the data. You should see contours being displayed behind the MODFLOW^{win32} dialog. Press ALT-TAB to see the entire map and ALT-TAB again to get back to MODFLOW^{win23}.



WinFlow Demo!

WinFlow is a powerful yet easy-to-use groundwater flow model. The model is similar to Geraghty & Miller's popular *QuickFlow* model, which was developed by Jim Rumbaugh, one of the authors of WinFlow. The most notable improvement over *QuickFlow* is compatibility with Microsoft Windows™ V3.1. It is not just a DOS program that happens to run under Windows in a DOS shell. WinFlow is a true Windows program incorporating a multiple document interface (MDI). The following discussion explains the basic features of WinFlow, how to install the demonstration version of the software, and the many differences between WinFlow and *QuickFlow*.

WinFlow is an interactive, analytical model that simulates two-dimensional steady-state and transient ground-water flow. The steady-state module simulates ground-water flow in a horizontal plane using analytical functions developed by Strack (1989). The transient module uses equations developed by Theis (1935) and by Hantush and Jacob (1955) for confined and leaky aquifers, respectively. Each module uses the principle of superposition to evaluate the effects from multiple analytical functions (wells, etc.) in a uniform regional flow field.

The steady-state module simulates the effects of the following analytic elements in two-dimensional flow: *wells, uniform recharge, circular recharge/discharge areas, and line sources or sinks*. Any number of these elements may be added to the model, including a uniform regional hydraulic gradient. The model depicts the flow field using streamlines, particle traces, and contours of hydraulic head. The streamlines are computed semi-analytically to illustrate ground-water flow directions. Particle-tracking techniques are implemented numerically to compute travel times and flow directions. **Both confined and unconfined aquifers are simulated** with the steady-state module.

The transient module simulates the effects of wells, circular ponds, linesinks, and a uniform regional gradient for confined and leaky aquifers. Numerical particle-tracking is also available in the transient module. The transient module computes hydraulic heads using the Theis (1935) equation for confined aquifers and the Hantush and Jacob (1955) equation for leaky aquifers.

WinFlow is simple to use and highly interactive, allowing you to create an analytical model in minutes. The software features standard Windows pulldown menus and dialogs to facilitate the model design. The model is recomputed and recontoured either by selecting a menu item or by pressing a toolbar button. Streamlines and particle-traces are added interactively and recomputed each time new wells or other elements are added.

WinFlow can import a Drawing Interchange Format (DXF) file (from AutoCAD for example) to use as a digitized base map. *QuickFlow* and *ModelCad*-format map files may also be imported into WinFlow. The digitized map gives the modeler a frame of reference for designing the analytical model.

WinFlow produces report-quality graphics using any Windows device driver. Output may also be exported to a wide variety of file types, including SURFER, Geosoft, Spyglass, Windows Metafiles, and AutoCAD-compatible DXF files.

WinFlow Features

The WinFlow demonstration version is fully functional, with the exception that models cannot be saved or printed. All other features are functional, including the on-line help system which contains the entire WinFlow manual. Press F1 at any time to view the Help file. WinFlow has the same basic capabilities as Geraghty & Miller's *QuickFlow* software, although many features are different or enhanced. The features that WinFlow has in common with *QuickFlow* include the following:

- ☞ Simulates both steady-state and transient flow;
- ☞ Simulates both unconfined and confined aquifers;
- ☞ Simulates effects of wells, linesinks, ponds, and recharge;
- ☞ Imports map files in DXF format, QuickFlow format, or ModelCad format;
- ☞ Visualizes model results with water-level contour maps;

- ☞ Illustrates groundwater flowpaths using streamlines and particle traces;

The most notable enhancement is that WinFlow is a standard Windows MDI (Multiple Document Interface) application. Experienced *QuickFlow* users should scan the following list to get a feel for these enhanced features:

- ☞ Linesinks and ponds included in the Transient model;
- ☞ Calibration targets and calculation of calibration statistics;
- ☞ Each analytic element may have a title with full font selection;
- ☞ Double-click an element to edit;
- ☞ Click and drag to reposition elements, streamlines, or particles;
- ☞ Click and drag to resize linesinks and ponds;
- ☞ Multiple document interface (MDI) allows multiple models to be open at the same time;
- ☞ Cut, copy, and paste elements to/from the clipboard;
- ☞ Maps may be printed using any Windows device driver;
- ☞ Coordinates and head are displayed as the cursor is moved;
- ☞ Full context-sensitive help system (the entire manual is on-line);
- ☞ DXF file import from within WinFlow;
- ☞ Common commands are available on the Toolbar; and
- ☞ Drag-and-drop input files into the WinFlow window.

Exploring WinFlow

After installing the demo version of WinFlow, you have several alternatives for exploring its features. The first is to run through the demonstration described below. This describes most of the basic WinFlow features. Another option is to run the example WinFlow data sets contained on the demo disk. Select File from the main menu and Open from the dropdown menu. You should see three files (ex1.wfl, ex2.wfl, and ex3.wfl). Select any of the three examples and choose OK. Use the on-line help and your own intuition to explore from here (Press F1 at any time for help). The final alternative is to load existing QuickFlow files. This is done by selecting File and Open from the main menu. Next, select QuickFlow under the button labeled "List Files of Type" (NOTE: your QuickFlow files must end in the extension qfl).

Introduction

The WinFlow demonstration guide introduces you to most of the important features of this software in a step-by-step example. You will be given very specific instructions to show how to use WinFlow to solve real-world problems. In a graphical user environment such as Windows, it is difficult to tell you exactly what to do during each step, however, because many of the steps involve using the mouse. This demonstration guide provides several snap-shots of the WinFlow screen to show you what your screen should look like. In addition, the final data files are provided so that you may skip most of the following steps but still get a feel for how WinFlow works.

The demonstration guide tells you to select menu items and a short-hand method is used to indicate what menu items are to be selected. Thus, the menu items are listed in order of selection and are separated by a "->" symbol. For example, instead of writing the following text:

*Select **File** from the main menu and then select **Open** from the pulldown menu*

the following shortened version is adopted in this manual:

Select File->Open

Some instructions for using the mouse should also be explained here, although most Windows users understand these concepts. The word "click" refers to pressing the left mouse button once and "double-click" means to press the mouse button twice in rapid succession. The word "drag" means to press the left mouse button and move the mouse to another location on the screen while keeping the button pressed down. Release the mouse button when the cursor is on the desired location.

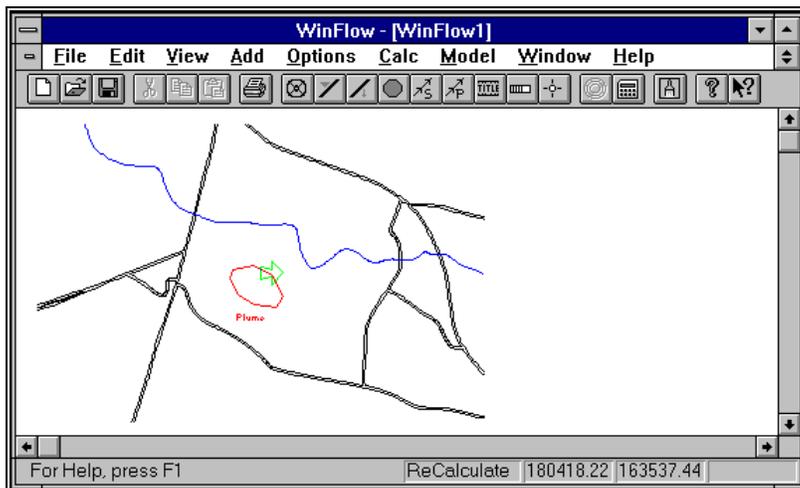
Setting Up the Model



Start the WinFlow program by double-clicking on its icon, . You should see a simple menu displayed above a blank window. A new model is started by selecting File->New or by pressing the  button on the toolbar below the menu. Now you can start to design the model.

The first step in most models is to import a base map that will serve as your reference point for the model. Two maps have been supplied for use in the demonstration guide. You will import the map called PLUME1.MAP. Select File->Map->WinFlow and choose *plume1.map* in the standard Windows file dialog. Your screen should now look like the screen shown below. Notice the green arrow in the center of the screen, which denotes the location of the “reference head”. The reference head is actually located at the center of the base of the arrow. The reference head is a point in the aquifer where the head is fixed during the simulation and cannot change. This is analogous to a “constant head” in a numerical model, such as MODFLOW. You must be careful to locate the reference head as far from the area of interest in your model (e.g., pumping wells) as you can.

In this example, we want to move the reference head to the upper left portion of the screen. You should now move the green arrow to coordinates X=167600 and Y=163180. Place the cursor over the green arrow until you see the  cursor. Now, hold down the left mouse button and drag the green arrow to the desired coordinates. The coordinates are displayed in the lower right corner of the status bar at the bottom of the screen. The coordinates do not have to be exact, since you will be able to edit them when you release the mouse button.



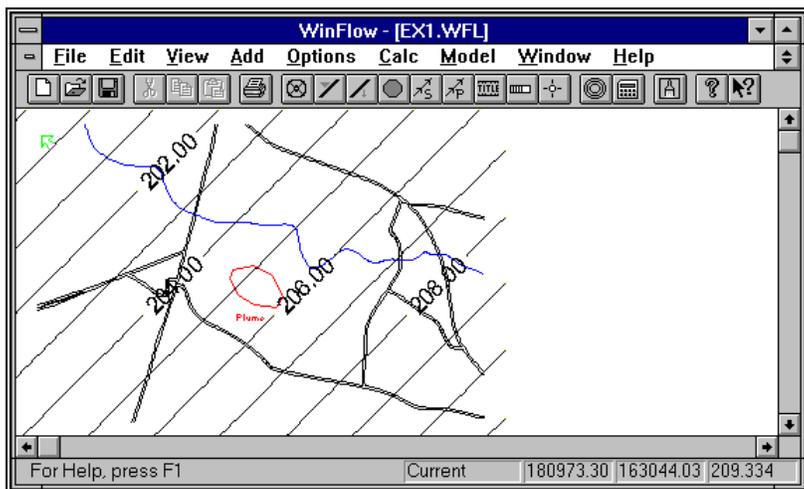
Now, again place the cursor over the reference head (green arrow) and double-click the left mouse button. You will see a dialog that lists data corresponding to the reference head. Check the coordinates listed in the dialog and make sure the coordinates are close to the ones given above. Next, change the hydraulic gradient to 1.0e-03 and the angle to 135 degrees. Click on the OK button when you are done.

The next step is to make sure that the aquifer properties are set correctly. Select **Edit->Parameters** and change the top elevation of the aquifer to 500 ft. In the steady-state model, WinFlow automatically determines whether an aquifer is confined or unconfined. The aquifer is unconfined when the head is below the top of the aquifer and it is confined if the head is above the top of the

aquifer. In this example, the aquifer is unconfined, so you will make the top of the aquifer very high (e.g., a large distance above the reference head value of 200 ft.). The top elevation is the only parameter to change in this example, so click the OK button.

The previous dialog contained a list of all aquifer properties in the WinFlow model. You need to maintain a consistent set of units when you assign these parameters. This is done by selecting one unit of measure for length (e.g. feet) and one unit of measure for time (e.g. days). Using these examples, hydraulic conductivity has units of ft/d and pumping rates are in ft³/d.

You have now set up the most important parts of the WinFlow model! To see the results, simply select **Calc->Recalculate** or click the  button. You should see a series of contours that are straight lines, representing the uniform regional hydraulic gradient. You will also note that the contour interval is uneven. When a new model is started, WinFlow simply computes the maximum head difference in the model and divides this number by ten to get the starting contour interval. You should change the contouring parameters by selecting **Options->Contour->Parameters**. Change the minimum contour level to 198 and the contour interval to 1.0 ft. Click OK when you are done. WinFlow will now ask whether you would like to recontour the current model. Choose YES and your screen should look like the one shown below.



This is a good time to save your work. Select **File->SaveAs** and enter the name EX1 in the dialog. Click OK when you are done. WinFlow automatically attaches the file extension “.wfl” to the file name. Sorry, the demonstration version does not save files, but this is how you would do it in the full version.

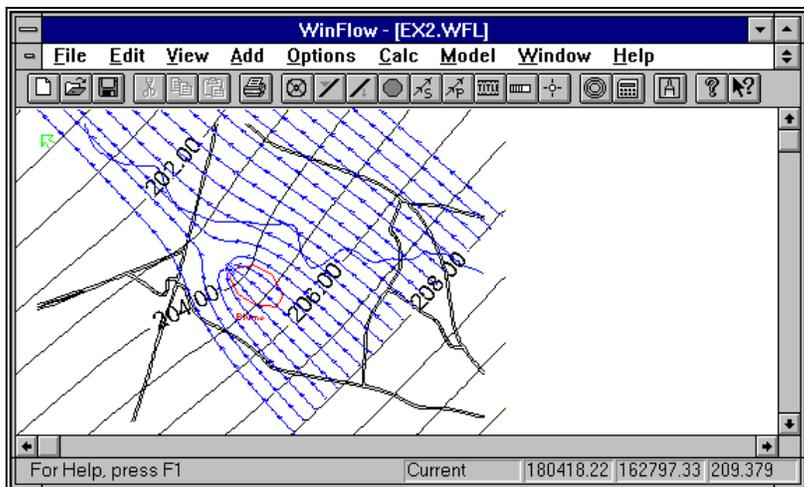
Adding Features

Most models are more complex than the one you just created. Usually, you would like to analyze the effects of pumping wells or other features. In the next section, you will add a pumping well to the model and evaluate the capture zone of the well by adding streamlines. You will also learn how to change the pumping rate of the well.

You should see a red oval shape in the center of the base map. This area is labeled “plume” and represents an area to be captured in a pump-and-treat system. Add a well at the downgradient edge of the plume by selecting **Add->Well** or by clicking the  button. Move the cursor to coordinates X=171500 and Y=160500 and click the left mouse button. The well will be added where the point of the arrow cursor indicates. Next, a dialog box for the well will be displayed; enter a pumping rate of 20,000 ft³/d and click the OK button. Note that the word “ReCalculate” appears in the status bar. This reminds you that the contours on the screen may not be representative of your current model. Click the calculate button  to recompute the model.

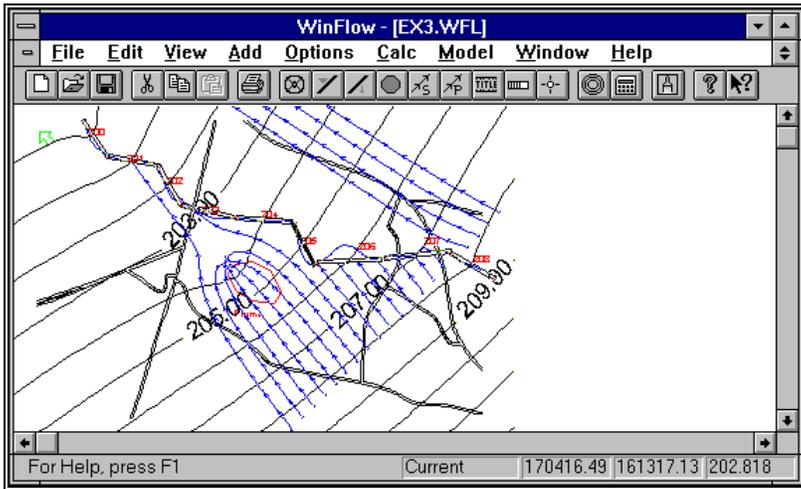
The contours should show a slight cone of depression around the well you added. To see if the well captures the plume, you will need to add several streamlines. Streamlines are used to illustrate groundwater flow directions. Select **Add->Streamlines->Line**. You will now drag a line along the 208 ft contour line in the lower right portion of the screen. After dragging the line, a dialog is displayed. Change the number of streamlines to 20 and click the OK button. The streamlines will then be drawn on the contour map. It is often useful to add arrow-heads to the streamlines to more clearly show the direction of flow. Select **Options->Arrow** and change the distance between arrows to 500 ft and the arrow size to 100. Finally, click on the “Display Arrows” field and then the OK button.

Your screen should look like the one shown below. Note that the capture zone of the pumping well does not quite cover the entire plume area. You can increase the amount of pumping from the well by moving the cursor over the well until you see the  cursor. Now, double-click on the well and the well dialog appears. Change the pumping rate to 50,000 ft³/d and recalculate the model. The capture zone should now cover the entire plume.



Now let’s examine other aspects of the map. The blue line on the map represents a river. Thus far, we have ignored the river in our model. WinFlow can simulate the influence of rivers or drains using linesinks. Linesinks can be defined using either a head value or a flux value. In a head linesink, WinFlow computes the flux needed to maintain the head at the specified value at the center of the linesink. In a flux linesink, you supply WinFlow with the flow rate into or out of the linesink per unit length (take the total flow and divide by the length of the linesink). In this example, you will add 9 linesinks to the model along the river to evaluate the impact on the recovery system.

First, in order to give you a better frame of reference for the linesinks, you will import a new map file. WinFlow allows only one digitized map to be used at any one time. When you import a map file into a model that already has a map, WinFlow deletes the first map and displays the second one. Select **File->Map->WinFlow** and choose PLUME2.MAP. This map has a series of “x”s along the river and numbers between the “x”s. You must add a head linesink between each pair of “x”s and set the head to the value displayed on the map.



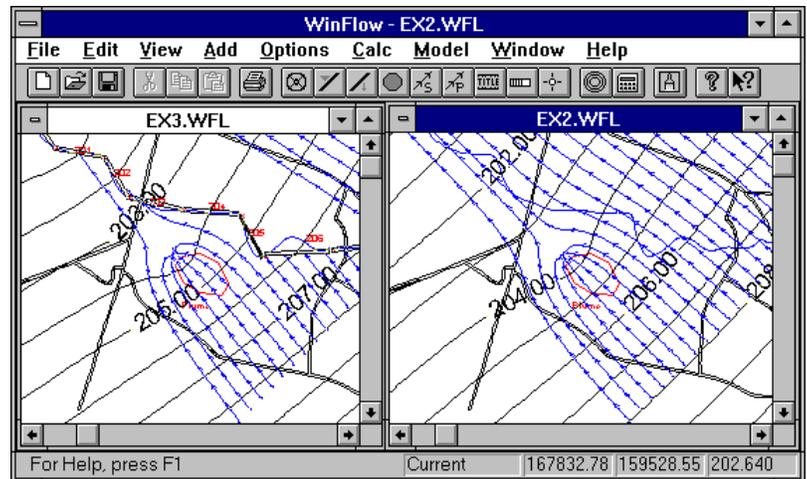
For example, you will add a head linesink in the upper left corner at a head value of 200 ft. Select **Add->Head Linesink** or click the  button. Drag a line between the last two “x”s on the left side of the river. A dialog will be displayed; enter a head value of 200 ft and click the OK button. Now add the remaining 8 linesinks in intervals of 1 ft between the “x”s. Finally, recompute the model and your screen should look like the one to the left.

After adding all of the linesinks, your screen should look similar to the one at the left. You can see that the linesinks have a subtle effect on the flow field. You can also determine how much groundwater the river is gaining or losing. To find out the flow rate of a linesink,

place the cursor on a linesink (you will see the 

cursor) and double-click. The dialog shows the flow rate in ft^3/d per ft of river. All of the linesinks are removing water from the aquifer except the one labeled 204. A linesink removes water from the aquifer if the sign on the flow rate is positive and losing water to the aquifer if the flow rate is negative. Check the linesink flows with your model.

You may want to compare the two models (with and without linesinks) more closely to determine the impact of the river on the aquifer. You could print the contour maps using **File->Print** or you could have both models displayed simultaneously on your screen. You can do this by selecting **File->Open** and choosing the file EX2.WFL. By default, WinFlow overlays the two models so that you can only see one at a time. Select **Window->Tile** and both models will be displayed at the same time. You can resize the windows and move the model windows around. Try to get your screen to look like the one at the right.



WinTran Demo!

Thank you for trying the WinTran demo. We think you will find the software to be the easiest contaminant transport model to use, especially since it is a true Windows program. The WinTran demo is a fully functional copy of WinTran that will not print maps nor save files. It will perform all other functions, however. Please pay special attention to the installation section. WinTran is a 32-bit application that requires you to install Microsoft's Win32s system unless you are running Windows NT or 95. After you are finished evaluating WinTran, you may safely remove Win32s according to the uninstall section of this document.

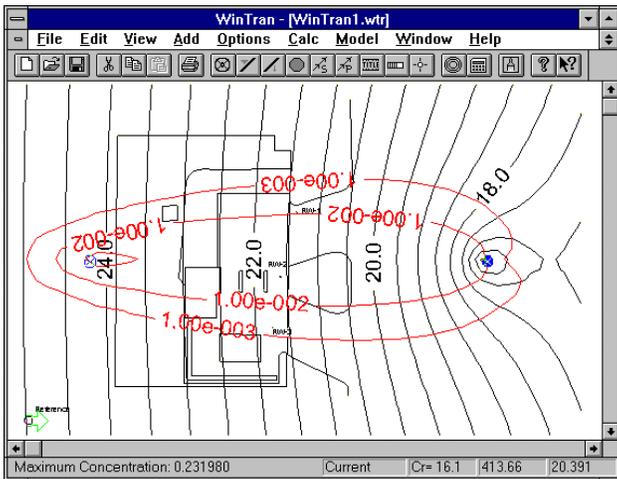
WinTran is unique in that it provides you with a sophisticated finite-element transport model that works like an analytic model. All of the finite-element mesh design chores go on behind the scenes, freeing you to work on the conceptual model and to try different solutions to your problem. WinTran incorporates the steady-state flow model from WinFlow and, therefore, looks very much like WinFlow. The primary differences between WinFlow and WinTran are the following:

- WinTran requires substantially more memory than WinFlow and thus on some systems (< 8 Mbytes of RAM) it will be slower.
- WinTran does not have WinFlow's transient flow model.

If you plan to do a lot of flow simulation work in addition to transport, we recommend purchasing both models and we have provided special pricing considerations to allow you to obtain both models cost-effectively.

Please work through the tutorial at the end of this document for an introduction to WinTran. If you have any questions as you are running the demo please give us a call. Note that the minimum system requirements are Windows V3.1 or higher (or 95 or NT) running in enhanced mode with at least 8 megabytes of RAM (8 preferred) and 3 megabytes of disk space.

WinTran Features



WinTran couples the steady-state groundwater flow model from WinFlow with a contaminant transport model. The transport model has the feel of an analytic model but is actually an embedded finite-element simulator. The finite-element transport model is constructed automatically by the software but displays numerical criteria (Peclet and Courant numbers) to allow the user to avoid numerical or mass balance problems.

Contaminant mass may be injected or extracted using any of the analytic elements, including wells, ponds, and linesinks. In addition, constant concentration elements have been added. WinTran displays both head and concentration contours and concentration may be plotted versus time at selected monitoring locations. The transport model includes the effects of dispersion, linear sorption (retardation), and first-order decay. Other features include:

- ▣ Simulates steady-state flow and transient transport in confined & unconfined aquifers;
- ▣ Simulates effects of wells, linesinks, ponds, and constant concentration sources;
- ▣ 32-bit software uses all memory available to Windows;
- ▣ Supports Windows V3.1 (with Win32s), Windows NT, and Windows 95;
- ▣ Displays Peclet and Courant criteria during transport simulation;
- ▣ Displays mass balance error during simulation;
- ▣ Contours concentration at user-specified time steps during simulation;
- ▣ Velocities computed either analytically or using finite-element flow model;
- ▣ Maps may be printed using any Windows device driver;
- ▣ WYSIWYG printing with print preview and user-defined margins;
- ▣ Full context-sensitive help system (the entire manual is on-line);
- ▣ All other features are the same as WinFlow (except transient flow model)

Exploring WinTran

The WinTran demonstration, described below, introduces you to most of the important features of this software in a step-by-step example. You will be given very specific instructions to show how to use WinTran to solve real-world problems. In a graphical user environment such as Windows, it is difficult to tell you exactly what to do during each step, however, because many of the steps involve using the mouse. This demonstration provides several snap-shots of the WinTran screens to show you what your screen should look like.

The exercise below tells you to select menu items and a short-hand method is used to indicate what menu items are to be selected. The menu items are listed in order of selection and are separated by a “->” symbol. For example, instead of writing the following text:

*Select **File** from the main menu and then select **Open** from the pulldown menu*

the following shortened version is adopted in this manual:

*Select **File->Open***

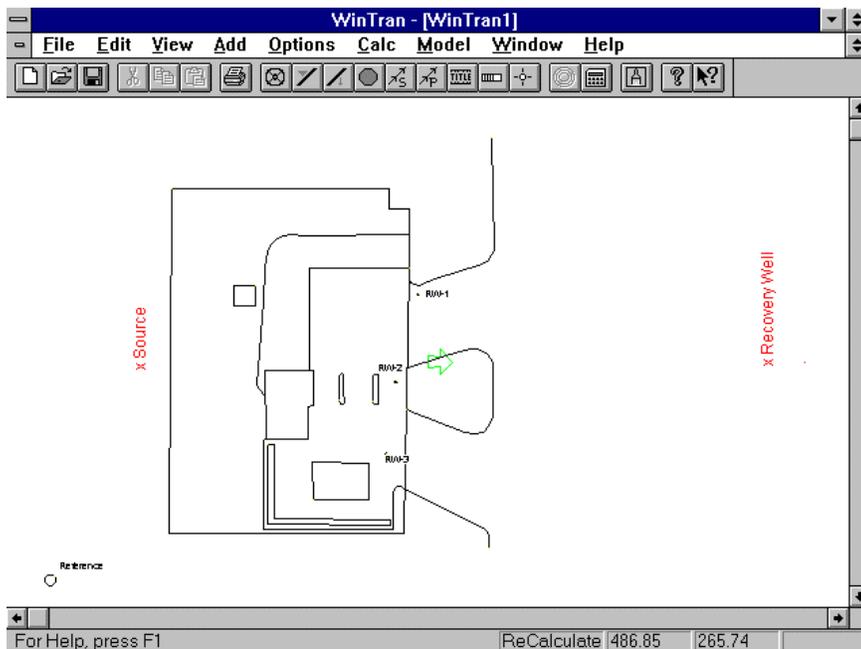
Some instructions for using the mouse should also be explained here, although most Windows users understand these concepts. The word “click” refers to pressing the left mouse button once and “double-click” means to press the mouse button twice in rapid succession. The word “drag” means to press the left mouse button and move the mouse to another location on the screen while keeping the button pressed down. Release the mouse button when the cursor is on the desired location.

Setting Up a New WinTran Model



Start the WinTran program by double-clicking on its icon, . You should see a simple menu displayed above a blank window.

A new model is started by selecting File->New or by pressing the  button on the toolbar below the menu. Now you can start to design the model.



The first step in most models is to import a base map that will serve as your reference point for the model. An example of a base map has been supplied with the demo. You will import the map called SITE1.MAP. Select **File->Map->WinTran** and choose *site1.map* in the standard Windows file dialog. Your screen should now look like the screen shown at the left. Notice the green arrow in the center of the screen, which denotes the location of the “reference head”. The reference head is actually located at the center of the base of the arrow. The reference head is a point in the aquifer where the head is fixed during the simulation and cannot change. This is analogous to a “constant head” in a numerical model, such as MODFLOW. You must be careful to locate the reference head as far from the area of interest in your model (e.g., pumping wells) as you can.

In this example, we want to move the reference head to the lower left portion of the screen (near the word “Reference”). You should now move the green arrow to coordinates X=75 and Y=65. The X and Y coordinates are displayed on the status line at the bottom of the WinTran window as you move the cursor. Place the cursor over the green arrow until you see the  cursor. Now, hold down the left mouse button and drag the green arrow to the desired coordinates. The coordinates do not have to be exact, since you will be able to edit them later.

Now, again place the cursor over the reference head (green arrow) and double-click the left mouse button. You will see a dialog that lists data corresponding to the reference head. Check the coordinates listed in the dialog and make sure the coordinates are close to the ones given above. Next, change the hydraulic gradient to 0.01 and the head value to 25. Click on the OK button when you are done.

The next step is to make sure that the aquifer properties are set correctly. Select **Edit->Flow Parameters** and make sure that the parameters are set to the values below:

Hydraulic conductivity	= 100 ft/d
Bottom Elevation	= 0.0 ft
Top Elevation	= 100.0 ft
Reference Head	= 25.0 ft
Gradient	= 0.01 ft/ft
Recharge	= 0.0 ft/d
Porosity	= 0.2

Note that in the steady-state flow model, WinTran automatically determines whether an aquifer is confined or unconfined. The aquifer is unconfined when the head is below the top of the aquifer and it is confined if the head is above the top of the aquifer. In this example, the aquifer is unconfined, so you will make the top of the aquifer very high (e.g., a large distance above the reference head value of 25 ft.). Click the OK button to end this dialog.

The previous dialog contained a list of all aquifer properties in the WinTran model. You need to maintain a consistent set of units when you assign these parameters. This is done by selecting one unit of measure for length (e.g. feet) and one unit of measure for time (e.g. days). Using these examples, hydraulic conductivity has units of ft/d and pumping rates are in ft³/d.

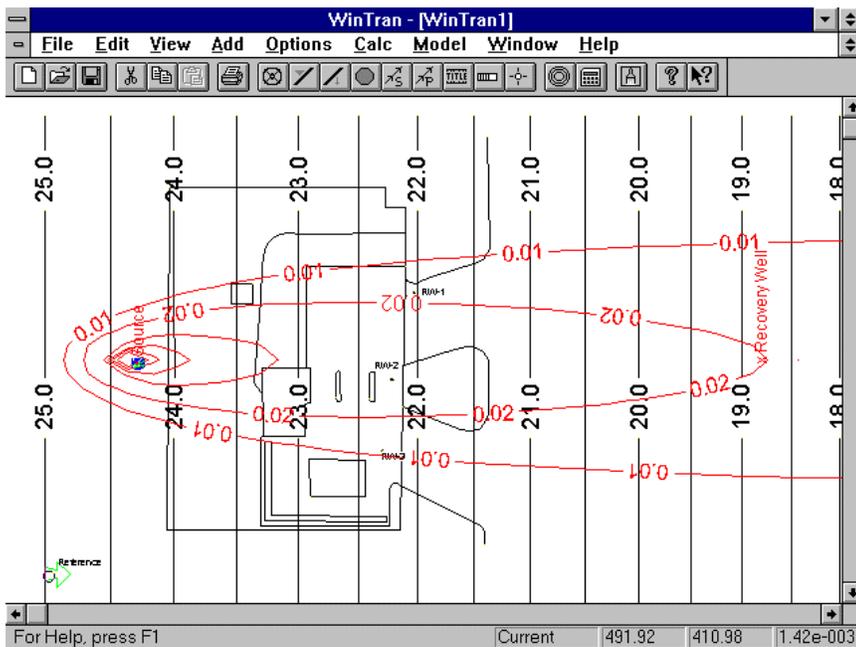
Now, edit the transport parameters by selecting **Edit->Transport** from the menu. Change the longitudinal dispersivity to 30 ft and the transverse dispersivity to 6 ft. Click the OK button when you are done. Finally, we will change the time-stepping parameters for the transport simulation. Select **Edit->Time Stepping** from the menu. Change the number of time steps to 30 and the maximum time step size to 100 days. Click the OK button when you are done.

In order to run the transport model, you need either initial concentrations other than zero or a source of contamination in the model. Sources of contaminant can be injection wells, ponds that recharge the aquifer, or linesinks that are injecting water into the aquifer. In this example, you will add a well at the point on the map labeled "Source". Add a well at this location by selecting **Add->Well** or by clicking the  button. Move the cursor to coordinates X=145 and Y=225 and click the left mouse button. The well will be added where the point of the arrow cursor indicates. Next, a dialog box for the well will be displayed; enter a pumping rate of -1 ft³/d (the negative sign denotes an injection rate) and a concentration of 100. Now, click the OK button. Note that the word "ReCalculate" appears in the status bar. This reminds you that the contours on the screen may not be representative of your current model.

You have now set up the initial WinTran demo model! To see the results, simply select **Calc->Recalculate** or click the  button. You should see a series of contours that are straight lines, representing the uniform regional hydraulic gradient, but there are no concentration contours. This is because we have not told WinTran to run the transport model yet; only the flow model was simulated.

Before running the transport model, change the contouring options for both the flow model and for the transport model. Select **Options->Contour->Flow** from the menu. Change the minimum contour level to 18.0 ft and the contour interval to 0.5 ft. Also, change the precision value to 1 and click the OK button. A dialog will prompt you to recontour; select YES to perform the recontouring. Next, choose **Options->Contour->Transport**. We will contour concentrations every 10 time steps (first parameter on the dialog) with a starting log cycle concentration of 0.01 and 5 contours per log cycle. Unlike flow, where head is computed on a regular contour interval, concentration is contoured using log intervals. Click the OK button when you are done. Finally, select **Options->Colors** and change the concentration contour color to red.

To run transport, select **Model->Transport** from the menu. Now select **Calc->Recalculate** and the flow and transport model will run. You will first see the flow model computed, followed by a progress dialog showing the percentage complete for the transport simulation. In one of the dialogs above, we told WinTran to contour the results every 10 time steps. You also specified a total of 30 time steps for the simulation. You should see WinTran contour concentration 3 times for this simulation. At each time step, you should see the plume grow larger. The final plume is shown below:

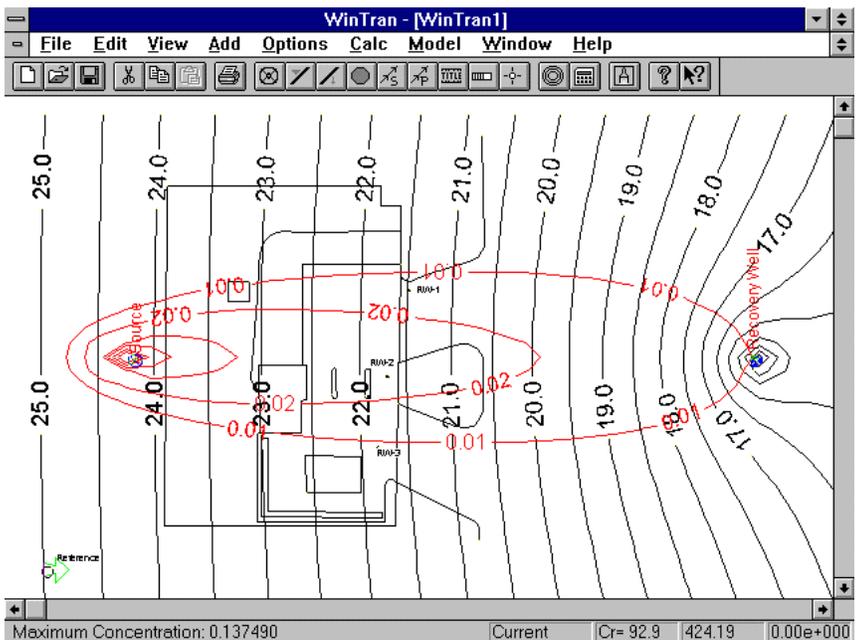


You should also see information displayed on the status line at the bottom of the window. The left side of the status line will show the current time step number and time value (time from the start of the simulation), as well as the mass balance error. This value should be small (0.01 percent in this case). To the right, WinTran displays the Peclet and Courant Criteria for the transport simulation. Generally, the Peclet number should be less than 2 and the Courant number less than 1. In this case, the Courant number is a bit high, but the mass balance is good (so we will not worry about it for this demo).

Simulating Remediation

The next step in the demonstration is to add a recovery well to the model. Select **Add->Well** from the menu or click the  button on the toolbar. Move the cursor to coordinates X=620, Y=225 (near the word "Recovery Well") and click

the left mouse button. Change the pumping rate to 10,000 ft³/d and click the OK button.

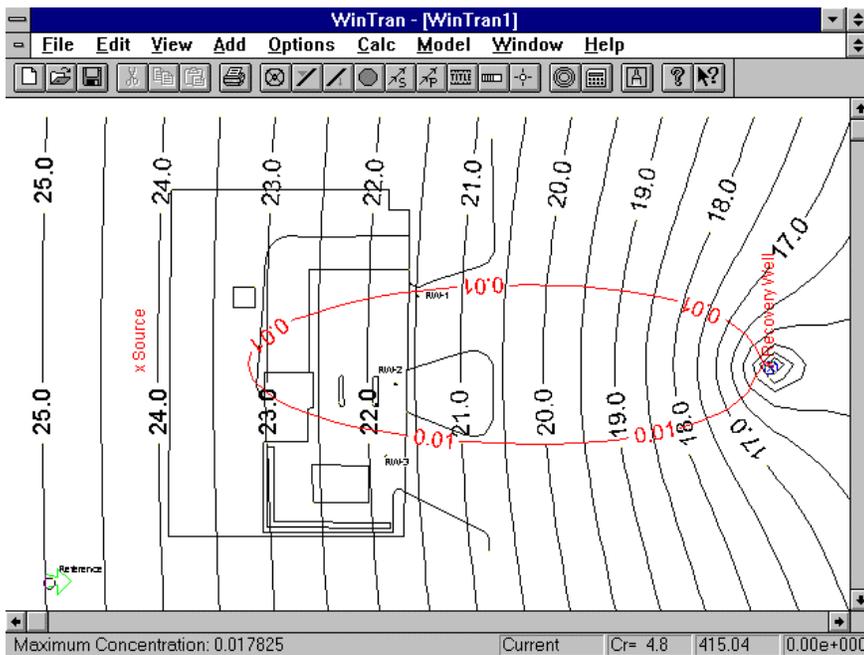


You will now save the concentrations computed at the end of the last simulation as initial concentrations for the remediation scenario. By default, WinTran assumes that initial concentrations are zero everywhere in the aquifer. You may override this default by selecting **Calc->Restart** from the menu. This action saves the current concentrations as initial conditions for the next run. Do this now.

To see the results, simply select **Calc->Recalculate** or click the  button. This time, you will see the cone of depression around the recovery well and the concentration contours stop at the recovery well. Note that the source is still in place, so a steady-state concentration distribution develops between the source and recovery well, as shown in the figure at the left.

As a final demonstration, we will remove the source

to observe the clean-up of the plume. First, place the cursor over the source well until you see the  cursor. Click the left mouse button to select the well. Now, press the Del key or select **Edit->Delete** from the menu to remove the well. Select **Calc->Restart** to save the previous concentrations as initial conditions and select **Calc->Recalculate** to see the results. Notice that the plume disappears quickly. To get a better picture of the cleanup, select **Edit->Time Stepping** and change the number of time steps to 10 and the maximum time step size to 10. Select **Calc->Recalculate** and your screen should look like the following:



Concentration Breakthrough Curves

In addition to contours of concentration and head, WinTran can plot concentration over time at observation wells. You define observation wells by clicking on the check-box labeled “Monitor Concentration vs. Time” on the well dialog. The next time you run the model, WinTran will store concentrations computed at all observation wells for all time steps. You plot the results by selecting **Calc->Plot C vs. T** from the main menu. The following will show you how to create a Concentration vs. Time plot.

First, we will start with our original example. Close the current model by selecting **File->Close**. Now read in a new model by selecting **File->Open**. Choose the file called “ex1.wtr” in the file open dialog. Click OK when you have found this file. WinTran will read the file and compute heads only. WinTran does not compute the transport model when you open a new file because these simulations can be quite long for complex models.

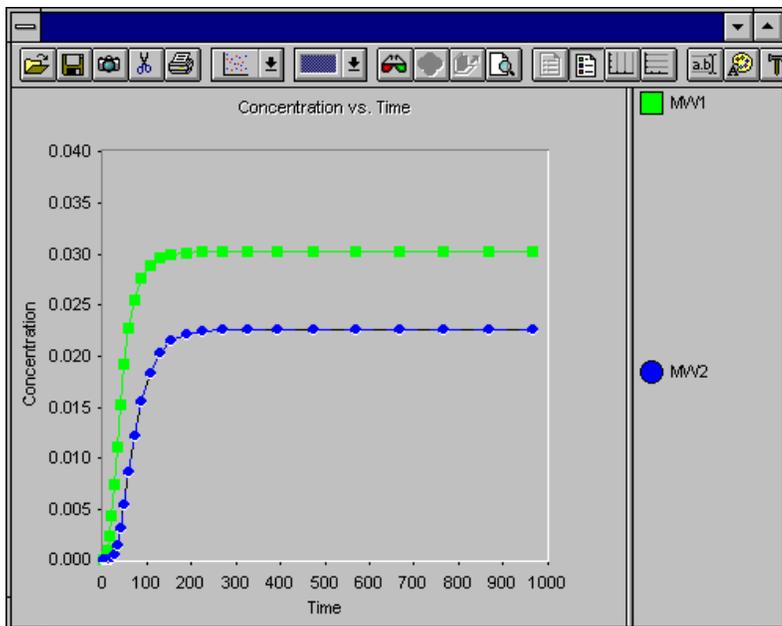
Before running the transport model, we will define two observation well locations. Select **Add->Well** from the menu or click the  button on the toolbar. Move the cursor to coordinates X=340, Y=230 and click the left mouse button. The coordinates do not need to be exact. Change the pumping rate to 0.0 and click the check-box labeled “Monitor Concentration vs. Time”. A check mark in this box tells WinTran to store concentration information at this well location. Click on the button labeled “Name” and enter a name of “MW1” for this well. Naming the well will allow you to distinguish between multiple observation wells on the plot. The zero pumping rate means that the well is for observation purposes only. You may also monitor concentration vs. time in pumping wells. Click the OK button to end this dialog. Now, repeat this procedure to add an observation well at X=500, Y=230. Call this well MW2.

Select **Model->Transport** and click the calculator button to run the transport model. When the model is finished running, select **Calc->Plot C vs. T** from the main menu. A dialog will be displayed that allows you to set up the axes of the graph. Enter the following numbers:

Minimum Time	0.0
Maximum Time	1000.0
Delta T	100.0
Number of decimals	0

Minimum Concentration	0.0
Maximum Concentration	0.04
Delta-C	0.005
Number of decimals	3

Click OK to end editing on this dialog and a plot of concentration versus time is created for the two observation wells. Your screen should look like the one shown below:



We hope that this brief demonstration has shown you what a powerful tool WinTran can be in simulating the fate and transport of dissolved contaminants in groundwater. If you have any questions or would like to order WinTran, please give us a call.

Scientific Software Group

PO Box 23041

Washington, DC 20026-3041

Phone: 703-620-9214

Fax: 703-620-6793

e-mail: info@scisoftware.com

web site: <http://www.scisoftware.com>