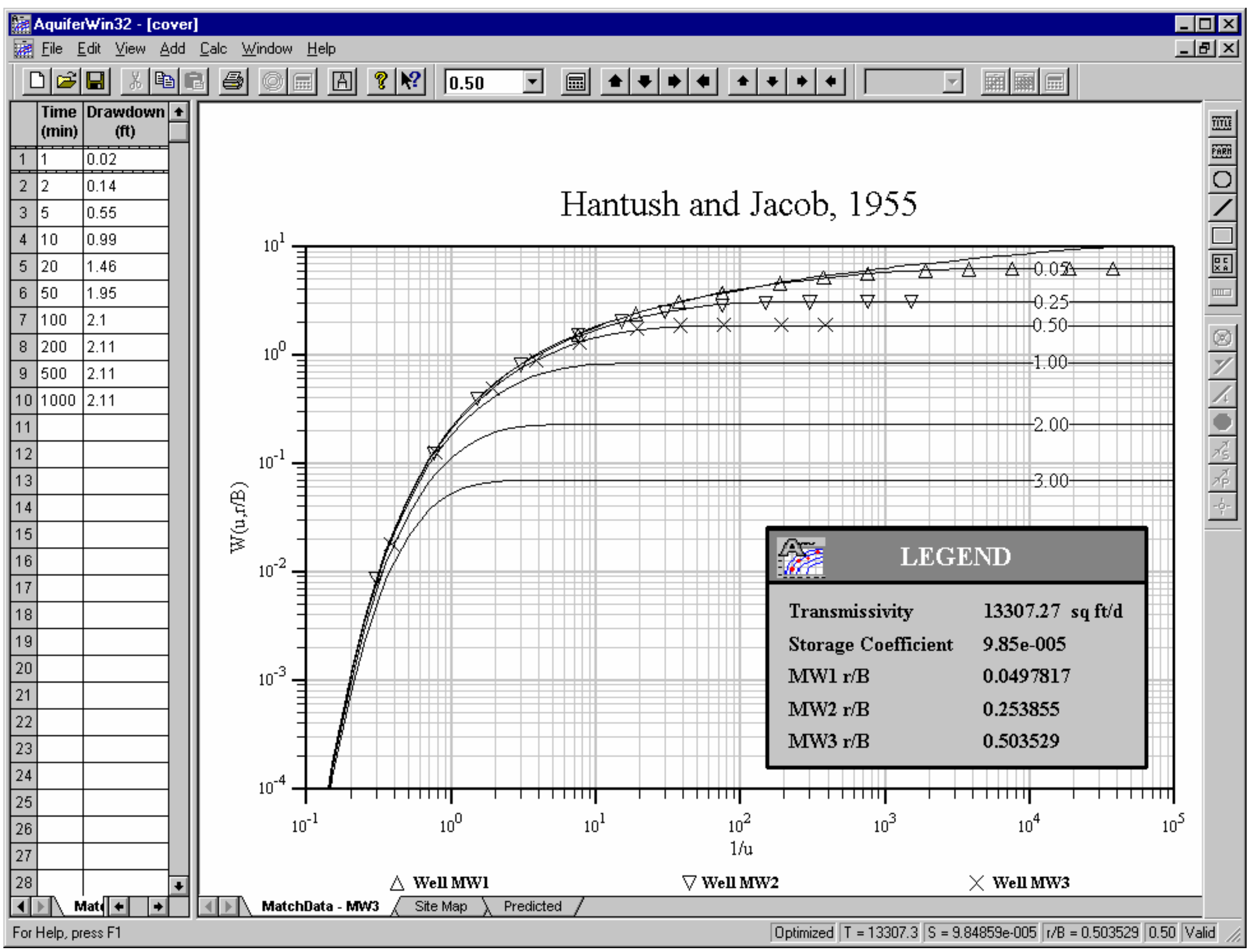
Guide to Using

Aquifer^{Win32}

Version 4

by Environmental Simulations, Inc.



This manual was produced using *Doc-To-Help*®, by ComponentOne LLC

ESI Programming by:
Douglas B. Rumbaugh
James O. Rumbaugh

Environmental
Simulations
Inc.

Environmental Simulations, Inc.
300 Mountain Top Rd
Reinholds, PA 17569
tel. (610) 670-3400
fax. (610) 670-9239

support@groundwatermodels.com
www.groundwatermodels.com

Contents

Introduction

Combined Manual

Aquifer^{Win32}, WinFlow and WinTran are now basically the same product. With the exception of the program name, the default icon, and the types of document files you can create everything is the same. An Aquifer^{Win32} Flow Model document file is identical to a WinFlow document file and the user-interfaces are identical as well. We no longer sell multiple products and only sell the Modelling Version of Aquifer^{Win32}.

What is Aquifer^{Win32}?

Aquifer^{Win32} is the most sophisticated and most Windows™ compliant application for the analysis and presentation of aquifer tests including pump tests, slug tests, step tests and analytic element flow and contaminant transport models. The analysis of these data incorporates a wide variety of solution types with comprehensive plotting features.

Aquifer^{Win32} comes in four different versions, *Modeling*, *Professional*, *Standard*, and *Slug Test*. Aquifer^{Win32} is an OLE Full Server allowing the results to be linked to or embedded in OLE client applications. The *Slug Test* version is limited to the analysis of slug test data using any one of the 6 slug test solutions; these solutions range from the simple Hvorslev to the complex Kansas Geological Survey (KGS) Model supporting confined or unconfined conditions, partial penetration, well skin effects and the response of a monitoring well. The *Standard* version adds over a dozen pump test analyses including solutions for confined, leaky confined, unconfined and fracture rock aquifers with support for variable pumping rates, partial penetration, delayed yield and well bore storage. The *Professional* version adds derivative analysis, Step tests and a Pump Test Simulator. The *Modeling* version extends many of the pump test solutions into a modeling environment supporting any number of pumping wells with variable pumping rates. Output includes contour maps of hydraulic head or drawdown, color floods, particle traces and graphs of drawdown versus time at any number of monitoring wells. Auto-calibration to any number of transient targets is also supported.

The following are some of the most important features of Aquifer^{Win32}:

About data entry…

- As simple as entering or importing data into a spreadsheet, characterize pumping and monitoring well, select solution type and match data
- Alternatively, designed as a repository for raw aquifer test data with programmatic data conversions
  - Define a site plan including a site map, well locations and well construction information
  - Define an aquifer test including pumping schedule, wells monitored and raw drawdown versus time data
  - Define an analysis by grouping wells, transforming and clipping well response data, optionally adjust for radial distance on a well by well basis

About data analysis…

- Primary support for traditional manual curve matching techniques
- User selectable and unlimited type curves on curve match graph
- Multiple parameters available as type curves for many analyses
- Graphically visualize the impact of specific parameters with custom type curve suites
- Extensive curve match optimization capability
  - Control which parameters are optimized
  - Set minimum and maximum bounds on parameters
  - Optimize any parameters across multiple data sets
- Manual and optimized curve match of the first-order derivative of the data to first-order derivative type curves
- Support for variable pumping rates
- Pump test simulations with contour maps and time/drawdown graphs
- Steamline and particle trace analysis
- Analytic element modeling with recharge, ponds, linesinks etc.
- Auto-calibration of flow modeling parameters

About units…

- Full control of parameter and data units on a parameter by parameter and well by well basis
- On-the-fly unit conversions
- Peer review process assisted by instantaneous global unit conversions without affecting match results
- Parameter-based unit conversion calculator

About graphics…

- Full control of graphs including size, titles, axes, colors, fonts, dash patterns and line thickness

- Type curve graph, predicted drawdown curve through data points, observed drawdown data
- Contours of predicted drawdown at a given time and predicted drawdown versus time data at any number of monitoring wells
- Annotate maps and graphs with text, parameters, symbols, lines, frames and legends
- Frames support display of bitmaps and metafiles
- Exports to DXF, Windows Metafile and ArcView™ Shapefile formats
- Site map and well location plan displayed in map view
- Color flood maps in addition to or as an alternative to contour maps
- Three dimensional perspective display using the Visualization Toolkit (vtk), written and copyrighted by Ken Martin, Will Schroeder and Bill Lorensen.

About printing…

- WYSIWYG printing with Print Preview of all views
- Customizable margins and scaling
- Customizable headers and/or footers supporting bitmaps and metafiles
- Supports any Windows™ printer driver

About Windows™ features…

- Multiple Document Interface
- OLE Full-server supporting linked and embedded items
- Copy views to clipboard as metafiles and OLE objects
- Tab views including spreadsheet, type curves, predicted curves, map and simulator
- Data spreadsheet in split window
- Context-sensitive help
- Context menus
- Property Sheets (Tab Dialogs) to maximize ease of use
- Tip of the Day
- Dockable toolbars with tool tips

Slug Test Analyses

Hvorslev, 1951	Time Lag and Soil Permeability in Ground-Water Observations
Bouwer & Rice, 1976	Slug test for determining hydraulic conductivity of unconfined aquifers with completely or partially penetrating wells
Black, 1978	The use of the slug test in groundwater investigations (Modified Bouwer & Rice unconfined aquifer slug test analysis using an exponential type curve)

Cooper, Bredehoeft & Papadopulos, 1967	Response of a Finite-Diameter Well to an Instantaneous Charge of Water
Hyder, Butler, McElwee & Liu, 1994	Slug tests in partially penetrating wells (KGS Model including well skin and monitoring well response)
Kipp, 1985	Type Curve Analysis of Inertial Effects in the Response of a Well to a Slug Test

Pumping Test Analyses

Cooper and Jacob, 1946	A generalized graphical method for evaluating formation constants and summarizing well field history. (Cooper Jacob Straight Line Method)
Theis, 1935	Constant discharge from a fully penetrating well in a nonleaky aquifer*
Theis, 1935 (Unconfined)	Constant discharge from a fully penetrating well in a nonleaky aquifer*
Theis, 1946 (Recovery)	Recovery test after constant discharge from a fully penetrating well in a nonleaky aquifer
Hantush, 1961	Constant discharge from a partially penetrating well in a nonleaky aquifer*
Papadopulos and Cooper, 1967	Constant discharge from a fully penetrating well of finite diameter in a nonleaky aquifer*
Hantush, 1960	Constant discharge from a well in a leaky aquifer with storage of water in the confining beds*
Hantush and Jacob, 1955	Constant discharge from a fully penetrating well in a leaky aquifer*
Hantush, 1964	Constant discharge from a partially penetrating well in a leaky aquifer*
Neuman, 1972	Theory of flow in unconfined aquifers considering delayed response of the water table*
Neuman, 1974	Effects of partial penetration on flow in unconfined aquifers considering delayed aquifer response*
Moench, 1984	Double-Porosity Models for a Fissured Groundwater Reservoir with Fracture Skin*
Moench, 1985	Transient Flow to a Large-Diameter Well in an Aquifer With Storative Semiconfining Layers*
Moench, 1997	Flow to a well of finite diameter in a homogeneous, anisotropic water table aquifer

*** Analysis available for use in pump test simulator**

Step/Variable Rate Test Analyses

Eden and Hazel, 1973	Step-drawdown test analysis for fully penetrating well in a confined aquifer. Determines well losses and aquifer transmissivity.

Birsoy and Summers, 1980	Variable or intermittent discharge rate analysis for well in a confined aquifer. Determination of aquifer transmissivity and storage.
Model Solutions	
Theis, 1935	Constant discharge from a fully penetrating well in a nonleaky aquifer
Hantush, 1960	Constant discharge from a well in a leaky aquifer with storage of water in the confining beds
Hantush and Jacob, 1955	Constant discharge from a fully penetrating well in a leaky aquifer
Neuman, 1972	Theory of flow in unconfined aquifers considering delayed response of the water table
WinFlow	Analytic element flow model developed by ESI
WinTran	Analytic element flow and Finite element contaminant transport model developed by ESI

What is WinFlow?

WinFlow is a powerful yet easy-to-use groundwater flow model. The user-interface represents the most sophisticated and Windows™ compliant available today. WinFlow provides an extensible common user-interface for analytical analyses and models capable of hosting other calculation engines in the future.

WinFlow is an interactive, analytical modeling tool 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) for confined aquifers, Hantush and Jacob (1955) and Hantush (1960) for leaky aquifers, and Neuman (1972) for unconfined aquifers. 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, linear sources/sinks, 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) or Hantush (1960) equation for leaky aquifers. Neuman's method (1972) can also be used for unconfined aquifers with delayed yield from storage.

In addition to the WinFlow calculation engine described above, WinFlow extends other analytical solutions from the popular Aquifer^{Win32} pumping test analysis application into its modeling environment. These additional solutions support any

number of pumping wells with variable pumping rates. Auto-calibration to any number of transient targets is also supported for these additional solutions.

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 tab 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.

What is WinTran?

WinTran is designed to be an easy-to-use model for simulating the fate and transport of dissolved contaminants in fully saturated groundwater systems. The WinTran model couples the steady-state groundwater flow model from WinFlow, with a contaminant transport model. The transport model feels like an analytic model but is actually an embedded finite-element simulator. The software automatically constructs the finite-element transport model so that you may quickly get answers to your groundwater problems.

The steady-state flow model in WinTran uses analytic functions developed by Strack (1989) to simulate the effects of wells, uniform recharge, circular recharge/discharge areas (called ponds), and line sources or sinks. Any number of these elements may be added to the model. The model depicts the flow field using streamlines, particle-traces, contours of hydraulic head (water levels) and color floods of hydraulic heads. Both confined and unconfined aquifers may be simulated with the WinTran flow model.

The contaminant transport model uses a finite-element formulation whereby the finite-element mesh is identical to the head contour matrix. The contour matrix is a rectangular array of points where head is computed by the flow model. WinTran computes groundwater velocity at each "node" in the contour matrix for use in the finite-element transport model. Diagnostic information is displayed on the status bar at the bottom of the window as the transport model runs. These data alert you to potential problems in the numerical transport model. These diagnostic data include the mass balance error, Peclet number, and Courant number. If these error criteria indicate problems, you may stop the simulation, choose new simulation options, and start the simulation again.

Contaminant mass may be injected or extracted using any of the analytic elements from the groundwater flow model, including wells, ponds, and linesinks. In addition, constant concentration elements may be placed in the model to keep the source contaminant concentration at a specified value. WinTran displays both head and concentration contours. Concentration versus time data may be calculated and graphically displayed for selected monitoring locations. The transport model includes the effects of dispersion, linear sorption (retardation), and first-order decay. The latter may be used to simulate the biologic decay of organic compounds, such as benzene or the radioactive decay of elements such as uranium.

WinTran can import a Drawing Interchange Format (DXF) file (from AutoCAD, for example) to use as a digitized base map. The digitized map gives you a frame of reference for designing the flow and transport models.

WinTran 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.

Installation

Aquifer^{Win32} and WinFlow are distributed on CD-ROM and use a sophisticated installation wizard that is similar to other Windows™ products. You simply run "Setup" from the CD-ROM and follow the directions as the installation proceeds. Start by placing the CD-ROM in the drive. Now, select **Run** from the **Start** menu and enter the following:

d:\setup.exe

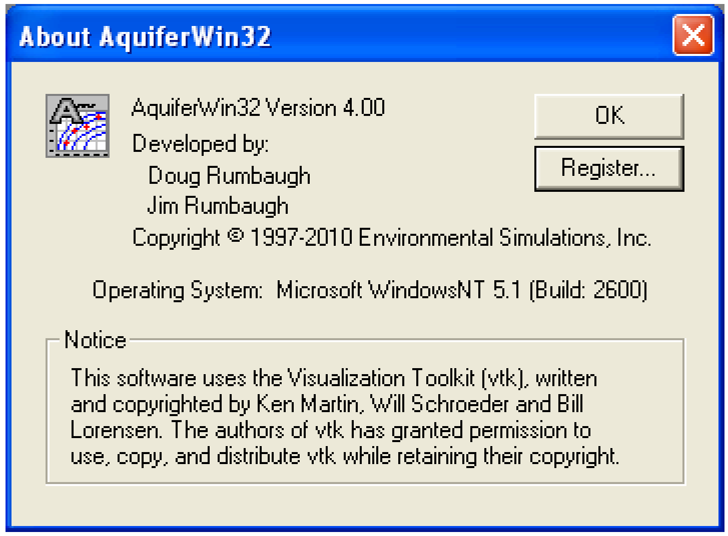
You must first agree to the "ENVIRONMENTAL SIMULATIONS SOFTWARE LICENSE AGREEMENT" in order to continue installing the software. The next page prompts for the directory where the files will be stored. The default is "c:\aquifer3" or "c:\winflow3". If you would like to place the files in a different directory, click the **Browse** button and locate a new directory. Click the **Next** button when you are done. Select **Cancel** at any time to terminate the installation process. The next step is to decide which optional components to install. By default, example files are installed and documentation files, in .pdf format, are not. Click the **Next** button after checking the optional components you want to install.

The next page of the wizard allows you to specify the name of the submenu to add to the **Start->Programs** menu; the default name is AquiferWin32 Version 3 or WinFLow Version 3. To change this, you may select from an existing submenu listed or type a new name. Select **Next** to accept your choice. Finally, select the **Next** button again to begin the installation. After all the files have been installed, another wizard will be started to install the security block device driver; you must have administrator rights to install this driver. After the device driver installation is complete, click the **Finish** button on this wizard and on the main installation wizard.

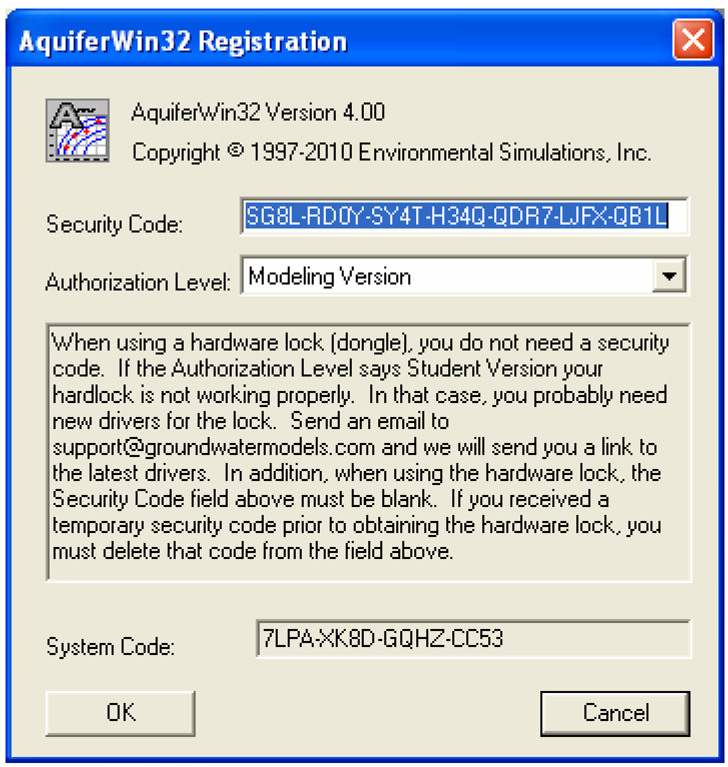
Optionally Obtaining a Security Code

Aquifer^{Win32} and WinFlow are protected by a security block (dongle) or an optional security code that is tied to the computer you install it on. By default, Aquifer^{Win32} and WinFlow both ship with a security block. If you opt for a security code instead, you are licensed to run Aquifer^{Win32} or WinFlow on only one computer and you must obtain a security code in order to complete the installation. If you obtain a security block, you are licensed to install Aquifer^{Win32} or WinFlow on any number of computers; however, Aquifer^{Win32} and WinFlow will require the security block to be installed on a given computer before its full functionality is activated.

If you have a security block, plug the block into the parallel port on the computer and start the program. If you do not have a security block, registration is a three-step process. The first step is to select the **Help->About** menu to display the *About AquiferWin32* or *About WinFlow* dialog as below.



Click the **Register** button to display the *AquiferWin32 Registration* or *WinFlow Registration* dialog as below.



The dialog will display a *System Code* which can be copied to the clipboard (you do this by highlighting the system code and pressing Ctrl-C) and pasted into an email to be sent to aquifer@groundwatermodels.com or aquifer@esinternational.com. We will reply with a security code that you paste or enter into the *Security Code* field in the above dialog. Alternatively, you can get a security code by calling either ESI in the United States at (610) 670-3400 or ESL in the United Kingdom at +44 1743 248600; however, it is strongly recommended that the transaction be done via email since the codes are rather long.

If the security code is invalid or expired, the program will run as a demo which is fully functional without exports, printing and OLE.

Uninstalling the Software

Uninstalling Aquifer^{Win32}

Aquifer^{Win32} and WinFlow are uninstalled like most other software packages. Click the **Start->Settings->Control Panel** menu, double click on the Add/Remove Programs icon, locate "AquiferWin32 Version 3" or "WinFlow Version 3" in the list box and click the Add/Remove button. At this point, you have 3 choices: (1) select **Automatic** to quickly remove all of the Aquifer^{Win32} or WinFlow 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 files in your Windows directory or any other directory. Only files placed on your computer related to Aquifer^{Win32} or WinFlow will be removed.

How to Use This Manual

You should start learning about Aquifer^{Win32} and/or WinFlow by working through the appropriate tutorials in the next chapter. The tutorials are quite extensive and cover most of the major concepts in Aquifer^{Win32} and WinFlow. You should also read the chapter entitled *Concepts* to learn about the primary assumptions that Aquifer^{Win32} and WinFlow use in analyzing aquifer test data, simulating aquifer tests, and flow and transport modeling. After you have read through these two chapters, you are ready to start.

Technical Support

At times, you will have questions about a particular menu or property sheet. The chapter entitled *Menus and Property Sheets* is a reference section that describes each menu and property sheet. If you cannot find the answer to your question, try looking at the index or search through the help file. Still puzzled? Call or e-mail the numbers listed below, our technical support is always free!

United States:

(610) 670-3400

email: support@groundwatermodels.com

United Kingdom:

+44 1743 248600

email: aquifer@esinternational.com

We always keep the latest version of Aquifer^{Win32} on the Internet. The updates are available on our web site at http://www.groundwatermodels.com. You may download the update file from our ftp site at any time. The update is a self-installing program that should be run from Program Manager (or **Start->Run**) to install the update.

What's New In Version 3.0

General

Aquifer^{Win32}/WinFlow have been updated to include many of the things users have requested. Every attempt has been made to make the application more user-friendly and full featured. If we have missed something you feel is important, let us know. We will continually be enhancing Aquifer^{Win32}/WinFlow and all interim releases will be free and downloadable from the internet until Version 4.0 is available.

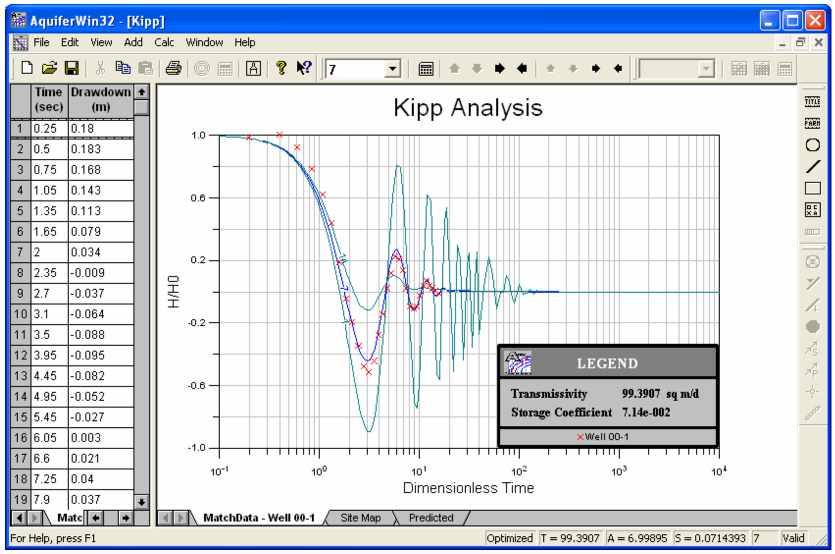
The major changes are summarized below:

- **New analyses**
  - Kipp, 1985 – Type Curve Analysis of Inertial Effects in the Response of a Well to a Slug Test
  - Moench, 1985 - Transient Flow to a Large-Diameter Well in an Aquifer With Storative Semiconfining Layers
  - Moench, 1997 - Flow to a well of finite diameter in a homogeneous, anisotropic water table aquifer
  - WinTran - WinTran is designed to be an easy-to-use model for simulating the fate and transport of dissolved contaminants in fully saturated groundwater systems.
- **New functionality**
  - On screen editing of items within legends
  - Legend Wizard – Enables easy setup of legends and optionally automatically resizes legend, adds default parameters and maintains position in the lower right corner of the graph or map
  - 3D Perspective in Flow Model
  - Line Calculations and Distance/Drawdown graphs
  - Shapefile export of contours, particle traces, streamlines and wells
  - Shapefile maps now supported

New Analyses

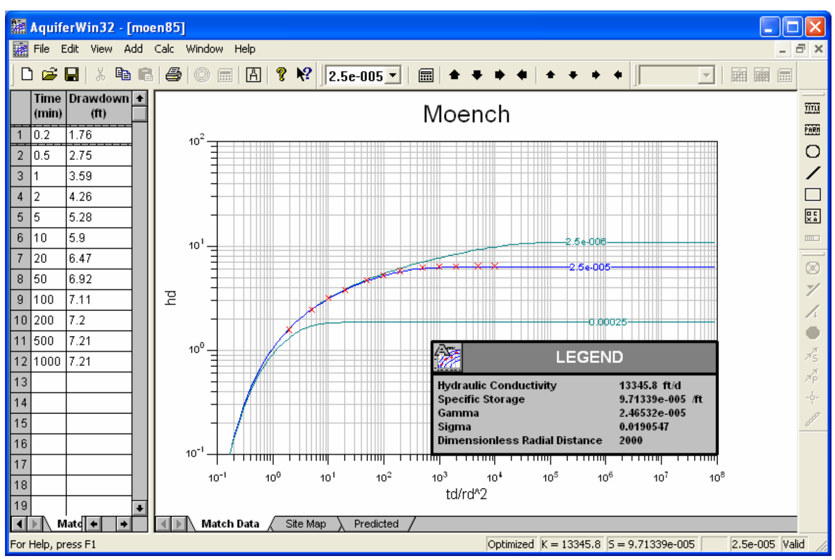
Kipp, 1985

The Kipp, 1985 analysis is used for slug tests that exhibit a sinusoidal response due to inertial effects.



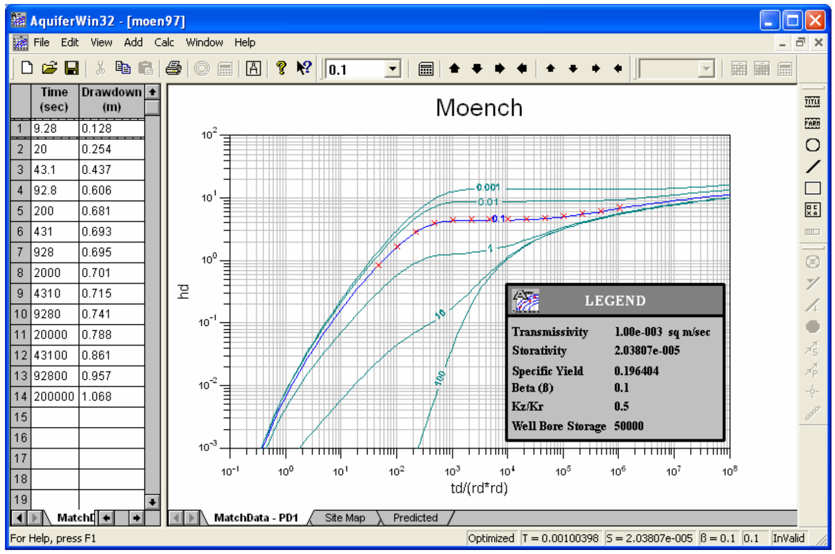
Moench, 1985

The Moench, 1985 analysis is very versatile and supports all three Hantush aquitard boundary types (Constant head, No Flow and Constant Head/No Flow)) and allows leakage from above, below or both. It also supports a line source well or a finite diameter well including well bore storage and well bore skin.



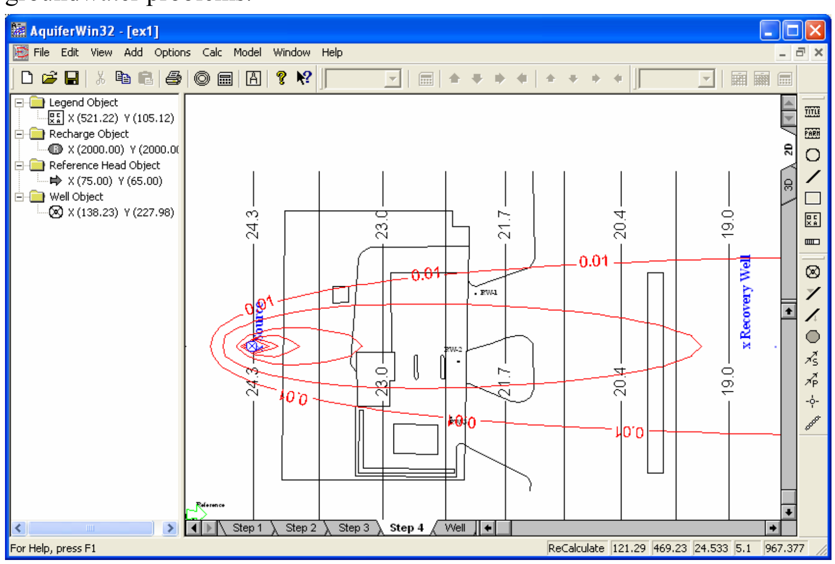
Moench, 1997

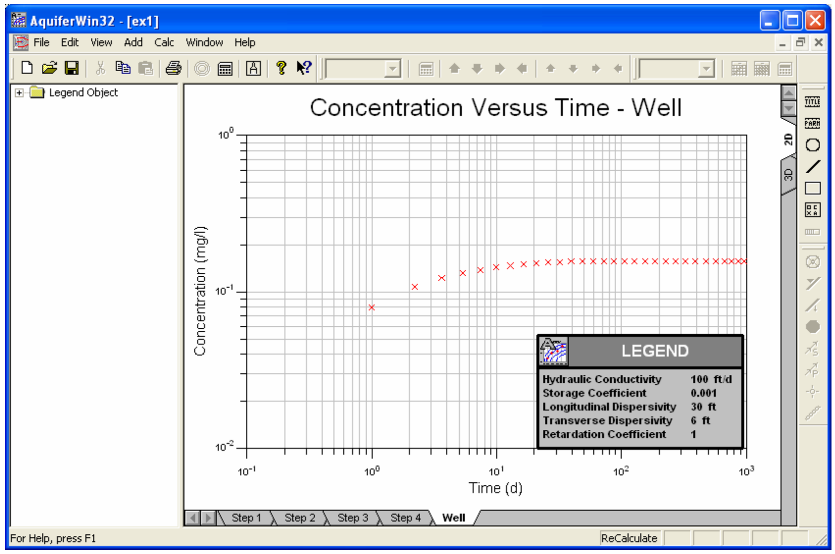
The Moench, 1997 analysis allows analysis of pump tests in unconfined aquifers in which well bore storage and/or well bore skin are a significant factor.



WinTran

WinTran is designed to be an easy-to-use model for simulating the fate and transport of dissolved contaminants in fully saturated groundwater systems. The WinTran model couples the steady-state groundwater flow model from WinFlow with a contaminant transport model. The transport model feels like an analytic model but is actually an embedded finite-element simulator. The software automatically constructs the finite-element transport so that you may quickly get answers to your groundwater problems.




AquiferWin32 - [ex1]
File Edit View Add Calc Window Help
Legend Object
Concentration Versus Time - Well
Concentration (mg/l)
Time (d)
LEGEND
Hydraulic Conductivity 100 ft/d
Storage Coefficient 0.001
Longitudinal Dispersivity 30 ft
Transverse Dispersivity 6 ft
Retardation Coefficient 1
Step 1
Step 2
Step 3
Step 4
Well
For Help, press F1
ReCalculate

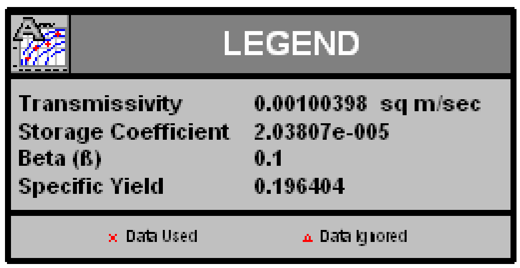
New Functionality

Legend Wizard

One of the most common support issues has been setting up legends and we have attempted to make this process much easier. In Version 3, adding a legend is as simple as clicking the mouse in the view, clicking the **Add->Legend** menu and dragging a rectangle. The following wizard is activated to help in setting up the wizard. If you simply click the finish button, a legend will be created containing the pertinent parameters and will be located in the lower right corner of the graph or map.

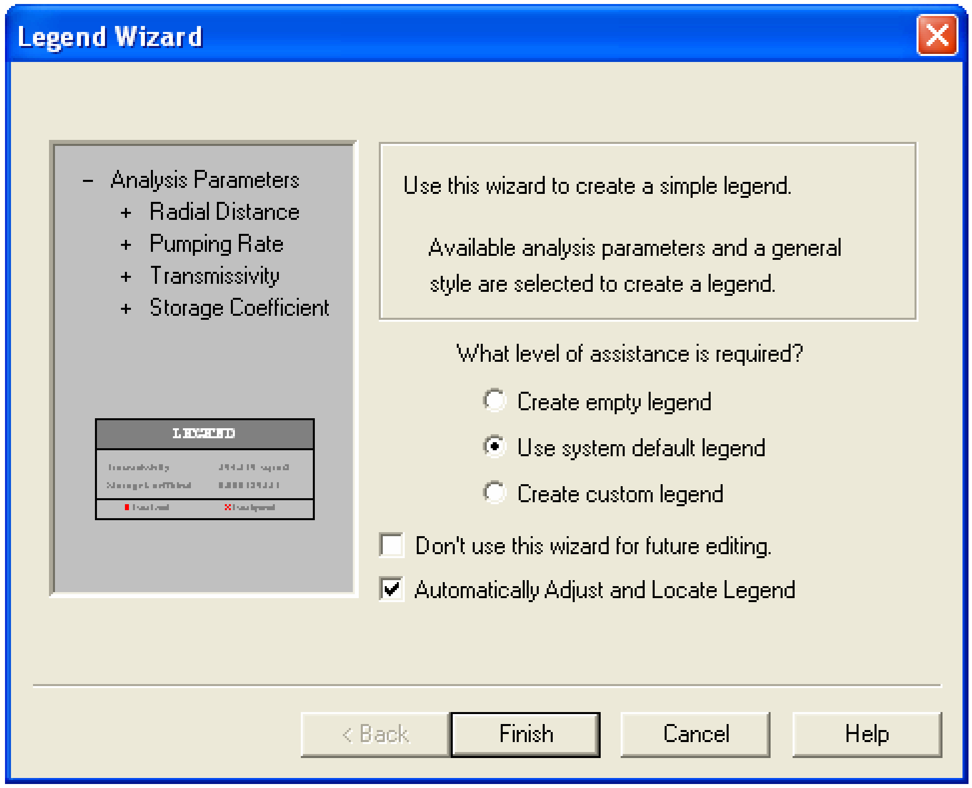
To make a legend as in previous versions, select the **Create empty legend** radio button and click the **Finish** button; double click on the newly created legend box and set it up as you see fit.

The legend generated by the wizard was patterned after the example legends created in the example files shipped with previous versions of the program. The specific legend created using the following example wizard screens is below.

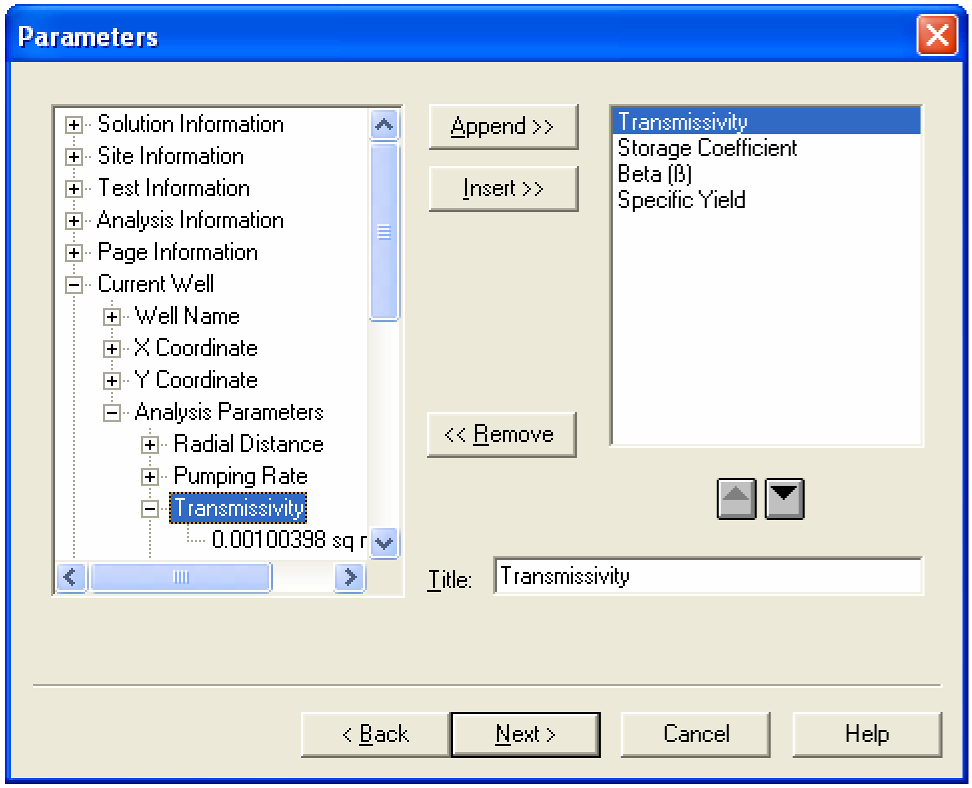


The legend has three sections, the Title sublegend, the Parameter sublegend and the Symbol sublegend. The Title and Symbol sublegends are optional as is the bitmap in the Title sublegend.

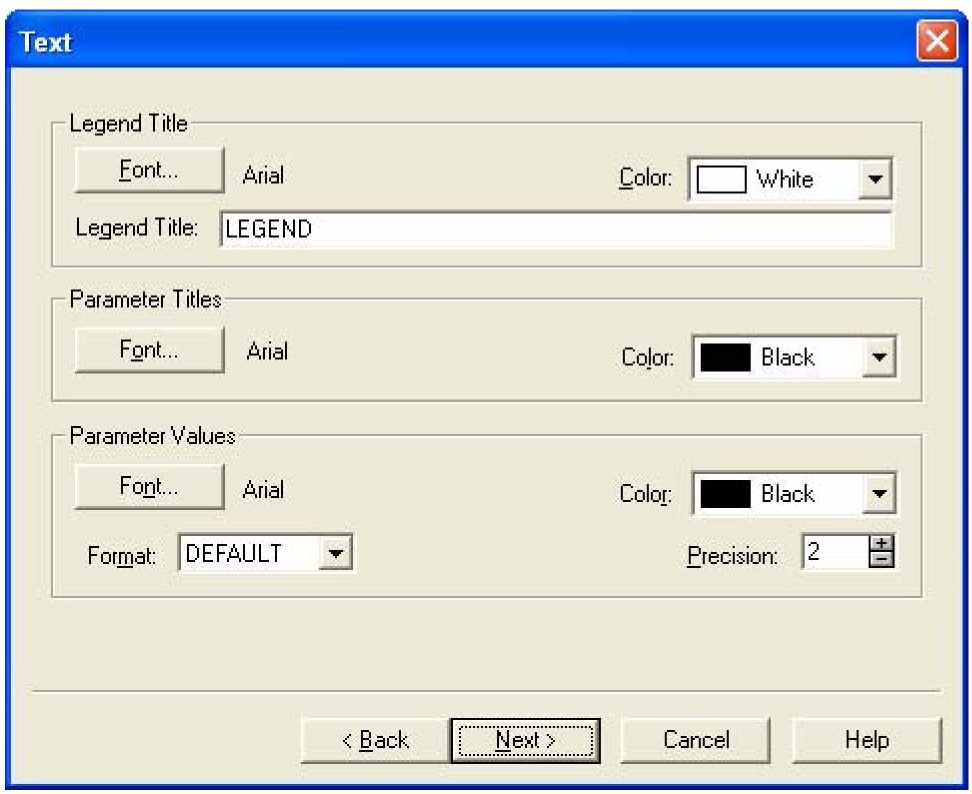
To customize the legend created with the wizard, click the **Create custom legend** radio button and the **Finish** button becomes a **Next** button to continue the customization. Two other options are available via check boxes. The **Don't use this wizard for future editing** option overrides the default behavior that, when you double click the legend to edit it, the wizard will not be used. The **Automatically Adjust and Locate Legend** check box controls whether the system will automatically maintain the location of the legend.



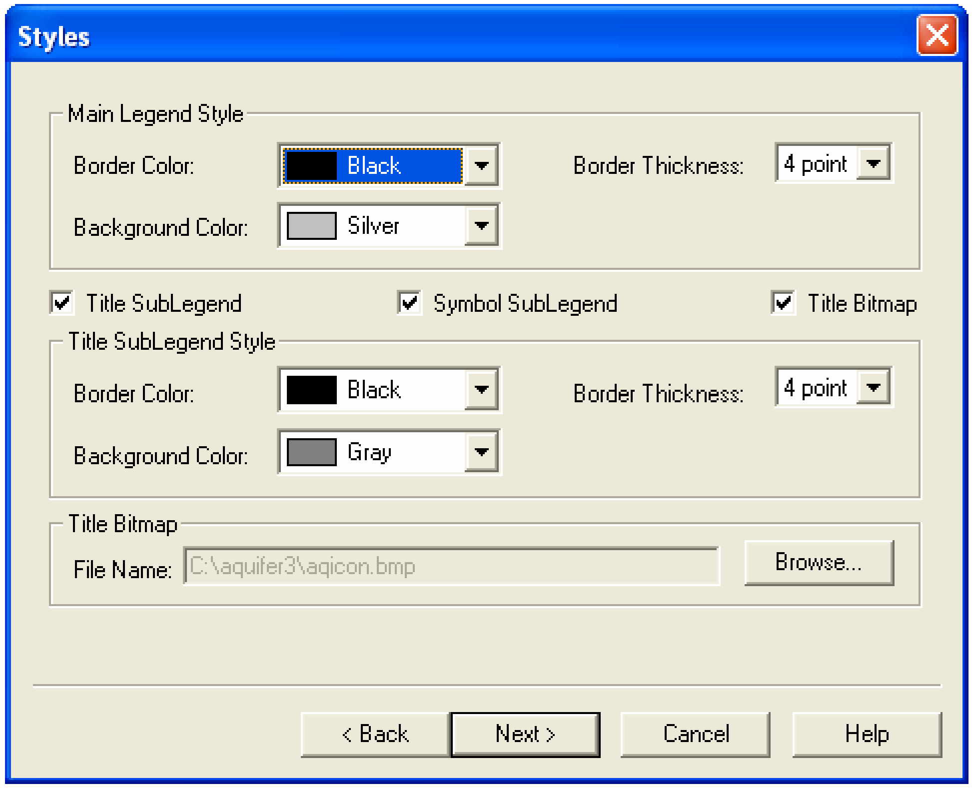
If you chose to create a custom legend, the next step in the wizard enables you to choose the items you want added to the legend. The ***Parameters*** step presents a list of parameter items which is dynamic and defined by the active document type, analysis type and view type. As with Parameters in previous versions, you drill down into the options and click on the item you want, in this case Transmissivity. Using the **Append**, **Insert** and **Remove** buttons, you select what you want. You can also change the default title for an item by selecting it in the right list box and entering the new title in the **Title** edit field.



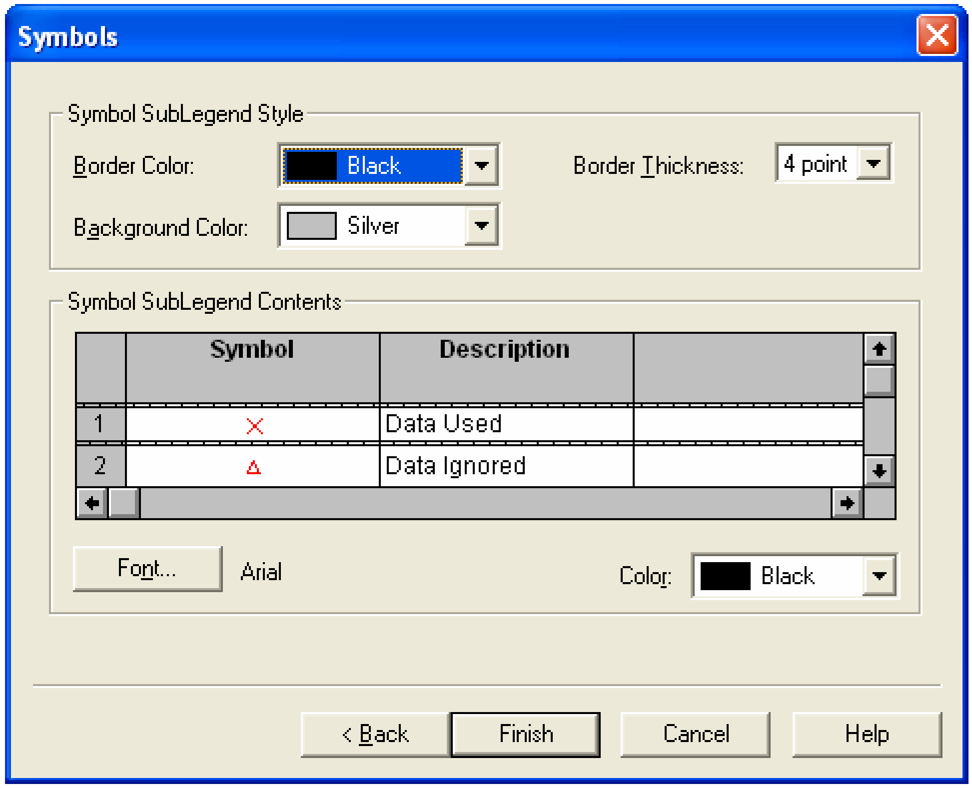
The ***Text*** step of the process controls colors and fonts of the legend title, parameter titles and parameter values. It also controls the overall format and precision for parameter values.



The ***Styles*** step defines the layout of the legend including sublegends, border colors and thicknesses, background colors and title bitmap.



The ***Symbols*** step is only displayed when the **Symbol Sublegend** checkbox has been checked in the previous step. You can control the contents of the symbol sublegend.



On Screen Editing of Legends

If you have created a legend using the legend wizard and have selected the **Automatically adjust and locate legend** option any changes made on screen will be automatically undone when the screen refreshes. The on screen editing features are for older legends, those created as empty legends using the wizard or those that have selected the **Don't use this wizard for future editing** option.

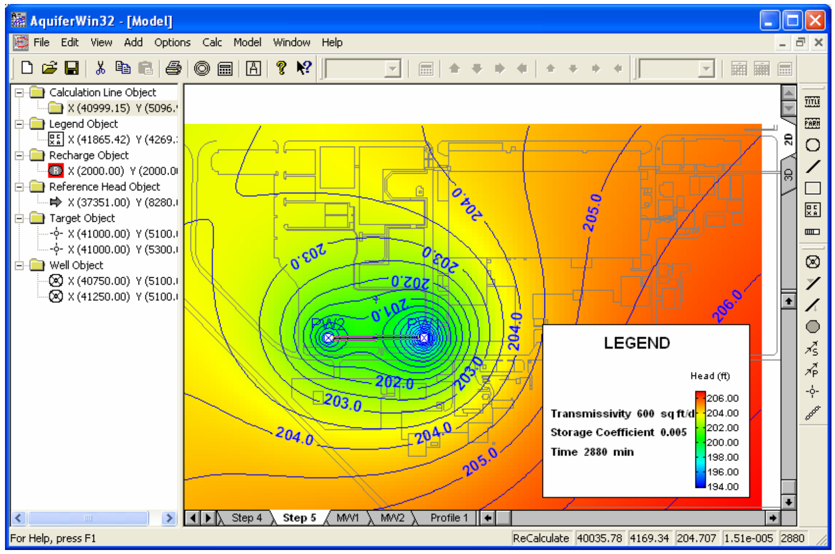
On screen manipulation now drills into legends no matter how deeply they are nested to allow the typical on screen manipulations to be performed. In addition, a context menu has been created for legends that allows for Cut/Copy/Paste/Select All/Delete/Add/Add to Legend operations within the legend.

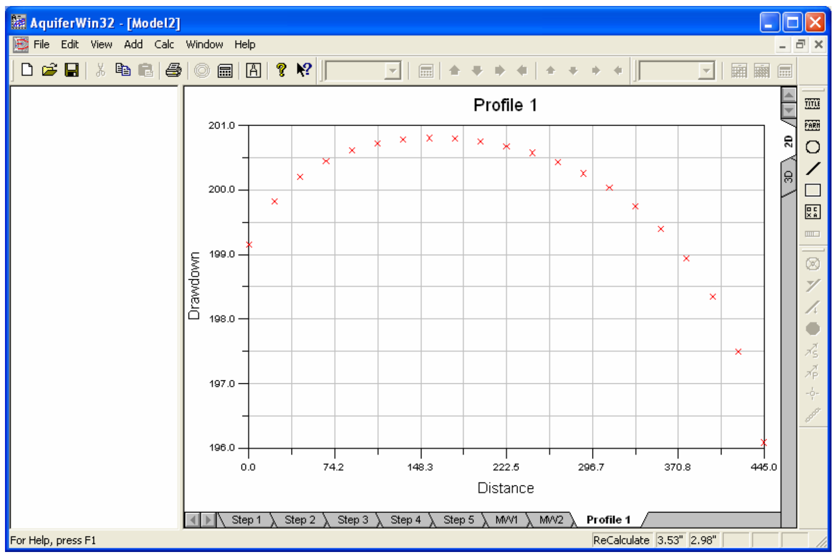
Of particular note is the **Add to Legend** menu. When items have been selected in the main view, they can be added to the legend using this menu. In most cases, you will want to drag the object into the legend and position it before using this option. You can, however, attach items to the legend that are not located inside the legend. When the legend is moved, they maintain their relative position to the legend.

Line Calculations and Distance/Drawdown graphs

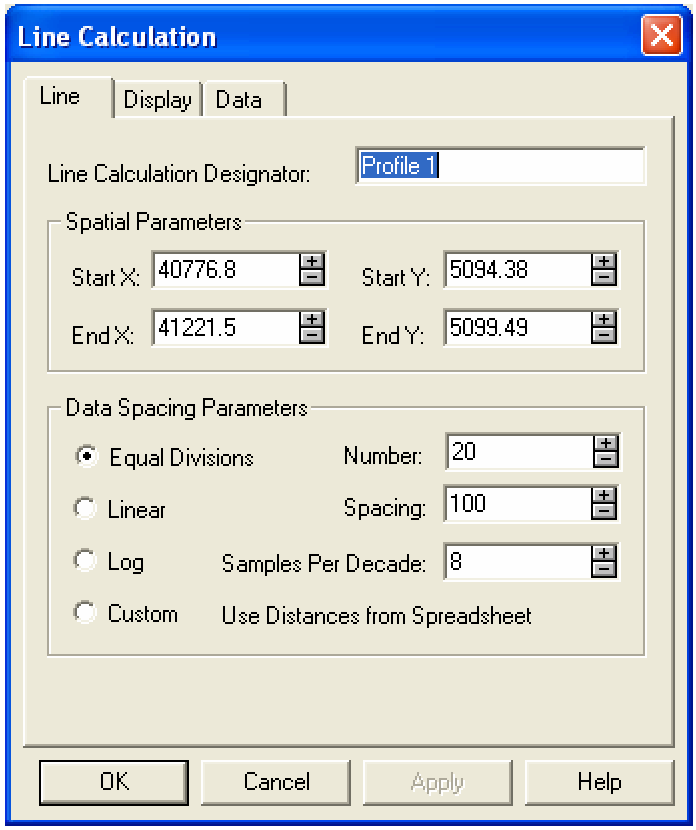
A line calculation is simply a line dragged onto the map view in a **Simulation** or **Flow Model** document along which head/drawdown calculations will be made and a graph generated. If one of the end points of the line corresponds with a pumping well, a drawdown versus distance graph can be generated.

Line calculation elements appear on the map with similar characteristics to a line and, when the **View->Well Data** menu is checked, a view tab is added for the graph of drawdown versus distance.





The specific options available for Line Calculations are as follows:



Line Calculation Designator: Each line calculation must have a unique name so that it can be identified on view tabs.

Spatial Parameters

Start X: The x-coordinate of the starting point for the line calculation in map coordinates. Distances are calculated relative to this point.

Start Y: The y-coordinate of the starting point for the line calculation in map coordinates. Distances are calculated relative to this point.

End X: The x-coordinate of the ending point for the line calculation in map coordinates.

End Y: The y-coordinate of the ending point for the line calculation in map coordinates.

Data Spacing Parameters

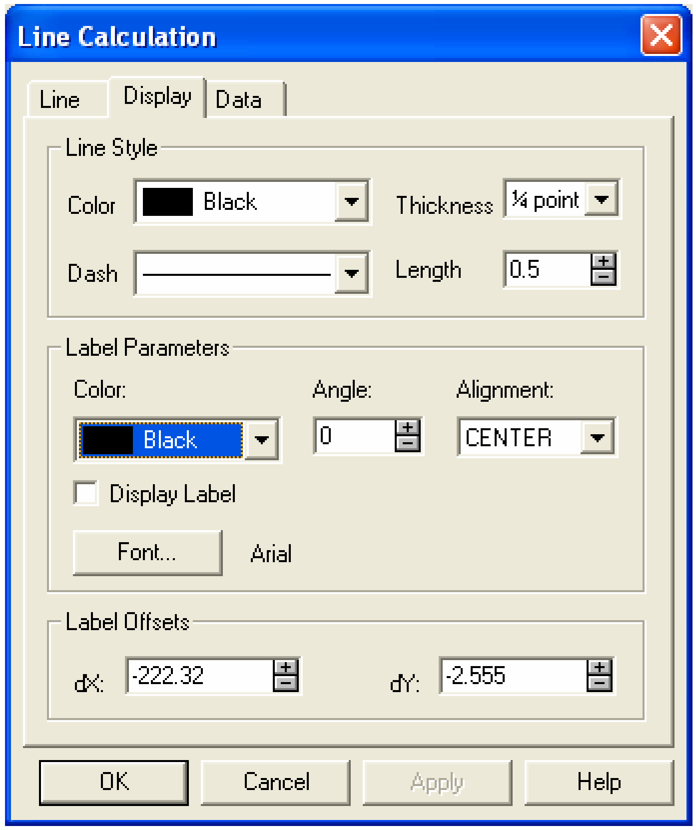
Equal Divisions Data points will be calculated at equal intervals between the start and end points of the line calculation.

Number: The number of equal intervals to calculate between the start and end points of the line calculations.

Linear Data points will be calculated at specified intervals starting at the start point of the line calculation.

Spacing: The spacing, in map units, between adjacent points in the line calculation.

Log	Data points will be calculated at logarithmic intervals starting at the start point of the line calculation.
Samples Per Decade:	The number of logarithmic intervals between adjacent powers of 10 in the line calculation.
Custom	Distances, in map units, entered into the spreadsheet on the ***Data*** tab will be used for the calculations.



Line Style

Color:	Sets the color of the line connecting the data points
Thickness:	Sets the thickness in points of the line connecting the data points
Dash:	Sets the dash pattern to use for the line connecting the data points
Length:	Sets the length in inches of the dash pattern for the line connecting the data points

Label Parameters

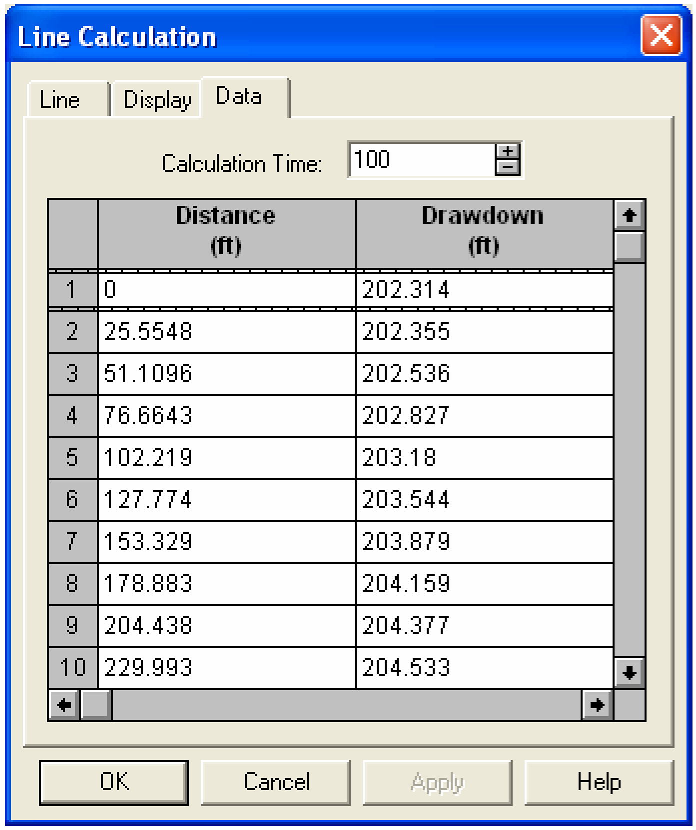
Color	The color to use when displaying the line calculation label
Angle	The angle to rotate the text when displaying the line calculation label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the line calculation designator is used to label the well

Font — Defines the font, font style, size and effects for the label

Label Offsets

dX: — The distance, in map units, to offset the label in x from the x-coordinate of the line calculation center point.

dY: — The distance, in map units, to offset the label in y from the y-coordinate of line calculation center point.

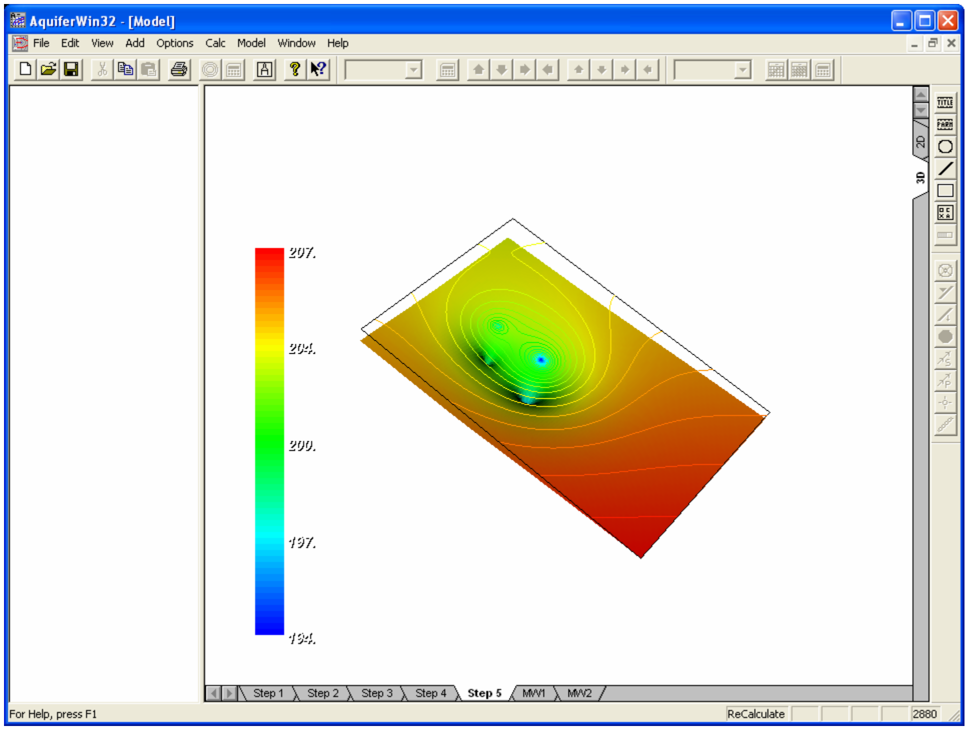


Calculation Time: — The time at which to calculate head/drawdown versus distance data.

Spreadsheet — The data in the spreadsheet reflect the calculated data points. If **Custom** has been selected on the ***Line*** tab, you can edit the number lines and the values of the distances at which to calculate; if **Custom** was not selected, any changes made will be ignored.

3D Perspective

The 3D vertical tab contains the 3D perspective view of the hydraulic head or drawdown values. Contour lines can be optionally displayed as well. All the normal annotations can be added; however, the annotations are fixed in position in the view and are not moved/rotated when the 3D perspective is manipulated.



.

View Manipulation

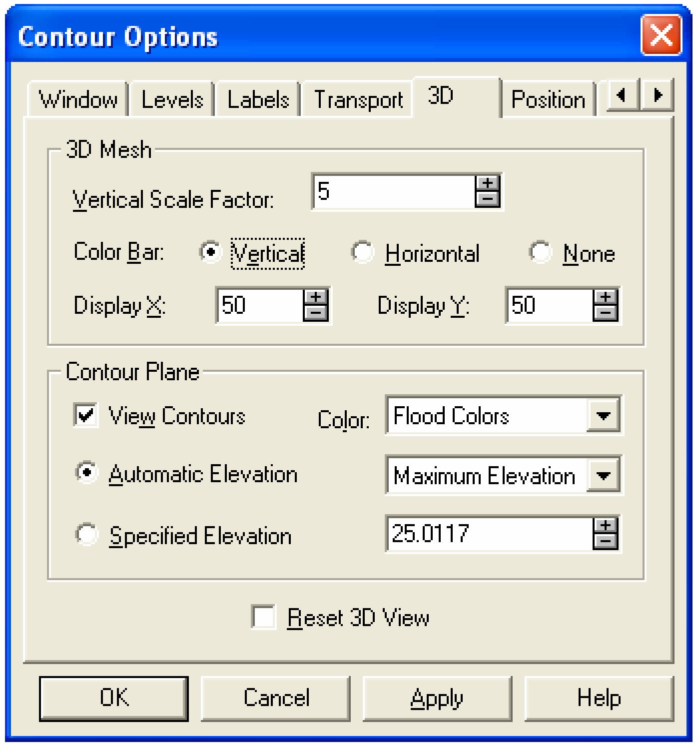
Selecting the **3D Manipulation** menu from the edit field context menu allows the manipulation of the 3D view using the mouse and keyboard. Clicking and/or holding the left mouse button in the view causes the display to rotate in the direction toward the cursor location. Holding the **Shift** button while clicking and/or holding the left mouse button causes the display to move in the direction toward the cursor location.

Clicking and/or holding the right mouse button causes the display to zoom in and zoom out. If the cursor is in the upper half of the view, it will zoom out. If the cursor is in the lower half of the view, it will zoom in. The amount of the zoom is controlled by how far the cursor is from the vertical center of the view.

The **Reset 3D** menu is used recenter the 3D perspective view. This is sometimes required when changes have been made via the Contour Options property sheet or on screen editing.

Contour Options

Four tabs on the *Contour Options* property sheet apply to the 3D view and are described below. The color flood parameters apply to both the contour view and the 3D view.

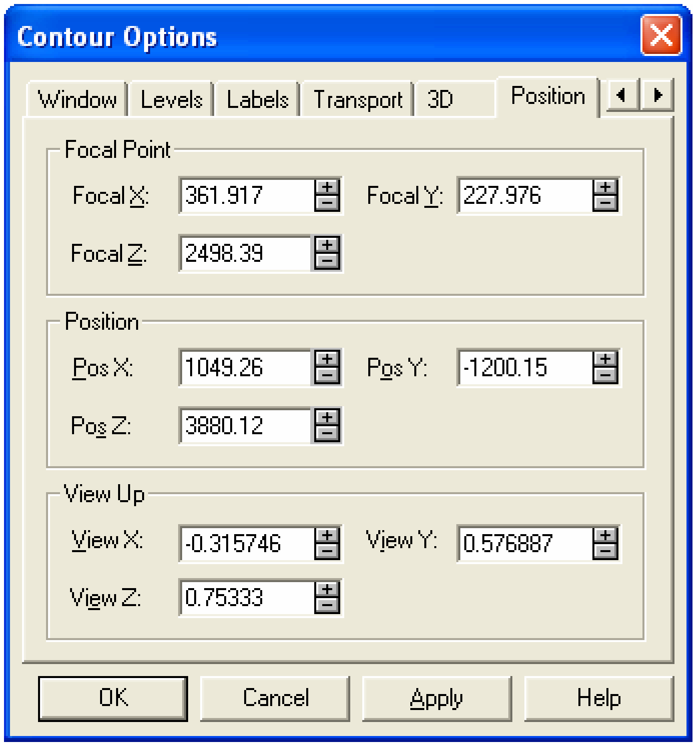


3D Mesh

Vertical Scale Factor:	A multiplier applied to head/drawdown to increase the vertical scale relative to the horizontal scale
Color Bar:	Controls the presence and location of the default scale bar
Display X:	The x-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar
Display Y:	The y-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar

Contour Plane

View Contours:	When checked, a contour map is displayed in the 3D perspective view
Color:	Controls the color of the contours which can be either Black or Flood Colors
Automatic Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked the contour map will automatically be relocated as the contour data changes
Specified Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked, the value entered in the adjacent edit field will be used to locate the contour plane
Reset 3D View	At times, the perspective drawing can leave the field of view. Checking this option will relocate it into view



Focal Point

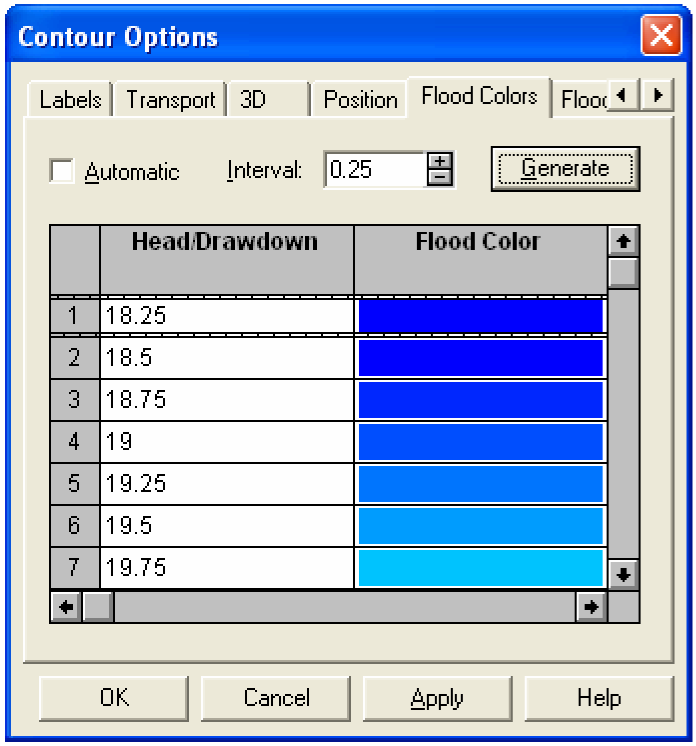
Focal X:	The x-coordinate of the focal point used to display the 3D perspective drawing
Focal Y:	The y-coordinate of the focal point used to display the 3D perspective drawing
Focal Z:	The z-coordinate of the focal point used to display the 3D perspective drawing

Position

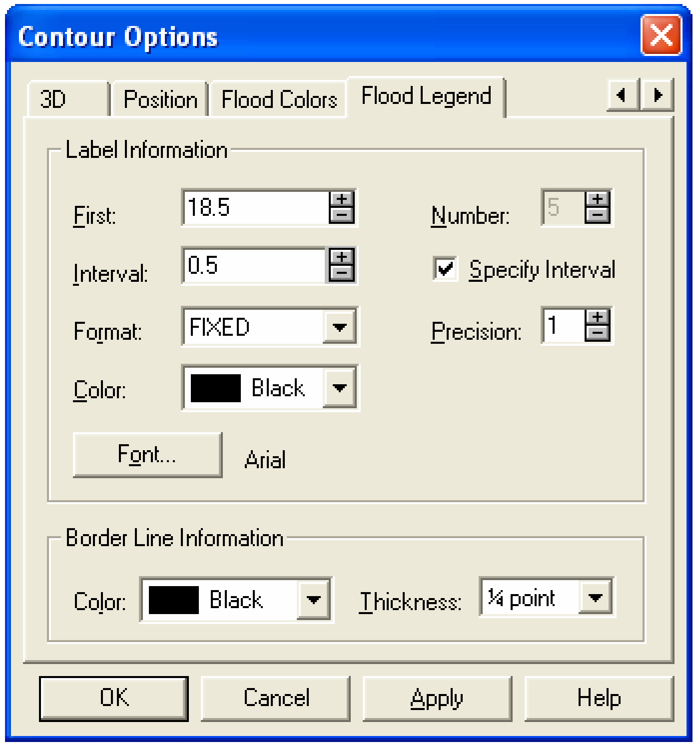
Position X:	The x-coordinate of the camera position used to display the 3D perspective drawing
Position Y:	The y-coordinate of the camera position used to display the 3D perspective drawing
Position Z:	The z-coordinate of the camera position used to display the 3D perspective drawing

View Up

View X:	The x-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Y:	The y-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Z:	The z-coordinate defining the view up direction for the camera used to display the 3D perspective drawing



Automatic	When checked, the distribution of colors is automatically determined by the application
Interval:	The interval to use when generating flood colors for values of head/drawdown to be used when the **Generate** button is clicked.
Generate	When clicked, a new color distribution will be generated using the same algorithm as the **Automatic** setting; however, these can be edited by the user



Label Information

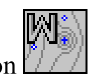
First	Sets the head/drawdown value for the first labeled value in color flood legends
Number:	The number of equally spaced labels to use when labeling the color flood legends
Interval:	The interval between adjacent labels; used when the Specify Interval option is activated
Specify Interval	When checked, the value for *Interval* is used when labeling color flood legends
Format:	The numeric format to use when displaying the labels on color flood legends
Precision:	The number of digits to the right of the decimal point to use when displaying the label on color flood legends
Color	The color to use when displaying labels on color flood legends
Font	Defines the font, font style and size for the label

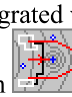
Border Line Information

Color:	The color to use when displaying the border around the color flood legend
Thickness:	The thickness, in points, of the line used when displaying the border around the color flood lege

Getting Started Using the WinFlow Solver

Starting WinFlow

Start WinFlow by simply double-clicking on the WinFlow icon  or activating the **Start->Programs->WinFlow Version 3** menu. If you are unfamiliar with Windows, please consult your Windows manual for common procedures, key strokes, and mouse procedures.

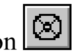
Since WinFlow is well integrated with the Windows shell, you can also double click on a WinFlow data file icon  on either the Desktop or from Explorer. You can also right click the mouse on the desktop and select the **New->WinFlow Document** to open WinFlow and create a new document.

WinFlow Interface

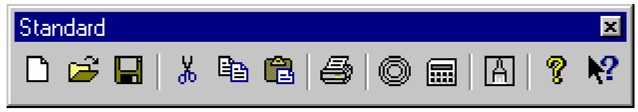
WinFlow uses a Microsoft Windows-compatible graphical user interface (GUI). A menu, part of the GUI, is displayed at the top of the WinFlow window. All modeling operations are controlled from this menu. The majority of the window below the menu is the design area. The models you create are displayed below the menu. You may have multiple model windows open in the design window. The maximum number of models that may be open at one time depends upon the complexity of the models and the amount of memory in your computer.

You have two options to select a menu item - mouse or keyboard. To use the keyboard, select a main menu item by pressing the **ALT** key and the underlined letter of the menu item. Pulldown menu items are then selected by pressing the underlined letter of the pulldown menu item. For example, to open an existing WinFlow model, press **ALT-F** (**File**) and then press **O** (**Open**). To use the mouse, simply click the left mouse button when the cursor is over the desired menu item.

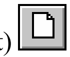
Toolbars are displayed below the main menu and contain icons that represent key WinFlow operations or commands. Using a toolbar button is a shortcut to the menu system. To execute one of these shortcuts, use the mouse to move the cursor over

the toolbar icon and click on the Toolbar button. For example, clicking on  adds a well to the model.

The available toolbars are shown below:

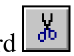


The Standard Toolbar icons represent (from left to right):

Create a new model (in MDI terminology, a document) 

Open an existing WinFlow data file 

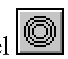
Save the current model to a file 

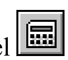
Cut the currently selected item to the clipboard 

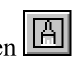
Copy the currently selected item to the clipboard 

Paste from the clipboard 

Print the current model 

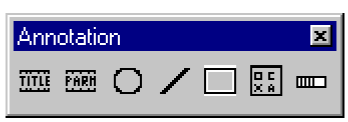
Recontour the current model 

Recalculate the current model 

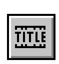
Refresh the current model screen 

Display program information (About) 

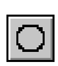
Help 

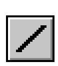


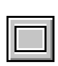
The Annotation Toolbar icons represent (from left to right):

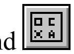
Add a title 

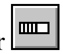
Add a parameter 

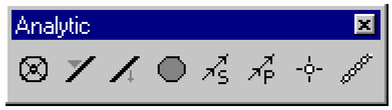
Add a symbol 

Add a line 

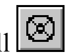
Add a frame 

Add a legend 

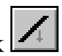
Add a scale bar 

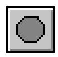


The Analytic Toolbar icons represent (from left to right):

Add a well 

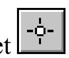
Add a head linesink 

Add a flux linesink 

Add a pond 

Add a streamline 

Add a particle trace 

Add a target 

Add a line calculation 

Use the mouse to move the cursor over these toolbar icons and a description of the icon is displayed in the status bar in the lower-left corner of the screen. The status bar also displays the X- and Y-coordinates in the lower-right corner of the screen. The model head (water level) is displayed in the lower-right corner if the model has been calculated.

You may choose to remove the toolbars and/or the status bar from the WinFlow window using the **View** menu. Pull down the **View** menu and click on **Standard Toolbar** to remove the Standard toolbar. Repeat the procedure to have it redisplayed. The specific toolbar is displayed when a check mark appears next to the appropriate **View** menu item. The status bar works the same way.

In addition to using the mouse, you may also use the arrow keys on the keyboard to move the cursor around the model design window. Pressing an arrow key moves the cursor a certain distance (set by selecting **Options->Map**). Other key combinations are used to simulate using the mouse, as described below:

Keyboard Action	What Happens?
arrow keys	cursor moves in direction of arrow
shift-arrow keys	cursor moves one pixel at a time for fine adjustment
space bar	equivalent to mouse double-click
enter key	equivalent to mouse click (left button)

shift-Enter	add another element to those selected; also used to start resizing of a line or circle if such an element has been selected and the cursor has changed to a two-prong or four-prong arrow.
ctrl-arrow	equivalent to **View->Page**
alt-arrow	equivalent to **View->Scroll**
shift-mouse click	add an element to those selected

Applying WinFlow

WinFlow is a powerful tool for analyzing two-dimensional groundwater flow problems. A flexible user interface using pull-down menus and tab dialogs (property sheets) provides you with an easy method of setting up the model and rapidly getting to a solution. The analytical model is developed in several steps, as follows:

Digitize a base map in a CAD package such as AutoCAD.

Export the drawing to a DXF format file.

Double-click on the WinFlow icon to run WinFlow.

Load the digitized map into WinFlow.

Specify aquifer parameters.

Choose a solution type (steady-state or transient).

Add analytic elements to the model (wells, ponds, etc.).

Calculate the model and contour the results.

Add streamlines or particle-traces to define flow paths.

Create a graphical or summary output of the results.

These steps can be performed in a matter of minutes after the base map has been digitized. The base map is optional; however, it provides you with a frame of reference in designing the analytical model.

Data Requirements

WinFlow requires you to specify a few simple pieces of information or data to define the analytical model. Data can be classified in four different ways: (1) fundamental data required by all problems; (2) data required for only transient applications; (3) data required for particle-tracking analysis; and (4) optional data.

Fundamental Data are required for all analytical models created by WinFlow. These data include:

regional gradient and direction of flow,

hydraulic conductivity,

aquifer top elevation and bottom elevation, and

reference head.

Regional gradient and direction of flow are used to superimpose a uniform groundwater flow field on the analytical model. You must define the regional gradient which has units of [L/L] (dimensionless) and the direction of flow. The direction of flow is entered in degrees, with 0.0 degrees representing east, 90.0 degrees representing north, etc. You may enter a gradient of 0.0. You may want to do this if you are computing drawdowns. Note that in unconfined aquifers, the gradient is defined at the reference point and may change throughout the model as the saturated thickness (and hence the transmissivity) changes.

Hydraulic conductivity is assumed to be homogeneous throughout the infinite aquifer and has units of [L/T], e.g. ft/d. The aquifer top and bottom elevations have units of length [L], e.g. ft, and are used to compute transmissivity. In addition, the steady-state module allows for conversion to unconfined flow. Therefore, if the head falls below the top of the aquifer the model becomes unconfined.

The reference head defines a point where the head is known. In the steady-state model, the reference head is always constant and never changes during simulations. The reference head may or may not be constant in the transient model, depending upon a user-selectable option. All computations are based upon the reference head, which should be located as far from wells, ponds, etc. as possible. The reference head is analogous to a constant head in a numerical model.

WinFlow allows you to use any set of units you would like; however, using consistent units throughout the model may minimize confusion. For example, if you are using length [L] units of feet and time [T] units of days, hydraulic conductivity could be expressed in units of ft/d and pumping rates could be in units of ft^3/d.

Transient flow problems require three additional data types, including

storage coefficient [dimensionless],

Hantush leakage factor (denoted by L or B) [L], and

time steps used to compute solution [T].

The Hantush Leakage factor, denoted by the letter L or B, has units of length (Hantush, 1956). The leakage factor is only required for leaky aquifers. It should be set to zero for confined flow.

Particle-tracking analyses require that porosity be defined. The porosity (dimensionless) is used to compute the average linear groundwater flow velocity at discrete points within the system.

Optional data types define the analytic elements of the model. These elements include the following information:

well locations and pumping rates,

recharge rate,

pond locations and recharge/discharge rates,

linesink locations and recharge/discharge rates, and

calibration targets.

All of the elements above are available for the steady-state model and all but recharge are simulated by the transient model.

Wells are defined by the coordinates of the center of the well, a pumping rate, and a well radius. The pumping rate has units of $[L^3/T]$, such as ft^3/d. A positive pumping rate indicates production and a negative rate indicates injection.

Recharge is defined only for steady-state models and has units of [L/T], such as ft/d. The recharge is distributed over the entire infinite plane of the model. An ellipse

defines the shape and position of the water-table mound created if there is no regional gradient. You must define the center coordinates of the ellipse, the length of the a- and b-axis of the ellipse, and the angle between the a-axis (long axis) of the ellipse and the x-axis of the model. The recharge rate should be a positive value.

Ponds are circular recharge or discharge areas. Ponds are defined in a manner similar to wells. The coordinates of the center of the pond are given, along with the radius of the pond. The pond infiltration rate has units of [L/T] and is computed by dividing the total discharge rate into or out of the pond by the area of the pond. Thus, the pond infiltration/discharge rate has the same units as recharge. A positive rate indicates infiltration and negative indicates discharge from the aquifer.

Linesinks are linear recharge or discharge features. The linesink is defined by providing the coordinates of each end point and either an infiltration/discharge rate or a head. If the linesink is defined in terms of head, the model will compute a discharge rate based on the given head value (assumed to be at the center of the linesink). Linesink discharge/infiltration rates have units of [L^2/T], which is computed by dividing the total discharge rate by the length of the line. Thus, the linesink rate (or strength) is a rate per unit length. The sign convention for linesinks is the same as wells: positive indicates pumping or production and negative signifies injection or infiltration.

After defining all aquifer properties and analytic elements, you run the model by selecting **Calc** from the main menu and **Recalculate** from the pull-down menu. WinFlow will now compute hydraulic head on a regular grid of points and contour the results using ten contour intervals. You may change the contour settings by selecting **Options** from the main menu and **Contour** from the pull-down menu and **Paramers** from the submenu.

Starting a New Model

The first step in using WinFlow to solve a two-dimensional ground-water flow problem is to digitize a site map. While this step is not required, it does help you by providing a convenient frame of reference. WinFlow map files are similar in format to QuickFlow map or ModelCad™ map files and may be created by first using any CAD package (such as AutoCAD) to digitize the map. Next, export the CAD drawing as a DXF (Drawing Interchange Format) file.

Double-click on the WinFlow icon to start the model. You translate the DXF map file into a WinFlow map file within WinFlow by first opening a new model document. Select **File** from the main menu and **New** from the pulldown menu. You may also click on the **New Document** button on the Toolbar. Now that you have a model to work with, select **File** from the main menu, **Map** from the pulldown menu. Set the *Files of Type:* selection on the resulting dialog to “DXF Files (*.dxf)”. Select your DXF file using the standard Windows file dialog. You must now select a new file name for the map file that will be created (again using a standard Windows file dialog).

Finally, you will be asked for one additional item, a multiplication factor. The multiplication factor is used to scale the drawing to model units. Normally, the multiplication factor will be 1.0; however, if your CAD system exports the DXF file in units of inches, this factor would be 1.0/12.0 or 0.08333333. You may need to use a multiplier other than 1.0 in AutoCAD if you are using engineering units instead of decimal units. If the map units look wrong in WinFlow, it is probably in units of inches. You must then re-import the DXF file with a new multiplication factor.

It is easy to check the units on the imported map by moving the mouse around the map and looking at the X and Y coordinates displayed in the lower right corner of

the status bar. Choose two points that are a known distance apart and check the distance with the status bar. If the distance is incorrect, compute a multiplication factor to use in the map import and start again.

The DXF file import feature first translates the DXF file into a WinFlow map file and displays the map on your screen. The map file will be displayed the next time you open this WinFlow model (assuming that you save it later).

When starting with a map that is already in the proper WinFlow map file format, simply select **File** from the main menu, **Map** from the pulldown menu, and set the *Files of Type:* selection on the resulting dialog to "Map Files (*.map)". Select the proper file name, and the map is displayed on your screen.

Now that the map has been loaded, you must define the aquifer parameters. Default parameters have been defined by WinFlow, however, these may not be the proper ones for your problem. To change these parameters, select **Edit** from the main menu and **Model** from the pulldown menu. The resulting property sheet contains tabs or pages on which you can define the following prarmeters:

hydraulic conductivity,

aquifer bottom elevation,

aquifer top elevation,

reference head value,

regional hydraulic gradient,

recharge rate,

porosity,

storage coefficient (transient models),

leakage factor (transient models), and

If you select the ***Reference Head*** tab or **Reference Head** from the **Edit** menu, a dialog appears that prompts for the reference head value, gradient, and angle of the gradient. A gradient angle of 0.0 degrees represents east, 90 degrees is north, etc. You may change the location of the reference head by dragging the reference head to a new location. You do this by moving the cursor over the reference head symbol until the move cursor appears. Now, hold down the left mouse button and move the cursor to the desired location. Release the mouse button to place the reference head at the new location.

You must always define a reference head! You do not necessarily need to define a gradient, however. If no gradient is defined, WinFlow will be simulating an "impact" model using a flat water-table. If the reference head is zero and the gradient is zero, WinFlow will compute drawdown. In this case, be sure that the bottom of the aquifer is set below zero also.

For a transient model, click the **Options** from the main menu and **Transient** from the pull-down menu. Click on the ***Time Steps*** tab and set up the time steps you want calculated. Each time step will appear as a tab along the bottom of the map view.

After confirming that all aquifer parameters have been properly set, select a solution type - steady-state or transient. If the solution currently selected does not support both, one will be disabled. The default solution type is steady-state. To change the solution type, select **Model** from the main menu. Now select either **Steady-state** or **Transient** from the pull-down menu. A check-mark is displayed next to the selected model type.

If you want to display contours representing the uniform regional gradient, you may now select **Calc** from the main menu and **Recalculate** from the pull-down menu. WinFlow will display the status of the computations in a status dialog. After calculating the head at a regular grid of points, a contour map will be displayed using a default contour interval. If you want to change the appearance of the contours, select **Options** from the main menu and **Contour** from the pull-down menu. A third menu will appear with two options for controlling the contouring, **Window** and **Parameters**. Use **Window** to change the size or location of the area that is contoured and **Parameters** to change the contour interval and font.

Adding Model Features

Wells, linesinks, ponds, streamlines, and/or particle-traces may be added to the model. To add one of these features to the analytical model, select **Add** from the main menu and select the desired feature from the pull-down menu. You must then move the cursor around the design screen to locate the item. This is accomplished using the cursor keys or the mouse. To alter the distance that the cursor moves after pressing an arrow key, select **Options** from the main menu and **Map** from the pull-down menu and **Parameters** from the second pull-down. The cursor step size is defined in model units of length, e.g., feet.

Wells are defined by moving the cursor to the desired coordinates or location and pressing the Enter key or left mouse button. You will then be prompted for a name, the well construction information and a pumping rate [L^3/T]. The name entered must be unique across all wells in the model. You will also be permitted to edit the well coordinates. If you change the coordinates, the well will be redrawn at the correct location. Remember that a positive pumping rate indicates production, while a negative rate indicates injection. There is no set limit to the number of wells or any other element that may be added to the WinFlow model. The actual limit depends upon the amount of memory in your computer.

Linesinks are defined by pressing the left mouse button at the beginning of the linesink. While holding down the left mouse button, move the cursor to the end of the linesink and release the button. You will then be prompted for a name, and the head or flux of the linesink. If you have selected a **Head Linesink**, WinFlow computes the flux required to maintain that head at the center of the linesink. If you have selected **Flux Linesink**, the flux is entered in units of [L^2/T]. This is normally computed by dividing the total flux rate [L^3/T] by the length of the linesink [L]. A negative flux indicates injection and positive indicates production. As with the wells, you also have the opportunity to edit the coordinates and name of the linesink.

Ponds are defined by selecting the center of the pond and dragging a circle to the desired radius while holding down the left mouse button. A property sheet will appear that allows you to edit the center coordinates, the radius, infiltration/discharge rate, and name. The pond rate is in units of [L/T] in the same manner as recharge. A positive pond flux indicates recharge, while negative signifies discharge. Note that the sign convention for ponds is opposite of wells and linesinks.

Streamlines are only computed for the steady-state model. You have the option of adding a single streamline, several streamlines started along a line, or several streamlines along a circle. If you choose the line or circle option, you will then enter the number of streamlines to add. These will be added at even increments along the line or circle. When you add a streamline, the point you select is the beginning of the streamline, which is drawn downgradient of that point. When you change the model in any way, all streamlines are recomputed during the recalculation of the model. Arrow heads may be added to the streamline by selecting **Options** from the main menu and **Trace** from the pulldown menu. You define the distance between

arrowheads and the size of the arrow (in model units of length) on the ***Streamline*** tab.

Particle-traces may be computed for either steady-state or transient models and are defined in the same manner as streamlines. Particle-traces take longer to calculate than streamlines, however. The particle-trace differs from the streamline in several ways: (1) the particle-trace is computed numerically instead of semi-analytically; (2) a time value may be posted on the resulting trace; and (3) particle-traces may be tracked upgradient (reverse particle-tracking). Options controlling the appearance and computation of particle-traces are modified by selecting **Options** from the main menu and **Trace** from the pull-down menu.

All of these analytical features may be added from the Analytic Toolbar, as described in the previous section. Simply click on the proper toolbar button. As the cursor is moved over the toolbar, a text is displayed in the status bar to help you remember which icons go with which elements.

Editing Analytic Elements

You may edit any analytic element on the map (well, linesink, pond, title, scale bar, etc) by moving the cursor over the item and double clicking. Make sure that the cursor changes to four arrows pointing up, down, left and right before double clicking. A property sheet will appear that allows you to edit the parameters associated with that element (e.g. pumping rate). To move the element, hold down the left mouse button and drag the item to a new location.

The size of linesinks and ponds may be changed by moving the cursor to the edge of the pond or end of the linesink. When in the proper location, the cursor will change to one with two arrows pointing left and right. Hold down the left mouse button and drag to a new size.

Computing the Model

If you have added a feature, edited a feature, or changed any aquifer property, WinFlow will display "ReCalculate" in the status bar. This means that the contour map on the screen may no longer reflect the changes that you have made. To recompute, select **Calc** from the main menu and **Recalculate** from the pull-down menu. If you have only changed contouring options, you may select **Recontour** from the pull-down menu. You may also use toolbar buttons to recalculate and recontour the model.

When WinFlow recalculates the problem, it displays the percent completion in a status dialog. The time required to recalculate depends on the speed of your computer and the number of features that you have added to the model. Recalculation normally only takes a matter of seconds or a few minutes. After recomputing the model, WinFlow will recalculate the locations of any streamlines and/or particle-traces that you have added to the model. Finally, WinFlow will recontour the results. Slide bars are displayed during each operation so that you can judge how long it will take to finish.

Displaying the Results

WinFlow displays the results from model calculations using contours, streamlines, and particle-traces. By default, the design window defines the area that will contain the contours. You may change the default by selecting **Options** from the main menu and **Contour** from the pull-down menu. A third menu will display two options. The **Window** selection allows you to drag a rectangular box to define the new contour

region. The **Parameters** selection on this third menu provides ways to change the labeling of the contours and the contour interval. If you make any changes to these options, a dialog will prompt you to recontour the results. If you have redefined the contour window, you must select **Recalculate** from the **Calc** menu.

You may change the current view of the model by selecting **View** from the main menu. A pull-down menu allows you to zoom on a point (**Zoom**), zoom on a window (**Window**), display the full map and contours (**Full**), scroll the screen a given distance in a certain direction (**Scroll** and **Left**, **Right**, **Up**, **Down**), or to simply redraw or refresh the current screen (**Refresh**).

Other options that control the appearance of the model results are found under the **Options** main menu selection. The **Map->Parameters** selection lets you change most of the colors and object sizes related to the design screen. **Contour->Parameters** allows you to change the way contours are produced, as described above.

Creating Output

WinFlow creates graphical and summary output in several different ways, including the following:

Print to any selected Windows device;

Create files that are compatible with SURFER^{TM}, Geosoft, or Spyglass;

Create standard Windows Metafiles (wmf);

Create ASCII text files containing X- and Y-coordinates and head (XYZ);

Create DXF file for importing into CAD systems; and

Create an ASCII summary output file.

Any of these options can be selected from the **File** option on the main menu. Select **Print** to send the plot to the current Windows printer. Select **Export** to create the other files listed above.

A common problem when printing to the current Windows printer is that the text for contour labels is not properly rotated. To correct this problem, select **File** from the main menu and **Printer Setup** from the pulldown menu. Most printer drivers have an option to "Print Truetype Fonts as Graphics" or something similar. Make sure that this option is switched ON. If this does not correct the problem, you may need an update for your printer driver. Contact your printer manufacturer or Microsoft for a new one. Many driver updates are also available on the internet.

Another problem can occur in getting the plot to fit on a single page. Select **Page Setup** from the **File** menu and click on the ***Scale*** tab and select "Scale to Page" in the *Map Scale* section. This will force the next printout to fit on a page. There are two other check boxes on the dialog, labelled "Display Window" and "Map Window". Placing a check in "Display Window" causes only what is currently displayed on the screen to be printed. Placing a check in "Map Window" causes the currently defined map window to be printed. The current Map Window is the area defined when you select **View->Full->Screen**. You may change the size of the map window by using **Options->Map->Window**.

DXF files created by WinFlow are compatible with AutoCAD and most other CAD software, such as Design-CAD. All entities, such as wells, contours, base map, streamlines, etc. are stored as separate layers in the DXF file. This allows for easy editing of the CAD drawing.

WinFlow creates up to three different SURFER files. The regular grid of head values computed by the analytical model is stored in a SURFER grid file that can be displayed by the TOPO and SURF programs. The grid file can also be used in the GRID program to make modifications. If you are using a base map or if you have added streamlines or particle-traces, a SURFER XYLINE file may be created. This file can be read by the TOPO program to display a digitized map on the contours. If you have generated time postings on the particle traces, a posting file may be created for use in TOPO.

In order to maintain a QA/QC record of your model run, a summary file may be created. The summary file contains a listing of all data making up the analytical model, the names of all files used, the current date, and a summary of all computations.

Saving a WinFlow Model

Eventually, you will want to save your WinFlow model in a data file for later use. If you have previously saved the file, select the **File->Save** menu; if not, select the **File->Save As** menu and give the file a name. The file created is totally self-contained and, unlike previous versions of WinFlow, contains the base map information as well.

Applying WinTran

WinTran is a powerful tool for analyzing two-dimensional groundwater flow and contaminant fate and transport problems. A flexible user interface using pull-down menus and simple dialogs provides you with an easy method of setting up the model and rapidly getting to a solution. The analytical model is developed in several steps, as follows:

Digitize a base map in a CAD package such as AutoCAD.

Export the drawing to a DXF format file.

Double-click on the WinTran icon to run WinTran.

Load the digitized map into WinTran.

Specify aquifer parameters.

Specify transport and time-stepping parameters.

Add analytic elements to the flow model (wells, ponds, etc.).

Add sources of contamination and/or initial contaminant distribution.

Calculate the model and contour the results.

Add streamlines or particle-traces to define flow paths.

Create a graphical or summary output of the results.

These steps can be performed in a matter of minutes after the base map has been digitized. The base map is optional; however, it provides you with a frame of reference in designing the analytical model.

Data Requirements

WinTran requires you to specify a few simple pieces of information or data to define the analytical model. Data can be classified in four different ways: (1) fundamental data

required by all problems; (2) data required for only transient applications; (3) data required for particle-tracking analysis; and (4) optional data.

Fundamental Data includes information for both the flow and transport model as noted below:

flow model

regional gradient and direction of flow,

hydraulic conductivity,

aquifer top elevation and bottom elevation, and

reference head.

transport model

longitudinal and transverse dispersivity,

porosity,

diffusion coefficient,

contaminant half-life, and

retardation coefficient.

Flow Model Data

Regional gradient and direction of flow are used to superimpose a uniform groundwater flow field on the flow model. You must define the regional gradient which has units of [L/L] (dimensionless) and the direction of flow. The direction of flow is entered in degrees, with 0.0 degrees representing east, 90.0 degrees representing north, etc. You may enter a gradient of 0.0. You may want to do this if you are computing drawdowns. Note that in unconfined aquifers, the gradient is defined at the reference point and may change throughout the model as the saturated thickness (and hence the transmissivity) changes.

Hydraulic conductivity is a coefficient of the aquifer that defines the ease at which water moves through the medium. Hydraulic conductivity is assumed to be homogeneous throughout the infinite aquifer and has units of [L/T], e.g. ft/d.

The **aquifer top and bottom elevations** have units of length [L], e.g. ft, and are used to compute transmissivity. In addition, the flow model allows for conversion to unconfined flow. Therefore, if the head falls below the top of the aquifer the model becomes unconfined.

The **reference head** defines a point where the head is known. In the steady-state model, the reference head is always constant and never changes during simulations. All computations are based upon the reference head, which should be located as far from wells, ponds, etc. as possible. The reference head is analogous to a constant head in a numerical model.

Transport Model Data

Dispersivity is a scale-dependent parameter which is generally larger as the scale of the contaminant plume increases. A typical rule of thumb is that the dispersivity is 10 percent of the length of the contaminant plume (National Research Council, 1990). However, values of dispersivity reported in the literature range generally range from 1 to 100 percent of the problem scale (Gelhar, 1986). There are two values of dispersivity used in the WinTran transport model, longitudinal and transverse. Longitudinal dispersivity represents the spreading of the contaminant

plume in the direction of groundwater flow. The transverse component represents spreading perpendicular to the flow direction. Usually, the longitudinal dispersivity is 5 to 10 times higher than transverse.

Porosity is used to compute the average linear groundwater velocity in WinTran. The porosity value entered in WinTran is actually the effective porosity and defines the ratio of connected void space to the volume of aquifer material. The porosity is used in computing velocity according to the following equation:

$$V = \frac{K}{n}\frac{dh}{dl}$$

where V is the average linear groundwater velocity (L/T), K is the hydraulic conductivity (L/T), n is the effective porosity, and dh/dl is the groundwater gradient.

Molecular diffusion is the spreading of a contaminant in water due to concentration gradients. That is, dissolved contaminants will spread in water from areas of high concentration to areas of lower concentration. This process is caused by random movement of molecules in a fluid. The coefficient of molecular diffusion (or simply the **diffusion coefficient**) is expressed in units of L^2/T (e.g., cm^2/s) and is often assumed to equal zero in advective-dominated transport. Only in very slow-moving groundwater is diffusion important. Bear and Verruijt (1987) estimate the diffusion coefficient to be approximately 1 x 10^{-5} cm^2/s in dilute systems.

Parameters are also assigned in the transport model based upon the type of constituents (chemicals) being simulated. These compound-specific parameters include the **half-life** for decaying species and chemical reaction types. Most transport models, including WinTran, lump chemical reactions into sorption processes in which a distribution coefficient (k_d) controls the relative velocity of the compound compared to the ground-water velocity. In this case, a **retardation coefficient** is computed which retards the velocity of the contaminant relative to the groundwater velocity.

Most contaminant transport models require you to enter a decay coefficient, which is different from the contaminant half-life. The decay coefficient is defined in the following equation:

$$\lambda = \ln 2 / t_{1/2}$$

where λ is the decay coefficient, ln 2 is the natural log of 2, and $t_{1/2}$ is the half-life of the contaminant. The half-life is the time required for half of the original mass of contaminant to decay. WinTran requires you to enter the half-life of the contaminant and computes the decay coefficient internally.

While the half-life is most often used for radioactive elements, such as uranium, it can also be used to express the decay of organic compounds through biodecay. The *Handbook of Environmental Degradation Rates* (Howard et al., 1991) is a good reference for contaminant half-life data.

WinTran requires you to enter the retardation coefficient directly rather than the distribution coefficient (k_d). Calculation of the retardation coefficient is given by the following equation:

$$R = 1 + k_d\,(\rho_b/n)$$

where R [dimensionless] is the retardation coefficient, k_d [L^3/M] the distribution coefficient, ρ_b [M/L^3] is the bulk density of the aquifer, and n [dimensionless] is the porosity. Other more complex reactions have been used in numerical models, however, these have not been commonly applied and are not supported in WinTran.

Optional Data

Optional Data types define the analytic elements and initial conditions of the model. These elements include the following information:

well locations and pumping rates,

recharge rate,

pond locations and recharge/discharge rates,

linesink locations and recharge/discharge rates,

calibration targets, and

initial contaminant conditions.

Wells are defined by the coordinates of the center of the well, a pumping rate, and well construction parameters. The pumping rate has units of [L^3/T], such as ft^3/d. A positive pumping rate indicates production and a negative rate indicates injection. Injection wells may also serve as sources of contamination. In addition to the injection rate, you may also enter a concentration of contaminant in the injected water.

Recharge is defined only for steady-state models and has units of [L/T], such as ft/d. The recharge is distributed over the entire infinite plane of the model. An ellipse defines the shape and position of the water-table mound created if there is no regional gradient. You must define the center coordinates of the ellipse, the length of the a- and b-axis of the ellipse, and the angle between the a-axis (long axis) of the ellipse and the x-axis of the model. The recharge rate should be a positive value.

Ponds are circular recharge or discharge areas. Ponds are defined in a manner similar to wells. The coordinates of the center of the pond are given, along with the radius of the pond. The pond infiltration rate has units of [L/T], such as ft/d, and is computed by dividing the total discharge rate into or out of the pond by the area of the pond. Thus, the pond infiltration/discharge rate has the same units as recharge. A positive rate indicates infiltration and negative indicates discharge from the aquifer. Infiltrating ponds may be a source of contamination by entering a concentration value. The concentration represents the contaminant concentration in the infiltrating water.

Linesinks are linear recharge or discharge features. The linesink is defined by providing the coordinates of each end point and either an infiltration/discharge rate or a head. If the linesink is defined in terms of head, the model will compute a discharge rate based on the given head value (assumed to be at the center of the linesink). Linesink discharge/infiltration rates have units of [L^2/T], such as ft^2/d, which is computed by dividing the total discharge rate by the length of the line. Thus, the linesink rate (or strength) is a rate per unit length. The sign convention for linesinks is the same as wells: positive indicates pumping or production and negative signifies injection or infiltration. Like injection wells and infiltrating ponds, linesinks may be sources of contamination when water is injected into the aquifer.

Calibration targets are values of hydraulic head that have been measured in the field. WinTran will compute the difference between the computed head at a target location and the measured head. This difference is called the residual and is displayed by selecting **Calc->Target Statistics** from the main menu.

Initial conditions are the contaminant concentrations in the aquifer at the start of the transport simulation. The default initial concentration is zero at all points. Initial conditions may be changed from the default of zero in two ways, (1) save the ending contaminant concentrations from a previous simulation as the initial conditions for a subsequent simulation, and (2) import contoured concentration data from a SURFER grid file. The tutorial in the next chapter discusses the saving of initial concentration

data from one run to another. The initial concentrations are graphically displayed on Step 1 of the tab view representing a time of 0.

After defining all aquifer properties, analytic elements, and transport parameters, you run the model by selecting **Calc** from the main menu and **Recalculate** from the pull-down menu. WinTran will now compute hydraulic head on a regular grid of points and contour the results using ten contour intervals. You may change the contour settings by selecting **Options->Contour->Parameters** menu.

To compute the transport model, you must select **Transport** from the **Model** menu and then select **Recalculate** from the **Calc** menu. If **Steady-State** is selected in the **Model** menu, only the flow model will be computed.

Starting a New Problem

The first step in using WinTran to solve a two-dimensional ground-water flow and transport problem is to digitize a site map. While this step is not required, it does help you by providing a convenient frame of reference. WinTran map files are similar in format to *QuickFlow* map or *ModelCad*™ map files and may be created by first using any CAD package (such as AutoCAD) to digitize the map. Next, export the CAD drawing as a DXF (Drawing Interchange Format) file.

Double-click on the WinFlow icon to start the model. You translate the DXF map file into a WinTran map file within WinTran by first opening a new model document. Select **File** from the main menu and **New** from the pulldown menu. You may also click on the New Document button on the Toolbar . Now that you have a model to work with, select **File** from the main menu, **Map** from the pulldown menu, and set *Files of Type* to "DXF Files (*.dxf)". Select your DXF file using the standard Windows file dialog. You must now select a new file name for the map file that will be created (again using a standard Windows file dialog).

Finally, you will be asked for one additional item, a multiplication factor. The multiplication factor is used to scale the drawing to model units. Normally, the multiplication factor will be 1.0; however, if your CAD system exports the DXF file in units of inches, this factor would be 1.0/12.0 or 0.08333333. You may need to use a multiplier other than 1.0 in AutoCAD if you are using engineering units instead of decimal units. If the map units look wrong in WinTran, it is probably in units of inches. You must then re-import the DXF file with a new multiplication factor.

It is easy to check the units on the imported map by moving the mouse around the map and looking at the X and Y coordinates displayed in the lower right corner of the status bar. Choose two points that are a known distance apart and check the distance with the status bar. If the distance is incorrect, compute a multiplication factor to use in the map import and start again.

The DXF file import feature first translates the DXF file into a WinTran map file and displays the map on your screen. The map file will be used the next time you open this WinTran model (assuming that you save it later).

When starting with a map that is already in the proper WinTran map file format, simply select **File** from the main menu, **Map** from the pulldown menu, and set *Files of Type* to "Map Files (*.map)". You may have to enter a rotation angle for the map after the map has been loaded. This option has been included because it is recommended that the simulated contaminant plume flow parallel to either the X- or Y-direction. This minimizes what is sometimes called "grid effect", which causes numerical problems in the transport solution. To set the rotation angle, click the **Options->Map->Parameters** menu, click the ***File*** tab, change the value for *Angle* and click **OK**.

Now that the map has been loaded, you must define the aquifer parameters. Default parameters have been defined by WinTran, however, these may not be the proper ones for your problem. To change these parameters, select **Edit** from the main menu and **Parameters** from the pulldown menu. The following parameters are entered on this dialog:

hydraulic conductivity,

aquifer bottom elevation,

aquifer top elevation,

reference head value,

regional hydraulic gradient,

recharge rate, and

porosity.

If you select **Reference Head** from the edit menu, a dialog appears that prompts for the reference head value, gradient, and angle of the gradient. A gradient angle of 0.0 degrees represents east, 90 degrees is north, etc. You may change the location of the reference head by dragging the reference head icon (➔) to a new location. You do this by moving the cursor over the reference head icon (➔) until the cursor appears. Now, hold down the left mouse button and move the cursor to the desired location. Release the mouse button to place the reference head at the new location.

You must <u>always define a reference head</u>! You do not necessarily need to define a gradient, however. If no gradient is defined, WinTran will be simulating an "impact" model using a flat water-table. If the reference head is zero and the gradient is zero, WinTran will compute drawdown In this case, be sure that the bottom of the aquifer is set below zero also.

After confirming that all aquifer parameters have been properly set, you must then define the transport parameters, which include the following:

longitudinal and transverse dispersivity,

coefficient of molecular diffusion,

retardation coefficient, and

contaminant half-life.

Finally, you must select parameters that pertain to the time-stepping of the transient transport model. These parameters include the number of time steps, the length of the first time step, the maximum time step size, and the time step multiplier. WinTran starts the simulation using the initial time step size. Subsequent time steps are multiplied by the time-step multiplier to obtain the new time step size. This multiplication continues until the maximum time step size is reached or the end of the simulation occurs. Normally, you need to start with small time steps and gradually move to larger ones in order to have a good mass balance. The mass balance error is displayed in the status dialog as the model runs. If the mass balance error rises above 10 percent, you should stop the simulation and adjust the time stepping parameters, dispersivity, or porosity until a better mass balance is achieved. The last chapter in the documentation provides some guidance on selection of these parameters.

If you want to display contours representing the uniform regional gradient, you may now select **Calc** from the main menu and **Recalculate** from the pull-down menu. WinTran will display the status of the computations in a status dialog. After calculating the head at a regular grid of points, a head contour map will be displayed using a default contour interval. If you want to change the appearance of the contours, select **Options**

from the main menu and **Contour** from the pull-down menu. A third menu will appear with two options for controlling the contouring, **Window** and **Parameters**. Use **Window** to change the size or location of the area that is contoured. **Parameters** is used to change the size of the contour matrix and contour parameters related to the head and concentration contours.

Adding Model Features

Wells, linesinks, ponds, streamlines, and/or particle-traces may be added to the model. To add one of these features to the analytical model, select **Add** from the main menu and select the desired feature from the pull-down menu. You must then move the cursor around the design screen to locate the item. This is accomplished using the cursor keys or the mouse. To alter the distance that the cursor moves after pressing an arrow key, select **Options** from the main menu and **Map** from the pull-down menu. The cursor step size is defined in model units of length, e.g., feet.

Wells are defined by moving the cursor to the desired coordinates or location and pressing the **Enter** key or left mouse button. You will then be prompted for the well construction parameters and pumping rate [L^3/T]. You must enter a unique well designator. You will also be permitted to edit the well coordinates. If you change the coordinates, the well will be redrawn at the correct location. Remember that a positive pumping rate indicates production, while a negative rate indicates injection. Injection wells may be contaminant sources by providing a concentration value in the ***Transport*** tab of the property sheet. The concentration represents the contaminant concentration in the injected water. Note that this is not necessarily the concentration that will be computed in the model because dilution in the flowing groundwater will normally reduce the injection concentration to a much lower value.

There are two other options available for wells: (1) observation wells, and (2) constant concentration wells. In observation wells, concentration is recorded by WinTran at each time step. These data, presented in the **Transport** tab, can also be graphically presented in a view tab (click the **View->Well Data** menu) or may then be exported to a text file that can be imported into another graphics program, such as Microsoft Excel, to plot concentration versus time. In a constant concentration well, the contaminant concentration is held constant in the aquifer at the well location. When using constant concentration wells, you should specify a pumping rate of zero.

Another option that effects wells in WinTran is the **Options->Snap Wells to Contour Grid** menu. Placing a check mark next to this options causes WinTran to move newly added wells to a contour matrix node. Remember that the finite-element mesh used for transport is identical to the contour matrix used to contour heads. If you enter a well at a location other than a node (intersection of contour grid lines), the velocities computed by WinTran around the well will not be accurate. The inaccurate velocities may produce undesirable mass balance errors. This option only effects wells added after the option is selected. It is recommended that you use this option to ensure proper mass balance in the transport model.

There is no set limit to the number of wells or any other element that may be added to the WinTran model. The actual limit depends upon the amount of memory in your computer.

Linesinks are defined by pressing the left mouse button at the beginning of the linesink. While holding down the left mouse button, move the cursor to the end of the linesink and release the button. You will then be prompted for the head or flux of the linesink. If you have selected a Head Linesink, WinTran computes the flux required to maintain that head at the center of the linesink. If you have selected Flux Linesink, the flux is entered in units of [L^2/T]. This is normally computed by dividing the total flux rate [L^3/T] by the length of the linesink [L]. To aid you in determining the proper flux rate,

WinTran displays the length of the linesink in the dialog. A negative flux indicates injection and positive indicates production. As with the wells, you also have the opportunity to edit the coordinates and name of the linesink. You may also specify a concentration value for the linesink. The concentration is only used if the linesink injects water into the aquifer. As with wells, the concentration represents contaminant concentration in the injected water.

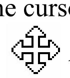
Ponds are defined by selecting the center of the pond and dragging a circle to the desired radius while holding down the left mouse button. A dialog will appear that allows you to edit the center coordinates, the radius, infiltration/discharge rate, and name. The pond rate is in units of [L/T] in the same manner as recharge. A positive pond flux indicates recharge, while negative signifies discharge. Note that the sign convention for ponds is opposite of wells and linesinks. As in wells and linesinks, you may specify a concentration to apply to any injected water.

Streamlines are only computed for the steady-state model. You have the option of adding a single streamline, several streamlines started along a line, or several streamlines along a circle. If you choose the line or circle option, you will then enter the number of streamlines to add. These will be added at even increments along the line or circle. When you add a streamline, the point you select is the beginning of the streamline, which is drawn downgradient of that point. When you change the model in any way, all streamlines are recomputed during the recalculation of the model. Arrow heads may be added to the streamline by selecting **Options** from the main menu and **Trace** from the pulldown menu and clicking the **Display Arrows** checkbox. You define the distance between arrowheads and the size of the arrow (in model units of length) on the ***Streamline*** tab.

Particle-traces are defined in the same manner as streamlines. Particle-traces take longer to calculate, however. The particle-trace differs from the streamline in several ways: (1) the particle-trace is computed numerically instead of semi-analytically; (2) a time value may be posted on the resulting trace; and (3) particle-traces may be tracked upgradient (reverse particle-tracking). Options controlling the appearance and computation of particle-traces are modified by selecting **Options** from the main menu and **Particle** from the pull-down menu. If you want to display arrow heads and/or post travel times, click the appropriate checkbox. You define the distance between arrow heads and the size of the arrow heads (in model units of length) on the ***Particle Trace*** tab and travel time label parameters on the ***Label*** tab

All of these analytical features may be added from the Toolbar, as described in the previous section. Simply click on the proper Toolbar icon. As the cursor is moved over the Toolbar, a text is displayed in the status bar to help you remember which icons go with which elements.

Editing Analytic Elements

You may edit any analytic element on the map (well, linesink, pond, title, or scale bar) by moving the cursor over the item and double clicking. Make sure that the cursor changes to  before double clicking. A dialog will appear that allows you to edit the parameters associated with that element (e.g. pumping rate). To move the element, hold down the left mouse button and drag the item to a new location.

The size of linesinks and ponds may be changed by moving the cursor to the edge of the pond or end of the linesink. When in the proper location, the cursor will change to a  . Hold down the left mouse button and drag to a new size.

Computing the Model

If you have added a feature, edited a feature, or changed any aquifer property, WinTran will display "ReCalculate" on the status bar. This means that the contour map on the screen may no longer reflect the changes that you have made. To recompute, select **Calc** from the main menu and **Recalculate** from the pull-down menu. If you have only changed contouring options, you may select **Recontour** from the pull-down menu. You may also use Toolbar icons to recalculate and recontour the model.

When WinTran recalculates the problem, it displays the percentage completion in a status dialog. The time required to recalculate depends on the speed of your computer and the number of features that you have added to the model. Recalculation normally only takes a matter of seconds or a few minutes. After recomputing the model, WinTran will recalculate the locations of any streamlines and/or particle-traces that you have added to the model. Finally, WinTran will recontour the results. Progress bars are displayed during each operation so that you can judge how long it will take to finish.

WinTran provides you with the option to compute just the flow model or both flow and transport. The **Model** menu contains three options: (1) **Steady-State**, (2) **Transient**, and (3) **Transport**. A check mark will appear next to either **Steady-State** or **Transport** and **Transient** will be inactive.

WinTran provides two options for computing groundwater velocities in the transport model. The default method is to compute velocities analytically using the analytic element flow model. This is the "hybrid model approach" described in the chapter entitled **Mathematical Models**. It is possible that this modeling approach may produce poor mass balance errors, especially when you are using linesinks and ponds. If you get large mass balance errors (above 1 to 10 percent), try selecting the **Edit**->**Model** menu, click the ***Solver*** tab and set *Flow Model Type* to "Finite-Element**"**. The finite-element flow model option adds an intermediate step in the simulation process. WinTran first computes the analytic flow model. Next, WinTran sets up a finite-element flow model using boundary conditions from the analytic model. After running the finite-element flow model, WinTran computes velocities for transport.

Evaluating Model Error

Numerical transport models require the user to carefully evaluate each simulation for potential errors. WinTran assists you in evaluating model error by displaying the mass balance error on the status bar when the transport model is running. The mass balance error is expressed as a percentage and should be less than 10 percent for a valid simulation. Usually, the mass balance error is less than 1 percent.

Even if the mass balance error is below 10 percent, there can be oscillations in the transport solution. Oscillations are indicated by negative concentrations computed by WinTran. In extreme cases, alternating nodes will have positive and negative concentrations producing diamond-shaped contours. When the transport solution oscillates, check the following:

(1) The Peclet number is displayed on the status dialog as "Pe =" and is computed by dividing the nodal spacing (the distance between nodes in the contour matrix) by the longitudinal dispersivity. The Peclet number should generally be less than 2 for a stable solution. If you are experiencing mass balance problems or oscillations, increase dispersivity until the Peclet number is less than 2.

(2) The Courant number is another criterion used to judge the stability of a transport simulation. The Courant number is computed as the velocity times time-step size divided by nodal spacing. This criterion is displayed as "Cr=" on the status bar and should generally be less than 1. Again, if you are experiencing mass balance or oscillation problems, try decreasing the initial and maximum time-step sizes.

There are also times when the Courant number is too low. In cases where the Courant number is less than 0.1, there can be round-off errors in the matrix solver. In this case, you should increase the initial and maximum time-step sizes until the Courant number is close to 1.

There are two other WinTran options that can aid in model stability. These include the time discretization method (backward and centered in time) and upstream weighting. The time discretization methods are selected using the **Edit->Model** menu and clicking the ***Solver*** tab. Backward in time is unconditionally stable but is only first-order accurate, while centered in time is second-order accurate but may be subject to instability (Javandel et al., 1984). It is usually best to start with backward in time.

Upstream weighting factors in the X- and Y-directions are edited from the **Edit->Menu Parameters** menu and ***Solver*** tab. Upstream weighting factors of 1.0 indicate full upstream weighting, while a weighting factor of 0.0 turns off upstream weighting. Upstream weighting adds stability to the solution (helps eliminate oscillations) at the expense of added numerical dispersion. Numerical dispersion is artificial dispersion that produces similar results to an increase in the dispersivity coefficient.

For more information on model stability, see the Problems with Model Stability section of the WinTran Application Guide chapter.

Displaying the Results

WinTran displays the results from model calculations using contours, streamlines, and particle-traces. By default, the design window defines the area that will contain the contours. You may change the default by selecting **Options** from the main menu and **Contour** from the pull-down menu. A third menu will display several contouring options. The **Window** selection allows you to drag a rectangular box to define the new contour region. The **Parameters** selection on this third menu provide ways to change the labeling of the head and concentration contours and the contour interval. If you make any changes to these options, a dialog will prompt you to recontour the results. If you have redefined the contour window, you must select **Recalculate** from the **Calc** menu.

You may change the current view of the model by selecting **View** from the main menu. A pull-down menu allows you to zoom on a point (**Zoom**), zoom on a window (**Window**), display the full map and contours (**Full**), scroll the screen a given distance in a certain direction (**Scroll** and Left, Right, Up, Down), or to simply redraw or refresh the current screen (**Refresh**).

Creating Output

WinTran creates graphical and summary output in several different ways, including the following:

Print to any selected Windows device;

Create files that are compatible with SURFER™, Geosoft, or Spyglass;

Create standard Windows Metafiles (wmf);

Create ASCII flat files containing X- and Y-coordinates and head (XYZ);

Create DXF file for importing into CAD systems; and

Create an ASCII summary output file.

Any of these options can be selected from the **File** option on the main menu. Select **Print** to send the plot to the current Windows printer. Select **Export** to create the other files listed above.

*Contour labels are often not properly rotated when printing! Make sure you select **Printer Setup** and select the option to **Print Truetype Fonts as Graphics**.*

A common problem when printing to the current Windows printer is that the text for contour labels is not properly rotated. To correct this problem, select **File** from the main menu and **Printer Setup** from the pulldown menu. Most printer drivers have an option to "Print Truetype Fonts as Graphics" or something similar. Make sure that this option is switched ON. If this does not correct the problem, you may need an update for your printer driver. Contact your printer manufacturer or Microsoft for a new one. Many driver updates are also available on Compuserve.

Another problem can occur in getting the plot to fit on a single page. Select **Page Setup** from the **File** menu, click the ***Scale*** tab and click on the **Scale to Page** radio button in the *Map Scale* section. This will force the next printout to fit on a page. Additionally, placing a check in **Display Window** causes only what is currently displayed on the screen to be printed. Placing a check in **Map Window** causes the currently defined map window to be printed. The current Map Window is the area defined when you select **View->Full->Screen**. You may change the size of the map window by using **Options->Map->Window** or directly changing it by clicking the **Options->Map->Parameters** menu on the ***Window*** tab.

DXF files created by WinTran are compatible with AutoCAD and most other CAD software, such as Design-CAD. All entities, such as wells, contours, base map, streamlines, etc. are stored as separate layers in the DXF file. This allows for easy editing of the CAD drawing.

WinTran creates up to three different SURFER files. The regular grid of head values computed by the analytical model is stored in a SURFER grid file that can be displayed by the TOPO and SURF programs. The grid file can also be used in the GRID program to make modifications. If you are using a base map or if you have added streamlines or particle-traces, a SURFER XYLINE file may be created. This file can be read by the TOPO program to display a digitized map on the contours. If you have generated time postings on the particle traces, a posting file may be created for use in TOPO.

In order to maintain a QA/QC record of your model run, a summary file may be created. The summary file contains a listing of all data making up the analytical model, the names of all files used, the current date, and a summary of all computations.

Saving a WinTran Model

Eventually, you will want to save your WinTran model in a data file for later use. If you have previously saved the file, select the **File->Save** menu; if not, select the **File->Save As** menu and give the file a name. The file created is totally self-contained and, unlike previous versions of WinTran, contains the base map information as well.

Tutorial

Starting with a Simple Test

What you will learn:

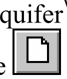
- *entering data into the program*
- *analyzing a test with one observation well using the Theis method*
- *manual type curve matching*
- *using weighted least-squares matching*
- *specifying units*
- *modifying the graph & displaying bitmaps*
- *printing your results*
- *pasting your graph into Microsoft Word*

Aquifer^{Win32} is designed to be both easy to use and very flexible in analyzing aquifer test data and displaying the results. The first part of the tutorial shows you how to analyze a simple aquifer test, one in which there is only one observation well. This first analysis will use the Theis solution (1935) for confined aquifers. The data are from a real aquifer test reported by Kruseman and de Ridder (1990; page 59).

Kruseman and de Ridder call this aquifer test Oude Korendijk for the area where the test was conducted. The pumping well was screened over the entire aquifer thickness of 7 meters. Piezometers were placed at distances of 0.8, 30, 90, and 215 meters. The well was pumped at a constant discharge rate of 0.547 m^3/min for 14 hours. This tutorial will use time and drawdown data for the 30 meter piezometer.

Entering Data

The easiest way to get time and drawdown data into Aquifer^{Win32} is to copy the data to the clipboard from a spreadsheet and then paste the data into the Aquifer^{Win32} spreadsheet. Any Windows spreadsheet, like Excel, can be used to manipulate the data before using it in the aquifer test analysis.

You start the Aquifer^{Win32} program using the Start menu. To start a new aquifer test, simply click the  button on the Standard toolbar or select **File->New**. Selecting **New** from the **File** menu displays five types of documents or analysis types,

including (1) “AquiferWin32 Flow Model”, (2) “AquiferWin32 Analysis”, (3) “AquiferWin32 Simulation”, (4) “AquiferWin32 Slug Test”, and (5) “AquiferWin32 Step Test”. A **Flow Model** document is used for analytical groundwater flow modeling or the analysis of very complex pump tests. An **Analysis** document is for normal analysis of aquifer test data and is the document created when clicking the button on the Standard toolbar. A **Simulation** document is for simple modeling of an aquifer test and presenting contours of drawdown. A **Slug Test** document is a special type of analysis document for evaluating the results of slug tests using several different methods. A **Step Test** document is for analyzing variable discharge aquifer test data from the pumping well.

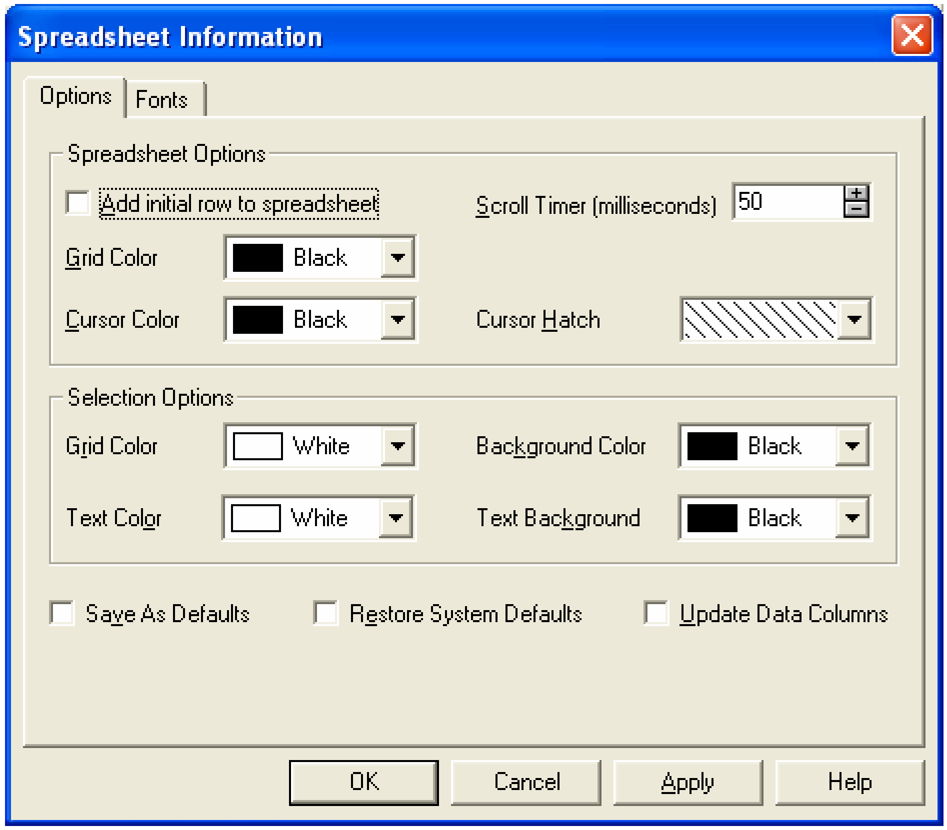
In this first tutorial exercise, simply click the button of the Standard toolbar to start. Two windows are created within the Aquifer^{Win32} frame, including a spreadsheet on the left and a graph on the right. The default graph type is for the Theis confined analysis and we will not change that for this example. The spreadsheet contains one default data from which to begin data entry. If the default data line is absent, it has been disabled using the *Spreadsheet Information* property sheet accessed using the **Edit->Options** menu. If the default data line is absent, depress the **Ins** key.

The first thing you should do when starting a new analysis is to set the units for the pumping test data. Select **Edit->Units** to tell Aquifer^{Win32} what units you are using. In this case, select minutes (min) for *Time Units*, meters (m) for *Length Units*, meters cubed per minute (cu m/min) for *Pumping Rate Units,* and square meters per minute (sq m/min) for *Transmissivity Units*. Click the **Apply Globally** check-box at the bottom of the property sheet to set these units for the entire analysis including all data types**. Take note of the Convert Data check-box at the bottom of the property sheet**. The convert option would transform all data from the units initially shown on the property sheet to those you want to use. This can be convenient when you want to convert units later on; however, we do not want Aquifer^{Win32} to perform any calculations at this point. When both the **Apply Globally** and **Convert Data** check-boxes are checked, the entire document will be updated to a new set of units without affecting the analysis results. This feature is of primary use when peer reviewing analyses that were done in a different set of units than the one you are comfortable with. You can simply change the units and review the analysis. In this example, we just want to define what our units are. Accept the property sheet by clicking the **OK** button.

Aquifer^{Win32} has a very sophisticated unit conversion scheme that allows you to convert just about any data item in the program from one set of units to another at any time. We will explore unit conversion in more detail later. Right now, we just want to set up the units that our data were entered in. The time and drawdown data points for this test are provided in a file called kdr_ok.xls which is in Microsoft Excel Version 7.0 format. You can enter the data for this test in three ways, (1) open this file in your spreadsheet, copy the data to the clipboard, and paste into the spreadsheet in Aquifer^{Win32} , (2) manually type the data into the Aquifer^{Win32} spreadsheet, or (3) input the data from an ASCII text file (*kdr_ok.dat*).

To paste the data into the Aquifer^{Win32}, simply go into your spreadsheet (Excel or Lotus, etc.), open the kdr_ok.xls file, drag a selection block around the first two columns of data from spreadsheet row 6 through row 39, and copy the data to the clipboard. Next, click on the left side of line 1 in the Aquifer^{Win32} spreadsheet. Type Ctrl-V or click the button on the Standard toolbar to paste the data into Aquifer^{Win32}. The default first point is still in the spreadsheet, however, so go to line 35 in the spreadsheet, click on the line to highlight the data, and hit the **Del** key.

Many users have indicated that they do not want the default first line in the spreadsheet when a new document is created. It's primary purpose is to facilitate manual data entry; however, most often the data is imported or pasted into the spreadsheet. To eliminate the default data row from appearing in future documents, click on the Edit->Options menu and click on the **Add initial row to the spreadsheet** check-box to remove the check as below.



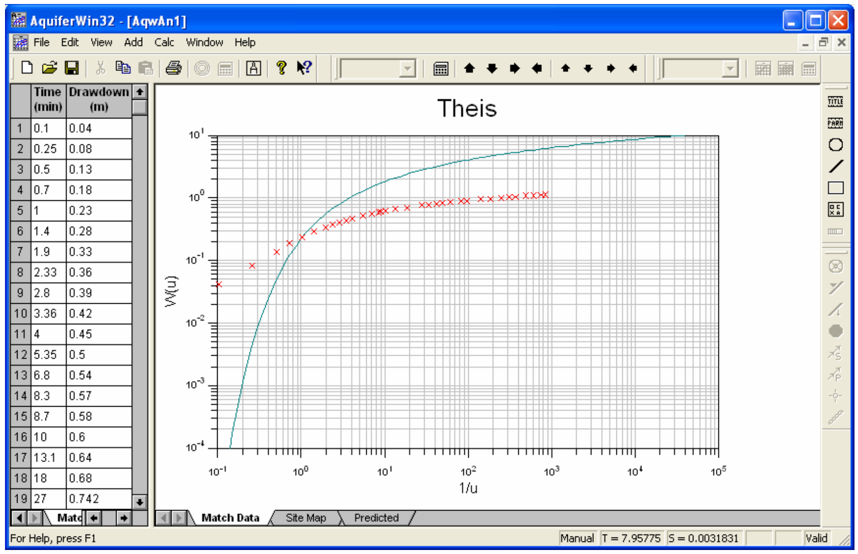
To import the data from a text file, click on the first line in the spreadsheet to highlight that row. Select **File->Import** and find the file called *kdr_ok.dat*. Aquifer^{Win32} will notify you that 34 lines were imported. As with pasting from the clipboard, there will be one extra line to be deleted. In this case, however, you will delete the first line.

You may also enter the data by hand. Simply click on the first cell in the spreadsheet and start typing the data. The first column is for time and the second column is drawdown. These data are shown below for the 30 meter piezometer.

Time (min)	**Drawdown (m)**
0.100	0.040
0.250	0.080
0.500	0.130
0.700	0.180
1.000	0.230
1.400	0.280
1.900	0.330
2.330	0.360

2.800	0.390
3.360	0.420
4.000	0.450
5.350	0.500
6.800	0.540
8.300	0.570
8.700	0.580
10.000	0.600
13.100	0.640
18.000	0.680
27.000	0.742
33.000	0.753
41.000	0.779
48.000	0.793
59.000	0.819
80.000	0.855
95.000	0.873
139.000	0.915
181.000	0.935
245.000	0.966
300.000	0.990
360.000	1.007
480.000	1.050
600.000	1.053
728.000	1.072
830.000	1.088

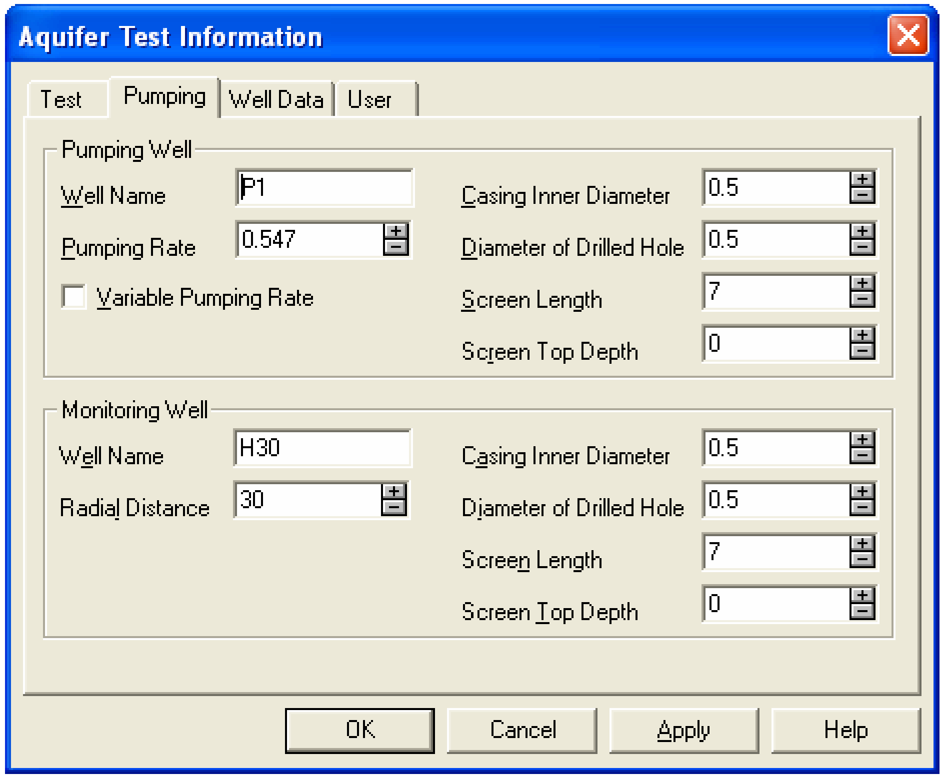
You will see the data being displayed in the graph window as they are entered. After entering the data shown above, your screen should look similar to the one shown below (if the data point locations are different from those shown below, select **Calc->Reset Data Offset**):



Now that we have the time and drawdown data, we will enter the remaining test data. Select **Edit->Aquifer Test** to enter the pumping rate and distance to observation well. Click on the ***Pumping*** tab on this property sheet and enter the following information:

Pumping Well Name	P1
Pumping rate	0.547 m^3/min.
Pumping well screen length	7 m
Monitoring Well Name	H30
Radial Distance	30 m
Screen length	7 m

The property sheet should resemble the one below. Click **OK** when you are done.



At this point, the data for our analysis have been entered. This involved essentially two steps, (1) enter the time and drawdown into the spreadsheet and (2) define the pumping rate and radial distance to the observation well. Now, click anywhere on the graph window. You will notice that the toolbar buttons that were gray are now ready for use. The toolbar and menu respond to the currently active window. The spreadsheet and graph are two different windows and they have two different menus.

Analyzing the Test

Aquifer^{Win32} provides two ways to estimate aquifer properties from time-drawdown data: (1) manual curve matching, and (2) a nonlinear least-squares statistical match. We will show you the statistical matching first.

Click on the graph to activate most of the toolbar. Click the  button on the Match toolbar to automatically choose the best value of transmissivity and storage coefficient for this test. Aquifer^{Win32} uses the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to find the best statistical match between the field data and the type curve you have chosen, in this case the Theis curve. You will see T (transmissivity) and S (storage coefficient) displayed on the status bar at the bottom of the Aquifer^{Win32} frame window. The values should be 0.334 m^2/min for T and 0.000112 for S after the optimization is complete.

You may view detailed results of the nonlinear least-squares match by selecting the **Edit->Solution** menu. The ***Solution*** tab on the *Solution Information* property sheet displays the type of analysis you have performed (in this case, the Theis analysis). Other tabs include:

- Parameters (the initial guesses for parameter values)
- Results (optimized parameters computed by either manual or nonlinear least-squares)

- Exceptions (unlink or unfix parameters; set enforced minimum and maximum values for parameters)
- Statistics (match point data and statistical measures)
- Data (time and drawdown data used to construct the type curve)

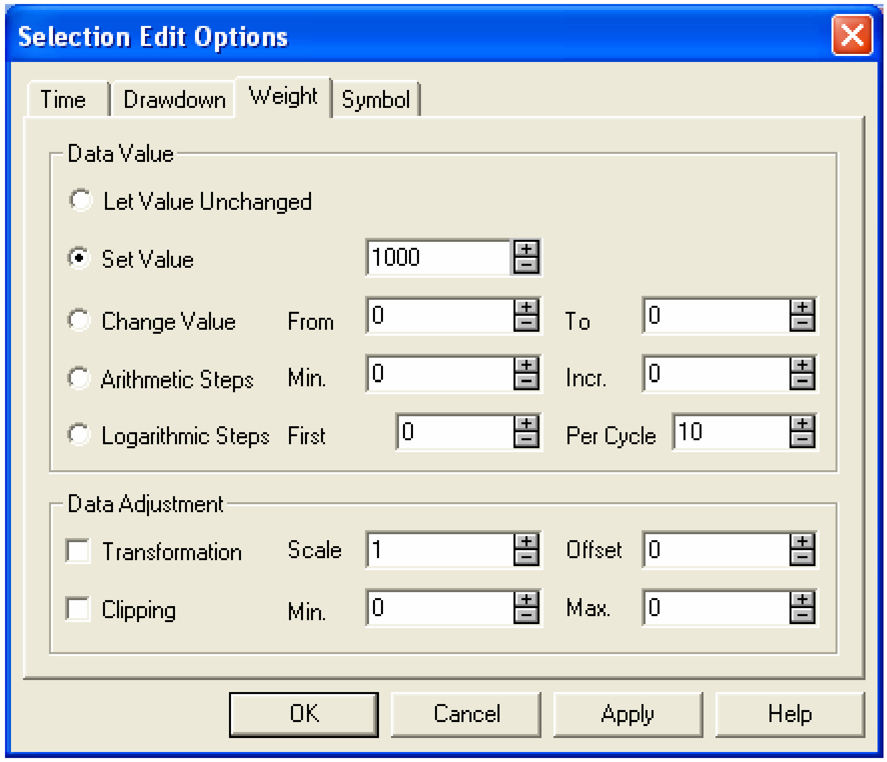
We will discuss this property sheet in more detail later in this tutorial. For now, just click the tabs to see what types of information are displayed. One unique feature of Aquifer^{Win32} is the ability to display the units for any field by moving the mouse cursor over that field. The units are displayed as tooltips (a box drops down from the field) and on the status bar. You may also right click on the field to display a context menu containing more options. One of the options on the context menu is a unit conversion feature.

The values reported by Kruseman and de Ridder are 0.272 m^2/min for T and 0.00016 for S. They obtained different results because they ignored late time data. You can come close to these results by pressing the down arrow key on the keyboard once followed by the left arrow key three times (**NOTE: the number of times you press the arrow keys is system-dependent and may vary**). This moves the data over the curve and yields T and S values that are closer to the Kruseman and de Ridder results.

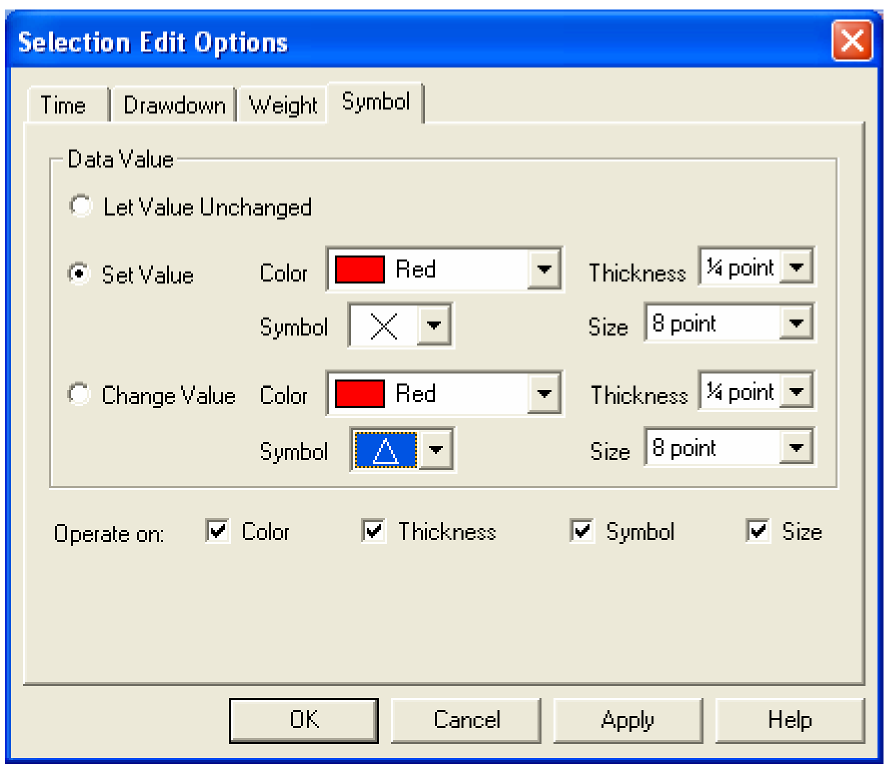
In addition to using the arrow keys on the keyboard, Aquifer^{Win32} provides two sets of four arrows (up, down, left, right) on the toolbar. There are four large arrows and four small arrows. The large arrows move the data farther than the small arrows. The keyboard arrow keys are equivalent to the large arrows on the toolbar. Holding down the shift key and pressing the keyboard arrow keys is equivalent to clicking the small arrows on the toolbar which move the data a small distance for fine tuning the match.

Weighted Data

Notice that the last 12 points on the curve do not match very well with the type curve. You can remove them from the analysis by changing their weights in Aquifer^{Win32}. Aquifer^{Win32} performs the nonlinear least-squares analysis on weighted residuals (errors). A higher weight means that the error has less significance to the results. By default all data points have a weight of 1.0. We will now change the weights of the last 12 data points to a value of 1000.0. Select **Edit->Aquifer Test** as before and click on the ***Well Data*** tab. You will see a spreadsheet showing time, drawdown, symbol type, and weight. Scroll to the last part of the spreadsheet and enter the value 1000.0 for the last 12 points. Alternatively, select the last 12 rows by clicking the mouse in the left most column of the spreadsheet containing row numbers on row 23 and dragging a selection around the remaining rows. Right click the mouse on the spreadsheet to activate the context menu and select the **Selection** menu. The *Selection Edit Options* property sheet is displayed. Tabs exist for each column of the spreadsheet. Since we are interested in the value of weight, click on the ***Weight*** tab, click on the **Set Value** radio button and enter “1000” into the adjacent edit field as below.



You might also change the symbol type to show that these points are being treated differently. To change the symbol type directly on the spreadsheet, double click on the spreadsheet cell containing the symbol you wish to change. You may change the color, size, and shape of the symbol. In this case, click on the ***Symbol*** tab, click the **Set Value** radio button, and set the *Symbol* combo box to a triangle as below. Accept the property sheets by clicking the OK buttons.



After modifying the weights click the button on the Match toolbar again and you should end up with a transmissivity value of 0.28 m^2/min and a storage value of 0.000167. These are close to the Kruseman and de Ridder results.

Changing Analysis Types

The example presented so far used the Theis method for confined aquifers. You may change the analysis type at any time by selecting **Edit->Solution**. We will cover the more advanced tests in later sections.

Modifying the Graph

You may modify just about any feature of the graph presented in Aquifer^{Win32}. Changing axes and labels is accomplished by selecting the **Edit->Graph** menu (**NOTE: you must first click on the graph side of the window to enable editing of the graph**). The *Graph Information* property sheet is displayed with the following tabs:

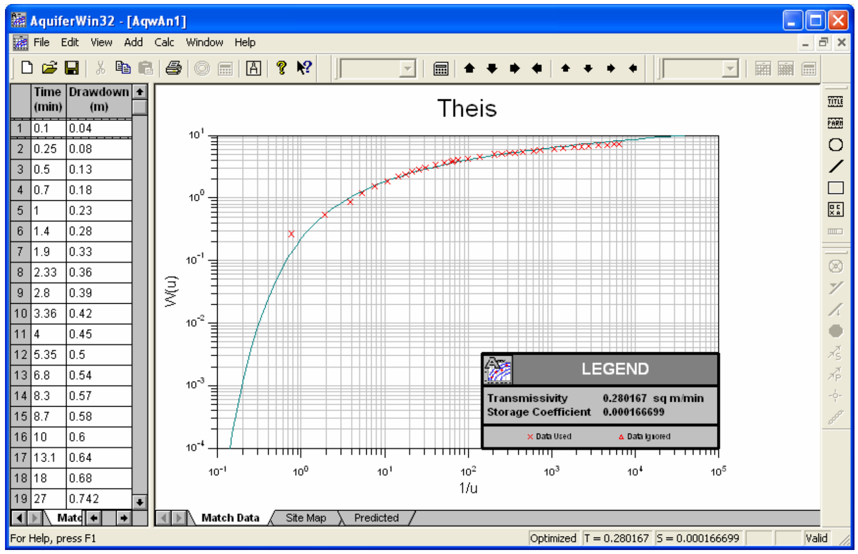
Graph	Title of the graph and size of the graph in inches
X-axis	Style and annotation of the X-axis
Y-axis	Style and annotation of the Y-axis
Line Types	Thickness and color of the graph border, axes, grid lines
Line Styles	Color, thickness, labeling, style of data and type curve line(s)

You should explore these options to see how you can modify the style of the graph. The default settings are usually adequate, however, for most applications.

Another useful feature to annotate the plot is the use of legends, titles, and frames to enhance the graph. These are selected on the **Add** menu when the graph window is highlighted. One common use of these features is to add a logo for your firm. To do this, select **Add->Frame**. Now drag a rectangle on the graph. The *Frame Information* property sheet will be displayed for this frame. Click on the ***Contents*** tab and change the *Type* field to "Bitmap". Click the **Browse** button to find the bitmap file you want to display (an example ESILOGO.BMP is provided for the tutorial). Most of the time, you will check the check-box **Scale to Rectangle** which forces the bitmap to fit within the rectangular frame. You may move the frame around on the screen and resize it. You may add as many frames as you like to the graph.

You will often want to add legends to the graph to display the value of T and S for example. The easiest way is to add a default legend by clicking the **Add->Legend** menu, drag a rectangle on the screen and click the **Finish** button on the Legend Wizard. Custom legends can be somewhat complex and are covered in another chapter. Another aproach to legends is to create one that you like and then copy/paste it from one Aquifer^{Win32} document to another. That way, you only have to create it once.

Below is an example of what a final analysis would look like by modifying the title, adding the legend, and performing the analysis with the weighted time-drawdown data as discussed above. This legend also includes information identifying which data points were used and which were ignored.



Printing Results

Aquifer^{Win32} prints the graph on your screen using any available Windows™ driver. Simply select **File->Page Setup** to decide margins etc. Now select **File->Print** or **File->Print Preview** to print the graph.

Displaying a Graph in Another Application

Aquifer^{Win32} is an OLE server application. This means that you can copy a graph from Aquifer^{Win32} into another Windows application (Word or Excel, for example). You have the choice of linking the graph to the Aquifer^{Win32} file so that it can be modified if you change your analysis in the future.

To copy a graph to Word, for example, you would run Aquifer^{Win32} and perform the analysis. Click on the graph window and select **Edit->Copy**. Now, run Word (or any other Windows application) and select **Edit->Paste Special**. A property sheet is displayed that allows you to paste the graph as a "Picture" or as an "AquiferWin32 Analysis Document Object". If you choose the latter, you may then double click on the graph in Word to modify it. You may also choose an option called *Paste Link* which would link the graph to your Aquifer^{Win32} file so that the Word document would be modified when you change the analysis. This is a powerful feature that allows you to create customized reports for your aquifer test analysis.

Multiple Type Curve Example

What you will learn:

- *Analyzing an aquifer test in a leaky aquifer with one well*
- *Displaying and manipulating multiple type curves*

Many of the aquifer test analysis methods use a family of type curves. In this case, your goal is to not only fit the data to a curve but to also choose the best curve. Aquifer^{Win32} is designed to help in this type of analysis by displaying a family of type curves for these methods. You may choose how many curves are displayed and may use this feature to prepare sets of master type curves for use in analyzing data in the field (i.e., the old-fashioned way!).

In this second example, we will analyze an aquifer test reported by Lohman (1979; page 33) using the Hantush (1960) method for leaky aquifers with storage in the aquitard. This method uses a family of curves with each curve having a constant value for β, which accounts for the thickness, hydraulic conductivity, and storage properties of the aquitard.

Follow these steps to prepare for the analysis of the Lohman data:

1. Click the  button on the Standard toolbar and enter the time and drawdown data for the test. These data are given in an ASCII text file *loh33.dat* and in the following table. Remember, to import the ASCII file click on the first row in the spreadsheet and select **File->Import**. You can also right click on the spreadsheet and select **Import** from the context menu.

Time (min)	Drawdown(ft)
6.37	0.01
8.58	0.02
10.23	0.03
11.9	0.04
12.95	0.05
14.42	0.06
15.1	0.07
16.88	0.08
17.92	0.1
21.35	0.12
21.7	0.13
22.7	0.14
23.58	0.15
24.65	0.17
29	0.21
30	0.22
32	0.24
34	0.26
36	0.28
38	0.3
41	0.33
44	0.36
47	0.38
50	0.42
54	0.46
60	0.52
65	0.56
70	0.6
80	0.65
90	0.75

100	0.82
137	1.04
150	1.12
160	1.17
173	1.24
184	1.27
200	1.35
210	1.4
278	1.68
300	1.76
315	1.83
335	1.87
365	1.99
390	2.1
410	2.13
430	2.2
450	2.23
470	2.29
490	2.32
510	2.39
560	2.48
740	2.92
810	3.05
890	3.19
1255	3.66
1400	3.81
1440	3.86
1485	3.9

2. Select **Edit->Units** and set *Time Units* to "min", *Length Units* to "ft", *Pumping Rate Units* to "gal/min", and *Transmissivity Units* to "sq ft/d". Click the **Apply Globally** check-box then click **OK** to accept the property sheet.

3. Select **Edit->Aquifer Test** and click on the ***Pumping*** tab. Enter the pumping rate of 750 gallons per minute (gpm) and the distance of the observation well from the pumped well of 1,400 ft. Click **OK** to accept the property sheet.

4. Click on the graph window and then select **Edit->Solution**. The ***Solution*** tab is shown first by default. Change the combobox from "Theis, 1935 (Confined)" to "Hantush, 1960 (Leaky Aquifer with Storage)". Click the ***Curves*** tab to see which values of β will be plotted as separate type curves. You may add additional curves by clicking the **Add** button.

5. Click the ***Parameters*** tab and ensure the all three *Hydraulic Parameters* are free to vary. The small check-boxes that are intended to resemble push pins located to the right of *Transmisivity*, *Storage coefficient* and *Beta* should all be unchecked. If they were checked, as with *Radial distance* and *Pumping rate*, the values would be held constant during optimizations.

6. Click **OK** to accept the property sheet.

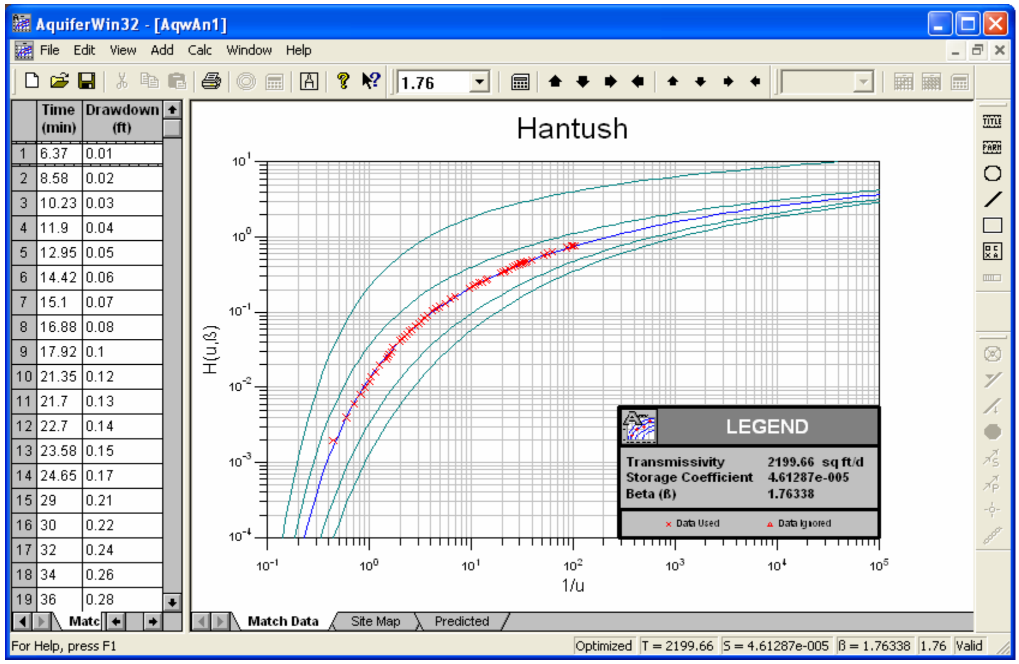
You are now ready to analyze the data. It is strongly recommended that a manual match be performed first. Using the arrow keys on the keyboard, attempt to manually match the data to the third type curve from the left. Essentially, you try to determine which type curve best matches your data. Then, set the combo box on the Match toolbar to that value, in this case use “2.00”. You could also toggle through the type curves by using **Ctrl-T** on the keyboard. Once you have changed the selected type curve, make final adjustments to the manual match. A manual match will update the initial guesses for parameter values based on the match point and the selected type curve so you must move the data after a new type curve has been selected to update its initial guess.

The default behavior of the application is to calculate an initial guess for T and S using a straight line method. Although this works well for some analyses, it doesn’t work for this one; therefore, deactivate this default behavior by clicking the **Edit->Solution** menu, click the **Advanced** button on the *Solution Information* property sheet and click the **Calculate Initial Estimates** check-box to remove the check. If you do not deactivate the calculation of initial estimates, you will get an error message during an optimization stating that the numerical solution has become unstable. This means that the Marquardt procedure could not come up with an improved estimate for T and S. One problem with the Marquardt method is that your initial guess for T and S must be fairly close to the “right” answer before it will work properly. You will get another message after the error dialog that asks if you would like to turn off the option to automatically estimate initial guesses for parameters. Selecting **Yes** on this dialog does the same thing as removing the check from the **Calculate Initial Estimates** check-box.

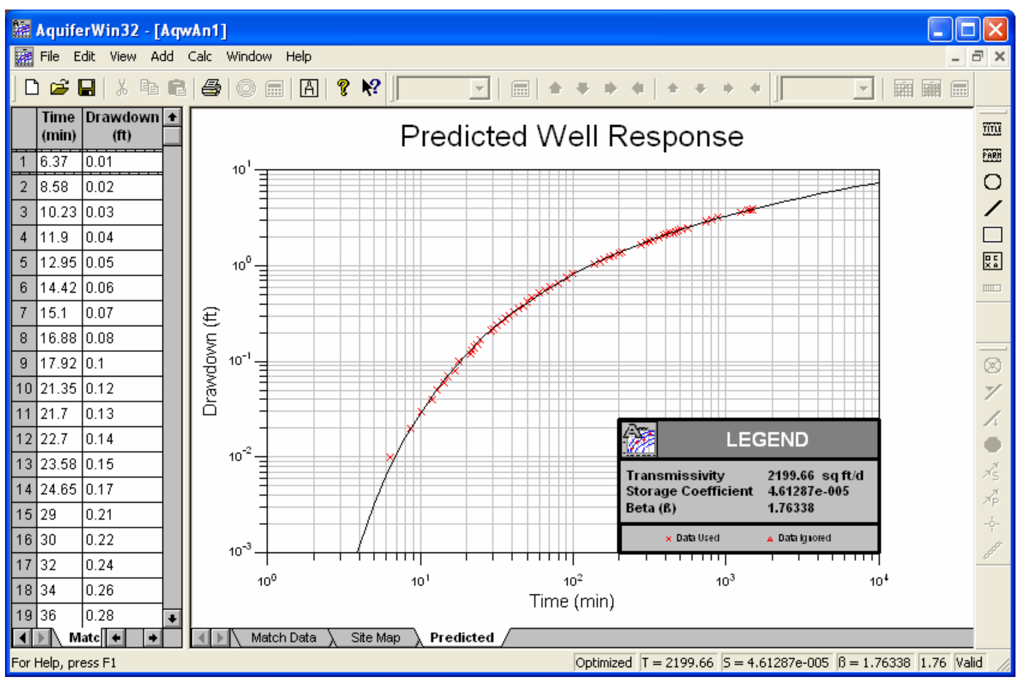
Click the  button on the Match toolbar to analyze the test results using the nonlinear least-squares technique. The solution should converge to a T value of about 2,190 to 2,200 ft^2/d, an S value of about 4.6e-05, and a beta value of 1.76.

In order to have the correct type curve on the graph (for a beta value of 1.76), select **Edit->Solution**, click on the ***Curves*** tab, select “2.00” in the *Curve Information* list box, click the **Edit** button, and change the value to “1.76”. Click the **OK** button on the dialog and the **OK** button on the property sheet. The type curves will then recalculate. Select the 1.76 type curve and click the calculator button. The values obtained by Lohman were 2,170 ft^2/d and 3.0e-05 for T and S, respectively.

In order to add a legend to the graph, click the **Add->Legend** menu and drag a rectangle anywhere on the screen, click the **Finish** button on the Legend Wizard that appears. This will add the default legend to the graph. Your screen should look similar to the one shown below.



If you are familiar with other aquifer test software, you will see that Aquifer^Win32 has a slightly different philosophy or focus. Whereas most other aquifer test analysis packages rely heavily on the automatic matching procedures, Aquifer^Win32 is designed to be more like the visual or manual matching techniques advocated by most hydrogeology text books (See Fetter, 1994 for example) and by ASTM. An alternative view of the match results is presented in the ***Predicted*** view tab. Click on this tab and also click on the **View->Clip Data** menu to clip the predicted curve to the graph window. Repeat the above procedure to add the default legend. The resultant graph displays the observed drawdown versus time data and predicted drawdown versus time curve. This graph can be annotated and modified as required and an example is shown below.

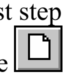


Using Multiple Observation Wells

What you will learn:

- *analyzing an aquifer test with multiple observation wells*
- *advanced discussion of unit conversions*
- *more about adding legends*
- *optimizing the solution for all wells and for individual wells*
- *more about using AquiferWin32 as an OLE server*

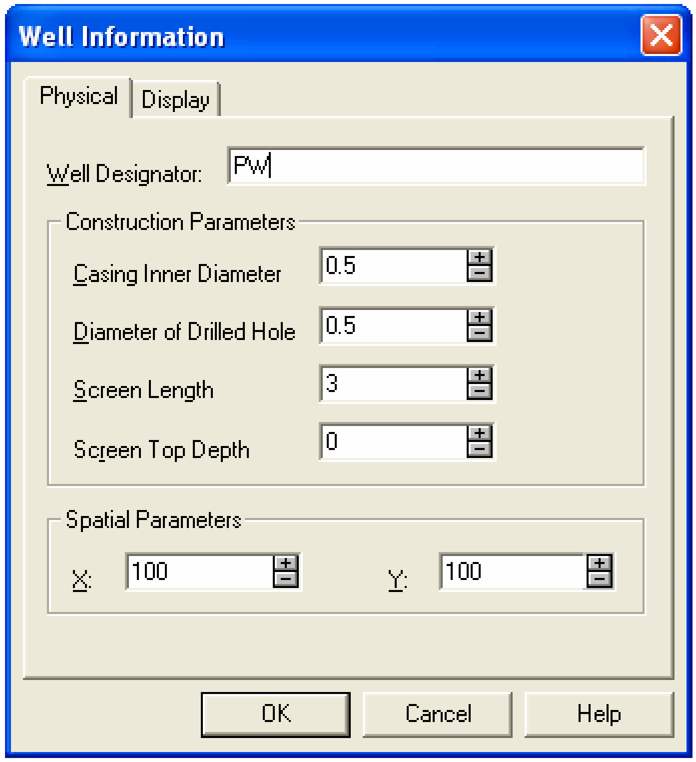
The remaining tutorial examples cover advanced topics in Aquifer^{Win32}. These examples are designed to show you how to perform unit conversions, annotate plots, and work with complex data sets.

This section outlines how to analyze a multiple well pump test. The particular data are taken from Lohman, (1979; page 19). The first step is to create a new Aquifer^{Win32} Analysis document by clicking on the  button on the Standard toolbar. Once the document is created, the first step is to set the units to be used in the analysis. The **Edit->Units** menu displays the *Unit Information* property sheet with two tabs. The first tab, ***Summary***, contains the four basic parameters and their respective units. The values represent either the program defaults or the values previously saved as default. Under normal circumstances, you would set the summary units to most closely approximate the units of the data collected. In this instance, we will intentionally deviate from the norm in order to demonstrate unit changes activated from other parts of the program. Set *Time Units*, *Length Units*, *Pumping Rate Units* and *Transmissivity Units* to “sec”, “ft”, “gal/min” and “sq ft/d”. Click the **Save As Default** check-box and the **Apply Globally** check-box. There is no need to click the **Convert Data** check-box because no data exist yet. (**NOTE: Whenever a new parameter or well is created in Aquifer^{Win32}, the default values saved in this way are used to set the initial units.**)

Before exiting the property sheet, click on the ***Units*** tab. This tab allows specific parameters to be assigned units. Aquifer^{Win32} allows the ultimate flexibility when it comes to setting units; however, it is imperative that one understands the paradigm. Every parameter contained in Aquifer^{Win32} has units that can be expressed in terms of three generic "units"; these are length (L), time (T) and volume (V) with the possible values organized into three sets of radio buttons. When a parameter is selected in the *Parameter* combobox, the appropriate radio button sets are activated. Simply select the desired unit from each radio button set. One of the most complicated parameters, Transmissivity, can be expressed in dimensionless units as V/T/L. If you select gallons, days and feet for V, T and L the resulting units would be gallons per day per foot (gal/d/ft). If you select cubic feet, days and feet, the resulting units would be square feet per day (sq ft/d). (**NOTE: If changes are made to both tabs of this property sheet, the values on the *Summary* tab take priority**.) Accept the property sheet by clicking the **OK** button.

Since we are setting up a multiple well scenario, click on the ***Site Map*** view tab to activate it. Now we should set up the map via the **Options->Map->Parameters** menu. Click on the ***Window*** tab and set both *Height* and *Width* to “1000”. If we had a basemap of the site, we would click on the ***File*** tab and enter a map file name

which would automatically set the map window dimensions based on the map being imported. (**NOTE: Unlike other ESI software, the map is stored with the document so that it is self-contained**.) Accept the property sheet and add the wells by using the **Add->Well** menu or clicking the  button on the Analytic toolbar. In either case, the cursor will change to perform a drag insert. Move the cursor to the desired location on the map and click the left mouse button. For our purposes, put the cursor anywhere and edit the *Well Information* property sheet as below.

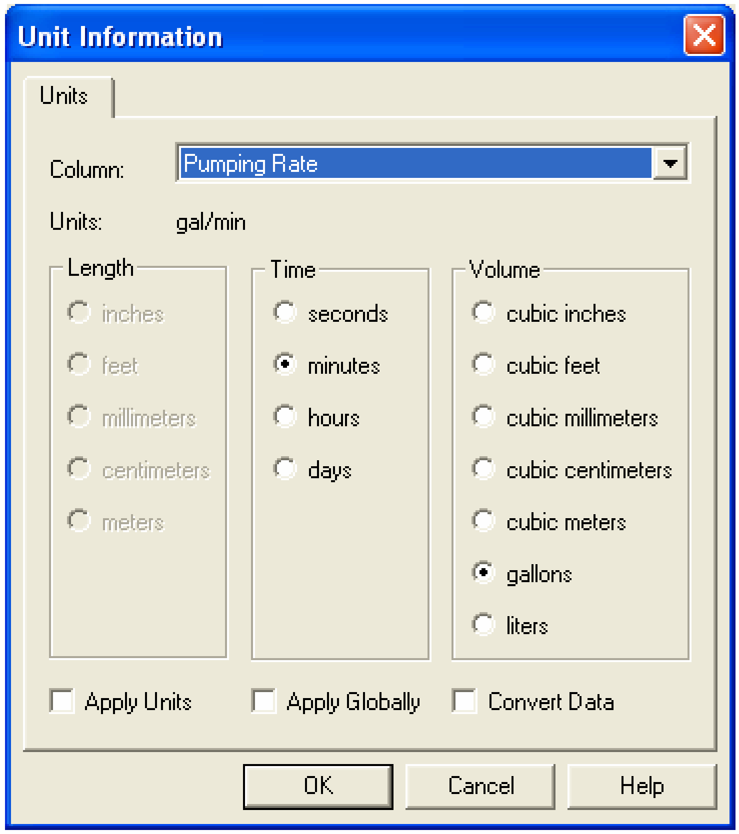


Although it is a good idea to fully define the pertinent construction parameters, it is not required for this example. The reason is that the Theis analysis (which we intend to use) does not use any of the well construction parameters. Click on the ***Display*** tab and click on the **Display** check-box so that the well name, PW, is displayed and click on the **OK** button. In similar fashion, add three more wells using the default well construction information:

Name	X	Y
N-1	100	300
N-2	500	100
N-3	900	100

Now that the "Site" has been defined, the next step is to define the "Aquifer Test". Click on the **Edit->Aquifer Test** menu. The first tab of the *Aquifer Test Information* property sheet contains user-fields. These user-fields can be used to annotate any Graph or Map View and are also included in the hard copy report. Click on the ***Pumping*** tab and select “PW” in the *Pumping Well* combobox. Now we must define the pumping schedule. Click the left mouse button in the *Pumping Rates* spreadsheet or **Tab** into it from the *Pumping Well* combobox. Hit the **Ins** key or right click the

mouse within the spreadsheet and select **Insert** from the context menu. We now want to set the pumping rate to 96,000 ft3/d but the units on the column are gallons per minute (gpm). Right click the mouse within the spreadsheet and select **Column Conversion** to display the *Unit Information* property sheet as below.



Select "Pumping Rate" in the *Column* combobox and set the *Time* units to "days" and the *Volume* units to "cubic feet". Click on the **Apply Units** and **Apply Globally** check-boxes and click on the **OK** button. The column title will change to reflect the new units.

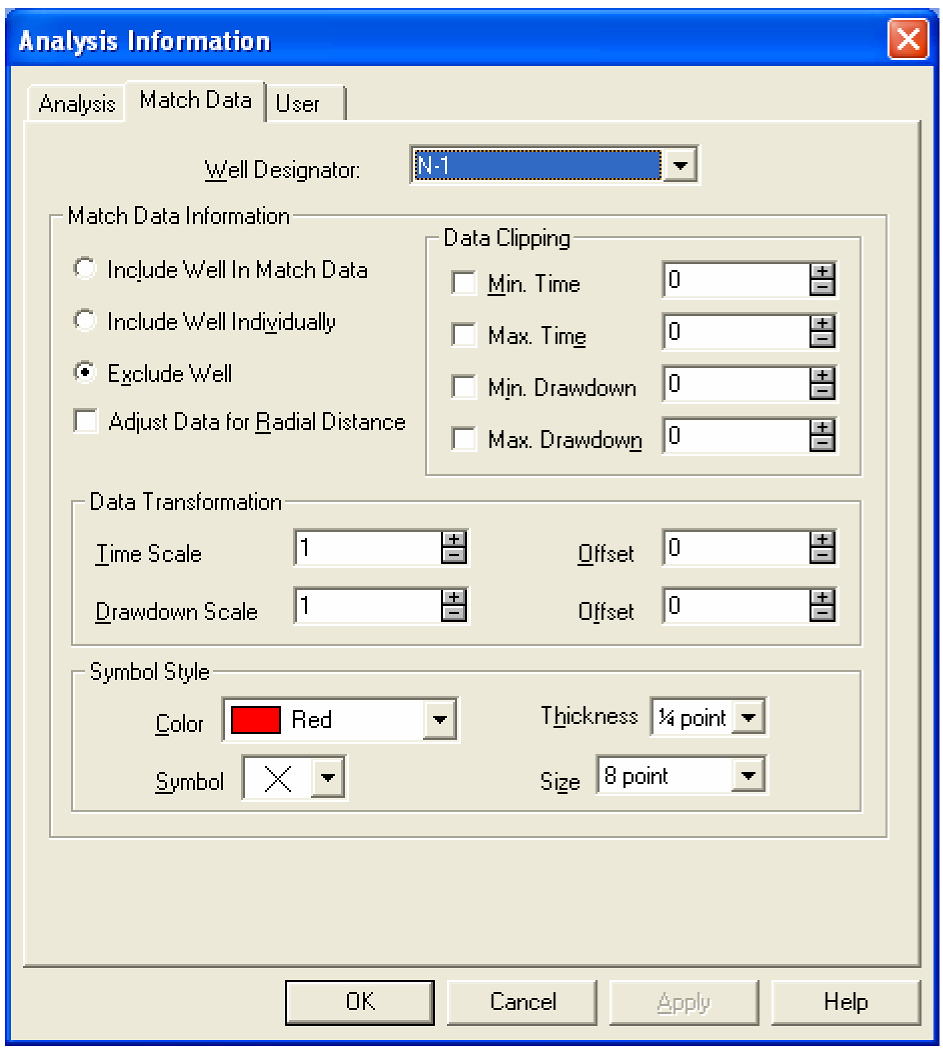
To enter the data, click the mouse in spreadsheet row 1 and column 2 and enter "96000". At this point, if you were to hit the **Tab** key or the **Enter** key a new data row would be added and you could continue entering the pumping schedule. Hit the **Esc** key once to clear the field. Now lets assume you really want your pumping rates displayed in gpm. Right click the mouse to bring up the context menu and activate the **Column Conversion** menu. With "Pumping Rates" selected in the *Column* combobox, set the *Volume* units to "gallons" and the *Time* units to "minutes". Activate **Apply Units**, **Apply Globally** and **Convert Data** then click on the **OK** button. The pumping rate is now displayed as 498.667 gal/min.

The next step is to define the wells monitored during the test. Click the ***Wells*** tab and move wells N-1, N-2 and N-3 from the *Available Wells* list to the *Monitored Wells* list by clicking the **Add** button three times. (**NOTE: This and other Add/Remove list box pairs are Drag and Drop enabled so you can drag items from one list box to the other.**)

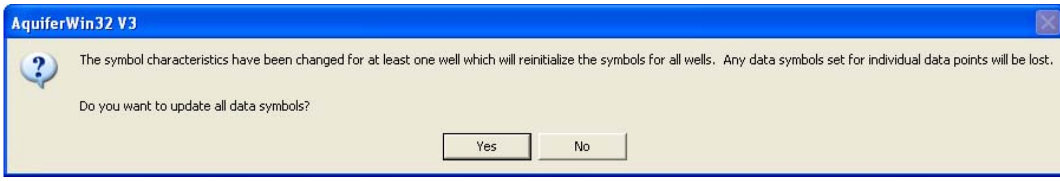
To enter the drawdown data for the monitoring wells, click on the ***Well Data*** tab. This spreadsheet has tabs along the bottom for the ***Match Data*** and each of the monitored wells. Click on the ***N-1*** tab to begin entering the data. By now you probably have experience entering data into the spreadsheet using the keyboard so

we will import the data from an ASCII file. Click the right mouse button within the spreadsheet to activate the context menu and select **Import**. Import the file *n-1.dat* which should be contained in the Aquifer^{Win32} directory. Repeat the procedure for the other three monitored wells using *n-2.dat* for well N-2 and *n-3.dat* for well N-3. Since this tutorial attempts to represent real world, we must now correct a mistake. The time data contained in the ASCII files are represented in minutes and the units on the *Time* columns in the respective spreadsheets is seconds. Activate the context menu (right click the mouse) and select **Column Conversion**. Set the *Column* combobox to "Time" and click on the "minutes" radio button. Click on the **Apply Units** and **Apply Globally** check-boxes and then click the **OK** button to perform the unit change. The **Apply Globally** check-box made the unit change to all time data throughout the data file saving us some time. Obviously, global unit changes without data conversion must be done with care.

After we accept the *Aquifer Test Information* property sheet by clicking the **OK** button, we are ready for the next step. You will note that the ***Match Data*** spreadsheet to the left of the map has only changed in that the units on the *Time* column are now minutes due to our global time unit change. We must now define the "Analysis" by using the **Edit->Analysis** menu. The *Analysis* tab contains additional user fields and determines how the pumping rate is calculated from the pumping schedule. In this case, the default, *Calculate Time Average Pumping*, is acceptable. Click on the ***Match Data*** tab as shown below.

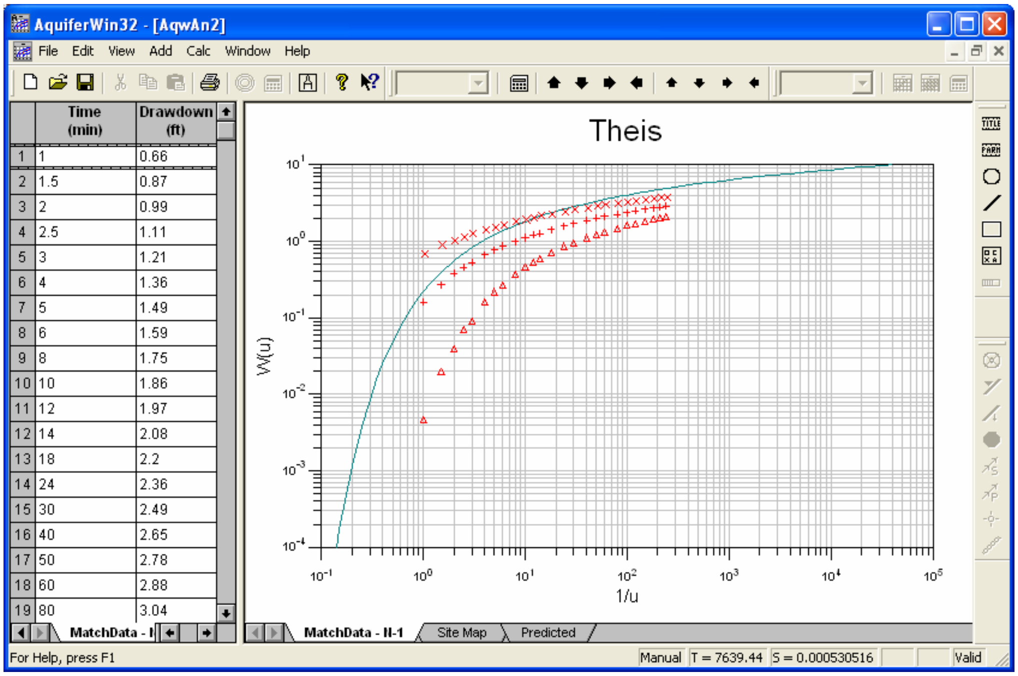


There are various ways to analyze this problem so we will begin with the most traditional which is to combine all the wells together adjusted by radial distance. To accomplish this, click on the *Include Well In Match Data* radio button and the ***Adjust Data for Radial Distance*** check-box. Change the selection in the *Well Designator* combobox to "N-2" and repeat the previous settings. Also, change the *Symbol Style* to something other than the default. Repeat the procedure for well N-3 and click the **OK** button. Since we have made changes to the symbol styles, the following dialog will be displayed to confirm that we want to reset all symbols. Click the **Yes** button.



The spreadsheet has now been populated with the resulting data to be analyzed. At this point the columns of the spreadsheet may need to be expanded to better see the column titles and data. The easiest way is to click within the spreadsheet and select the **View->Optimize** menu which will make the columns just wide enough to display their contents. Click on the ***Match Data*** tab to display the Type Curve Graph. Since we have made some unit changes without converting the data, it is good practice to click the **Calc->Reset Data Offset** menu to reinitialize the data offsets. Notice that the data is nowhere to be seen. This is because the *Adjusted Time* coordinates are much lower than the minimum for the graph. You can either repeatedly hit the right arrow key until they come into view or optimize the solution via the **Calc->Optimize** menu. In this example, optimizing the solution gives a Transmissivity value of 13376.4 ft2/d and a Storage coefficient value of .000201527.

Another way to analyze the same data is to include each well individually and optimize them as a group. Choose the **Edit->Analysis** menu and click on the ***Match Data*** tab. For each monitoring well click on the *Include Well Individually* radio button and click on the **Adjust Data for Radial Distance** to stop dividing the time data by radial distance squared. Click on the **OK** button and the window would look like the one below.



Now that there are multiple data sets being analyzed individually, things are more complicated; however, this flexibility allows more detailed analysis of the data. The group optimization procedure allows total control over which parameters are optimized across data sets. Before we delve into the intricacies of this procedure it is important to understand how to work with multiple data sets. The first thing to understand is that each well has its own set of analysis parameters that can be manipulated individually. To demonstrate, manually match the currently selected data set, N-1, to the type curve using the arrow keys. Fine tune the match by holding down the shift key while using the arrow keys. To see how close your visual match is to the "optimized" match, activate the **Calc->Optimize** menu. Toggle between data sets using the **Edit->Toggle Data Set** menu or the accelerator key **Ctrl-D**. Notice that the view tab changes to "*Match Data - N-2*" to indicate the active well. Also, note that the data values in the spreadsheet change to reflect the active well. Repeat the manual match and optimize match for this data set and for the N-3 data set. As you toggle between data sets the status bar reflects the differing values calculated for Transmissivity and Storage Coefficient. The same results could be achieved by simply using the **Calc->Optimize Group** menu which will optimize all data sets individually because we have not set any parameters to be optimized as a group.

Using this procedure, you can determine the variability of the calculated parameters (Transmissivity and Storage Coefficient) to data from each well. To simulate the previous results when all data were adjusted by radial distance and treated as one data set, use the **Edit->Group Optimize Parameters** menu. The *Group Optimize Parameters* property sheet is an Add/Remove list box pair in which the *Available Parameters* list box contains all analysis parameters that are free to vary in each data set. Click the **Add** button twice to move both “Transmissivity” and “Storage Coefficient” to the *Optimized Parameters* list box and click the **OK** button. Now, clicking the **Calc->Optimize Group** menu will optimize all three data sets such that the best value of Transmissivity and Storage Coefficient are determined. Click the **Calc->Optimize Group** menu and toggle through the data sets. Notice that the value of Transmissivity and Storage Coefficient displayed in the status bar are all the same and, not coincidentally, equal the values calculated using the more traditional analysis we performed first.

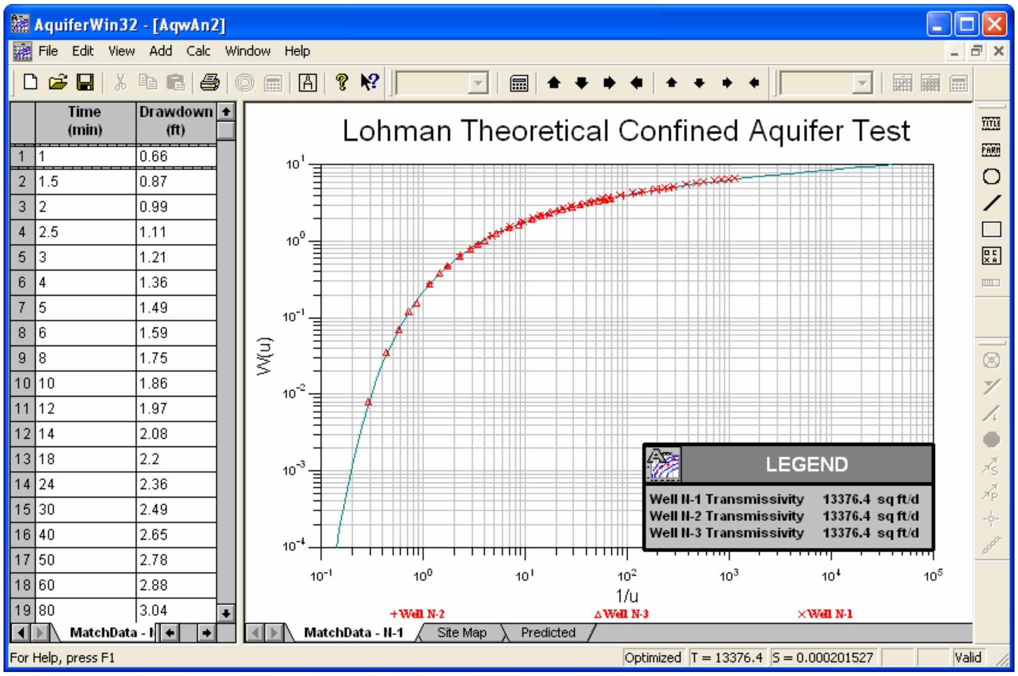
The final step in any analysis is to prepare a graph for the report. The first thing to do is add a legend to the graph indicating which symbols represent the well data. Click on the **Add->Legend** menu and drag a rectangle along the bottom of the window. After dragging a rectangle, the *Legend Wizard* will be displayed. As before, just clicking finish is sufficient to add the default legend. In this case, we want to click **Create Empty Legend** to create a different type of legend.

After the legend box is displayed, double click on it to display the *Legend Information* property sheet. Set the *Spatial Parameters* such that *X1* equals “1”, *Y1* equals “.25”, *X2* equals “9” and *Y2* equals “0”. Click on the ***Contents*** tab and click the **Display Border** check-box to deactivate it. Next click the ***Items*** tab and click on the **Automatic Maintenance of Graph Data Sets** check-box to activate it. Finally, click the **OK** button to accept the changes. A legend will be displayed containing the symbol and well name for each data set. In order to edit the label and/or font used, use the **Edit->Graph** menu to display the *Graph Information* property sheet. Click on the ***Line Styles*** tab, set the *Data Set* combobox to “N-1” and enter "Well N-1" in the *Label* field. This will override the default label which is the name of the well. Click on the **Font** button and change the *Font* to “Times New Roman”, the *Font Style* to “Bold”. Click the **OK** button and then set *Color* to “Red”. Repeat the procedure for N-2 and N-3 and click the **OK** button to accept the changes.

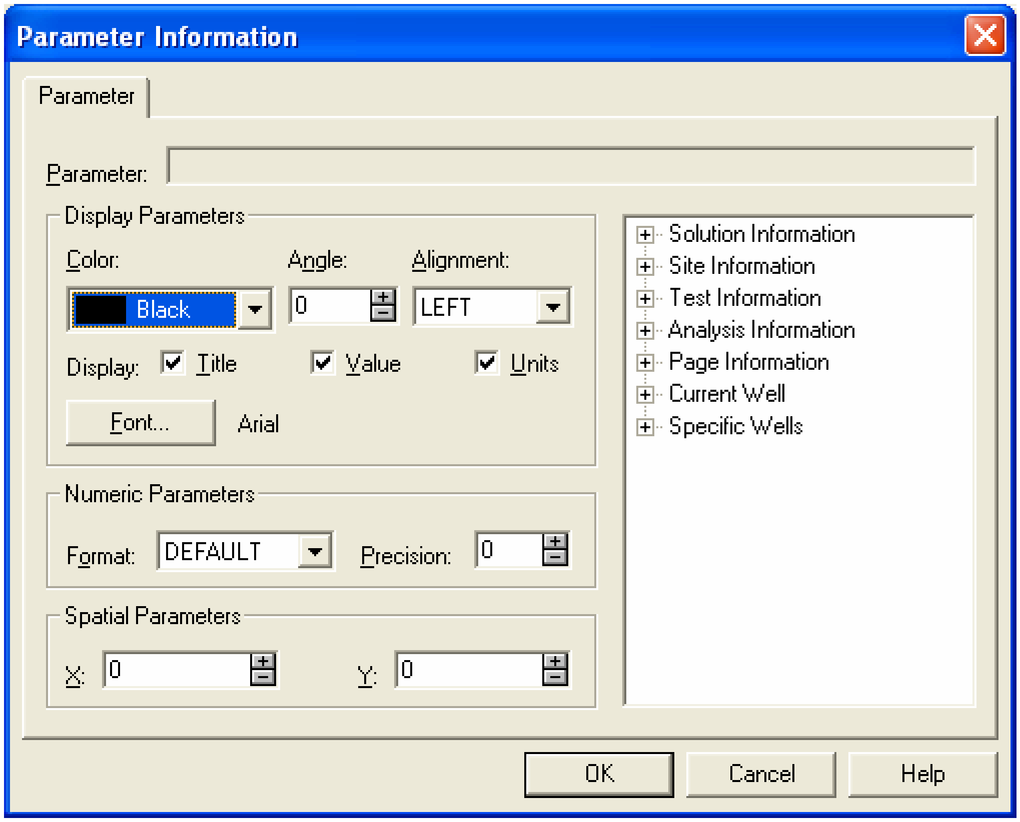
Add a more meaningful title to the graph by right clicking on the graph and selecting **Graph** from the context menu. On the ***Graph*** tab, change the *Title* field to be

"Lohman Theoretical Confined Aquifer Test" and click on the **OK** button. Now annotate the graph to display the calculated Transmissivity for all three wells.

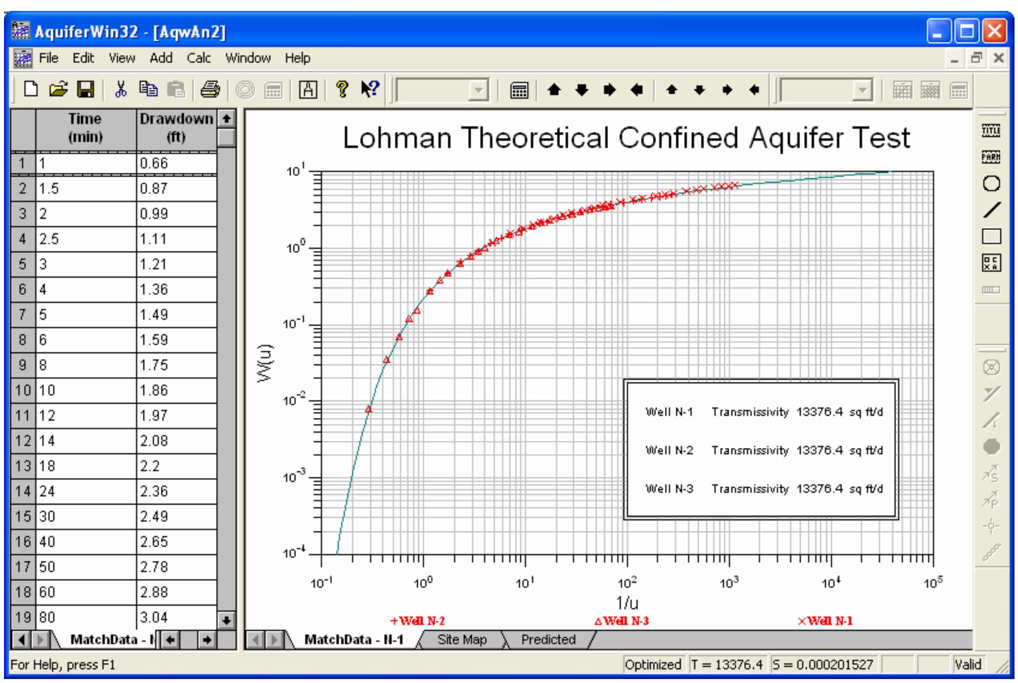
Click the **Add->**Legend menu and drag a rectangle starting from the lower right corner of the graph into the graph body. The size does not matter only the starting point since the legend size will be automatically adjusted. On the first page of the *Legend Wizard* click the **Automatically Adjust and Locate Legend** checkbox to uncheck it, click the **Create custom legend** checkbox to check it and then click the **Next** button. Remove Transmissivity and Storage Coefficient from the right list box since these represent the current well values which change as you toggle between wells. Now, drill down into the tree control into Specific Wells/N-1/Analysis Parameters and select Transmissivity. Click the Append button to add it to the right list box. Now, edit the Title field to be "Well N-1 Transmissivity". Repeat the procedure for the other two wells. Click the **Next** button twice, click the **Symbol Sublegend** check box to remove it from the legend and click the **Finish** button.



In order to demonstrate the manual procedure, you could accomplish a somewhat less complex legend as follows. Use the **Add->Legend** menu and drag a rectangle in the lower right area of the graph. On the *Legend Wizard* click the **Create empty legend** radio button and click the **Finish** button. Double click the mouse inside the legend you just crateed to display the *Legend Information* property sheet. Set *X1* equal to "5", *Y1* equal to "1.5", *X2* equal to "8.5" and *Y2* equal to "3.25". Click on the ***Contents*** tab and set the *Type* combobox to "Solid". Click on the ***Items*** tab and set the *Item Type* combobox to "Title Object" and click on the **Add** button. Set the *Title* field to "Well N-1" and *X*, *Y* to ".25", "1.25". Click on the **OK** button. Repeat the add procedure for wells N-2 and N-3 using *X*, *Y* of ".25", ".75" and *X*, *Y* of ".25", ".25" respectively. Now change the *Item Type* combobox to "Parameter Object" and click the **Add** button. The following property sheet should be displayed.



In the Tree Control List Box, click on the + sign for "Specific Wells" to reveal the detail. Continue to drill down into "Specific Wells/N-1/Analysis Parameters/Transmissivity" and select "Transmissivity". Set the *X* and *Y* values to "1.1" and "1.25". Add two more parameters for N-2 and N-3 using *X*, *Y* values of "1.1", ".75" and "1.1", ".25". Accept the legend by clicking on the **OK** button. The resulting graph is displayed below.

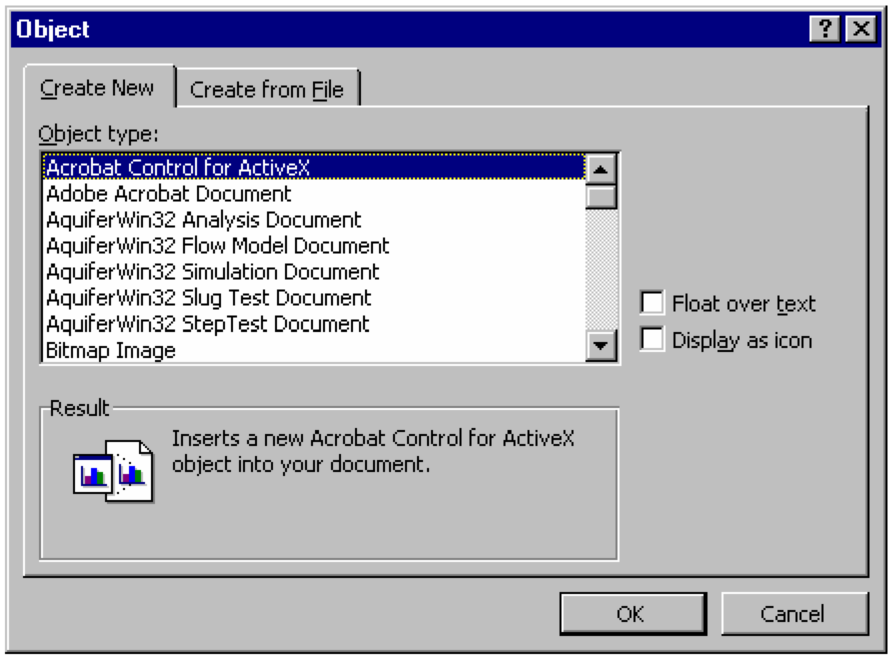


Time (min)	Drawdown (ft)
1	0.66
1.5	0.87
2	0.99
2.5	1.11
3	1.21
4	1.36
5	1.49
6	1.59
8	1.75
10	1.86
12	1.97
14	2.08
18	2.2
24	2.36
30	2.49
40	2.65
50	2.78
60	2.88
80	3.04

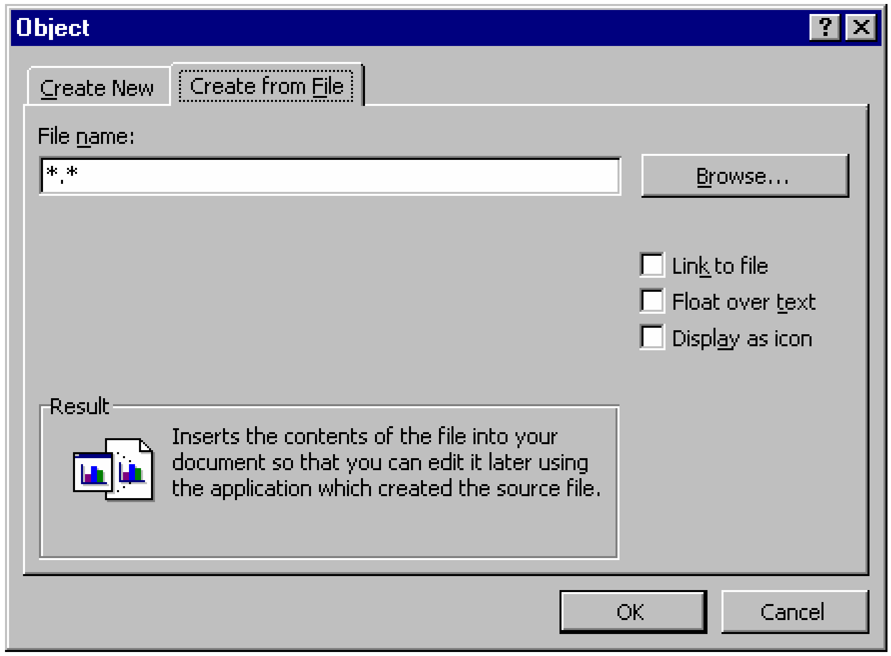
Now that the analysis has been performed and the beautification procedure is done, activate the **Edit->Group Optimize Parameters** menu and remove "Transmissivity" and "Storage Coefficient" from the *Optimized Parameters* list box by clicking the **Remove** button twice. Accept the property sheet by clicking the **OK** button. Activate the **Calc->Optimize Group** menu and notice that the legend is updated at the end of each iteration. When the process has completed, you now have a graph in which the data from all three monitoring wells have been optimized independently and the resulting transmissivity values are displayed.

Now save the analysis document via **File->Save As** and name the file *multwell.aqw* and exit the program.

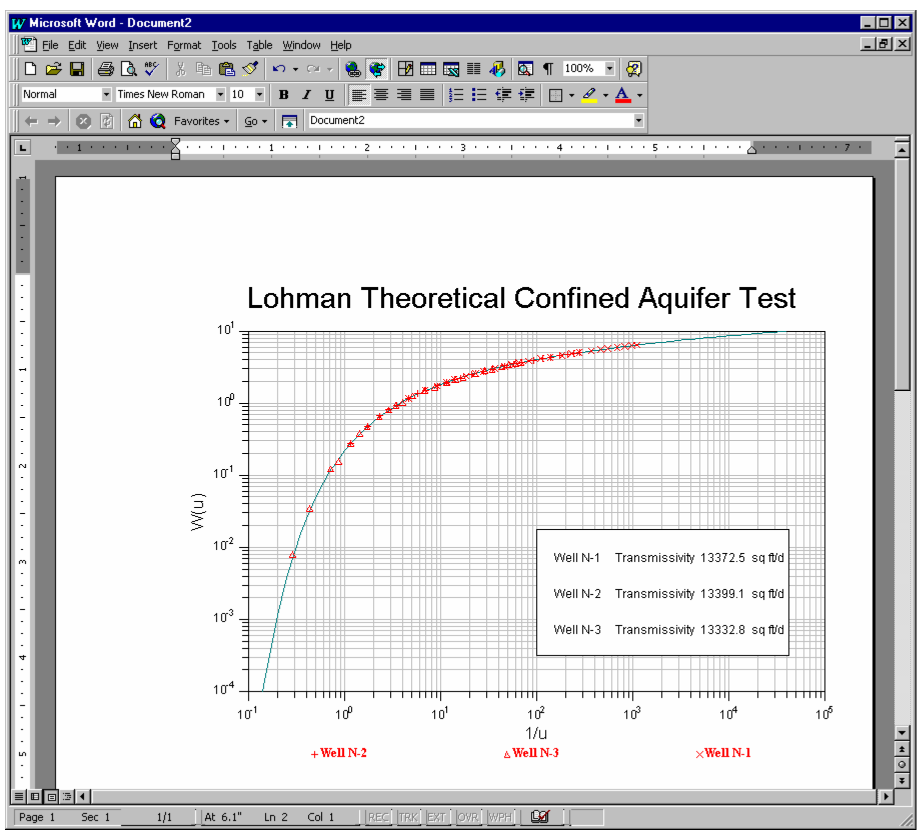
Suppose that you are generating your final report using an OLE Container program such as Microsoft Word. Since Aquifer^{Win32} is an OLE Server, you can link the newly created analysis document directly into the report such that subsequent changes to the analysis will be automatically updated in your report. To perform this operation, start up your OLE Container (e.g. Microsoft Word) and create a new document. Click on **Insert->Object** menu and you should get a property sheet something like this:



Notice that there are entries in the *Object Type* list box for the four Aquifer^{Win32} document types. If you were to select one of them and click the **OK** button, you would find yourself in Aquifer^{Win32} editing a new document of the selected type. Such a document would then remain an editable part of the document totally self-contained. Since we want to create an OLE Link click on the ***Create from File*** tab which will display the following property sheet.



Click on the **Browse** button and locate the file *multwell.aqw*, click the **Link to File** check-box and close the property sheet using the **OK** button. When all the window flashing and disk access stops, the graph from your analysis document will be part of the Word document.



Double click the mouse on the graph and you will find yourself in Aquifer^{Win32} editing *multwell.aqw*. Click the mouse within the graph in Aquifer^{Win32} and hit the down arrow key. Use the menu **File->Save** to save the change and change windows to the OLE client software in normal windows fashion. Notice that the graphic within the Word document has changed to reflect the change made to the Aquifer^{Win32} analysis.

Analyzing Recovery Tests

What you will learn:

- *analyzing the recovery of an aquifer test*
- *alternative solution using variable pumping*

The first tutorial demonstrated how to analyze a simple aquifer test, one in which there is only one observation well. This tutorial shows how to analyze the recovery data from that same test taken from a real aquifer test reported by Kruseman and de Ridder (1990; page 59).

As previously stated, Kruseman and de Ridder call this aquifer test Oude Korendijk for the area where the test was conducted. The pumping well was screened over the entire aquifer thickness of 7 meters. Piezometers were placed at distances of 0.8, 30, 90, and 215 meters. The well was pumped at a constant discharge rate of 0.547 m^3/min for 14 hours. This tutorial will use time and drawdown data including recovery for the 30 meter piezometer.

Entering Data

Start a new aquifer test by clicking the button on the Standard toolbar or select **File->New**. The default graph type is for the Theis confined analysis which we will change after the data has been entered. The first thing you should do when starting a new analysis is to set the units for the pumping test data. Select **Edit->Units** to tell Aquifer^{Win32} what units you are using. In this case, select minutes (min) for *Time Units*, meters (m) for *Length Units*, meters cubed per minute (cu m/min) for *Pumping Rate Units,* and square meters per minute (sq m/min) for *Transmissivity Units*. Click the **Apply Globally** and **Save As Defaults** check-boxs at the bottom of the property sheet to set these units for the entire analysis including all data types and accept the property sheet by clicking the **OK** button.

Import the drawdown versus time data including recovery from the file *fulltest.dat*. To import the data from a text file, click on the first line in the spreadsheet to highlight that row. Select **File->Import** and find the file. Aquifer^{Win32} will notify you that 51 lines were imported. If necessary, delete the default 1,1 data point.

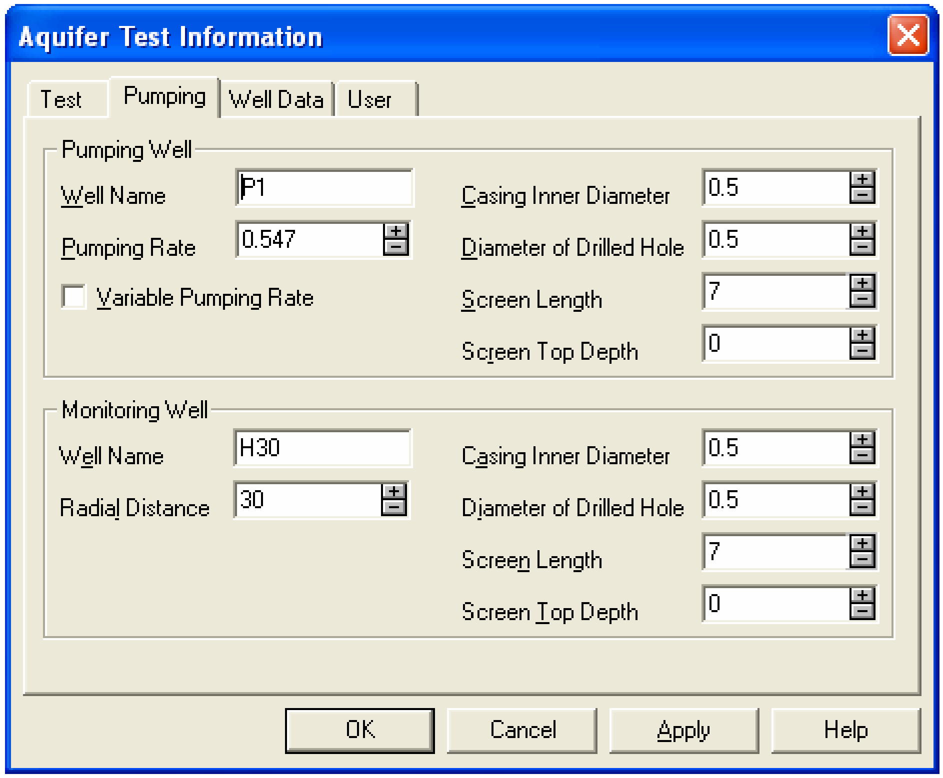
It is important to note that the data for a recovery test is the same as for a pumping test. Time is relative to the start of pumping and drawdown is relative to the original static water level before pumping began. You do not have to delete the data from the pumping test because the recovery analysis will automatically ignore it.

Now that we have the time and drawdown data, we will enter the remaining test data. Select **Edit->Aquifer Test** to enter the pumping rate and distance to observation well. Click on the ***Pumping*** tab on this property sheet and enter the following information:

Pumping Well Name	P1
Pumping rate	0.547 m^3/min.
Pumping well screen length	7 m
Monitoring Well Name	H30

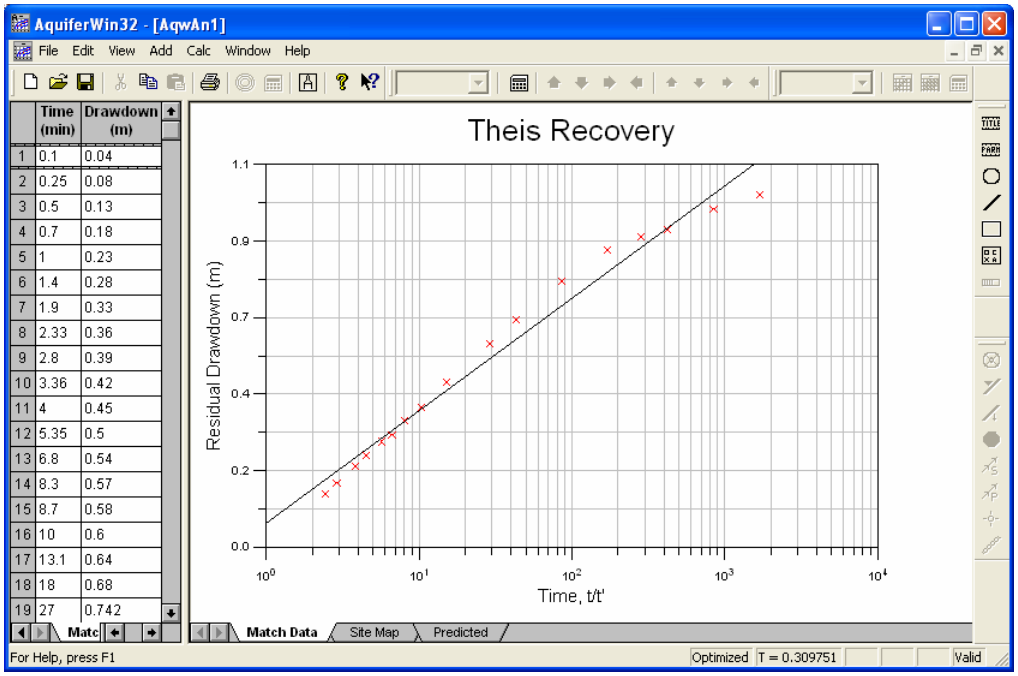
Radial Distance	30 m
Screen length	7 m

The property sheet should resemble the one below. Click **OK** when you are done.

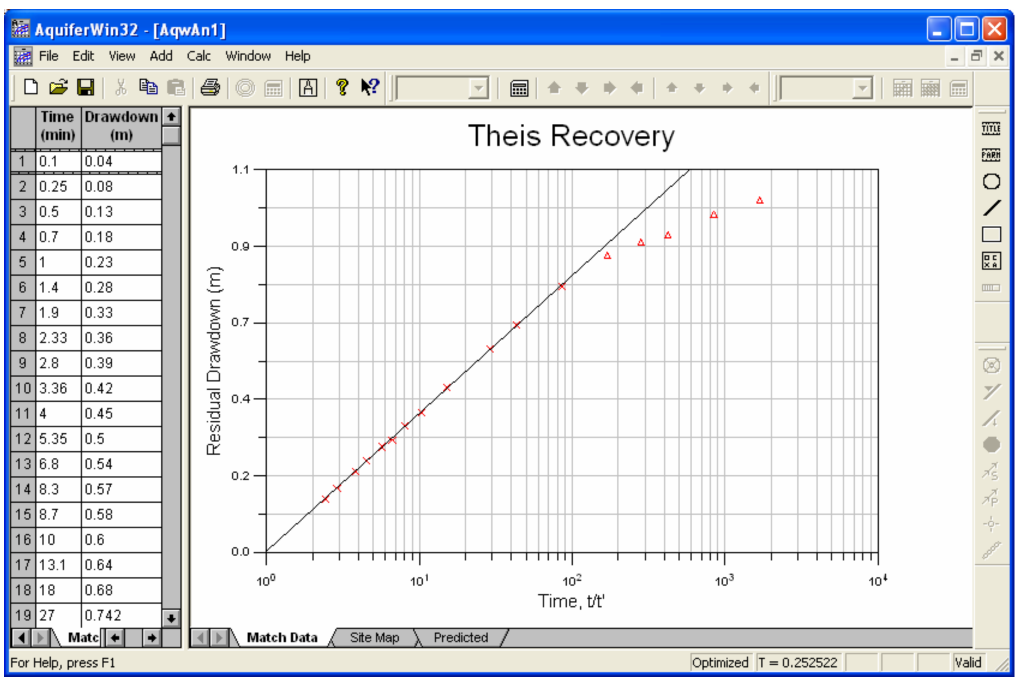


Traditional Analysis

At this point, click the **Edit->Solution** menu to activate the *Solution Information* property sheet. Select "Theis, 1946 (Recovery)" in the solution combobox. Click the ***Parameters*** tab and enter "830" into the *Time Recovery Starts* edit field. Make sure the units are in minutes by right clicking in the edit field and selecting **Units** from the context menu; make sure the Time Units are minutes. (**NOTE: Forgetting to enter the Time Recovery Starts correctly is the most common mistake made using this analysis; it is used to properly convert the data for the analysis.**) Accept the property sheet by clicking the **OK** button. The resulting graph is shown below. The data points prior to recovery are not displayed and a linear regression has been performed to calculate a transmissivity of .31 m^2/min.



In a recovery test, as with other tests, the late data is usually assumed to be more representative. In this case, the late data is on the left of the graph because it has been scaled during the analysis. Click the **Edit->Aquifer Test** menu and click on the **Well Data** tab. Change the weight on lines 35 through 39 to 100 which will remove them from consideration for the purposes of linear regression. Also, change the symbol to a triangle for the same data points to visually indicate which points are being ignored. Accept the property sheet by clicking the **OK** button. Now click the **Calc->Optimize** menu to recalculate an answer.



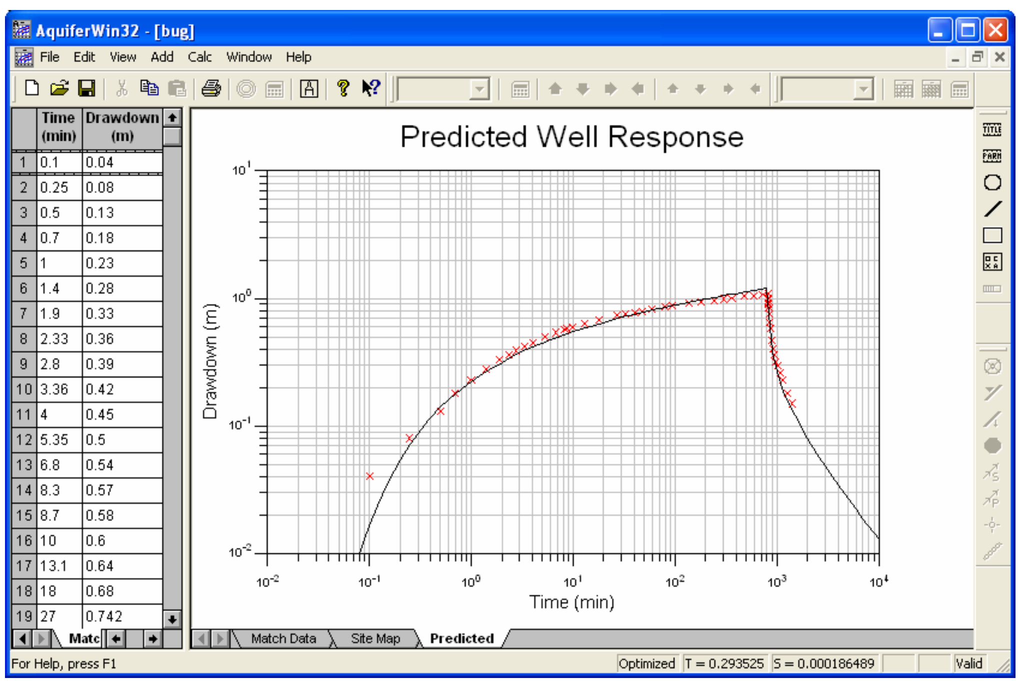
Variable Pumping Analysis

The linear regression Theis recovery analysis is the traditional method for analyzing recovery data; however, Aquifer^{Win32} offers another alternative. Since Aquifer^{Win32} supports variable pumping rates, we can analyze the pumping and recovery data simultaneously. To set up for this analysis we can define and use wells on the site map or continue using a simple solutions without wells defined.

Without Using Wells

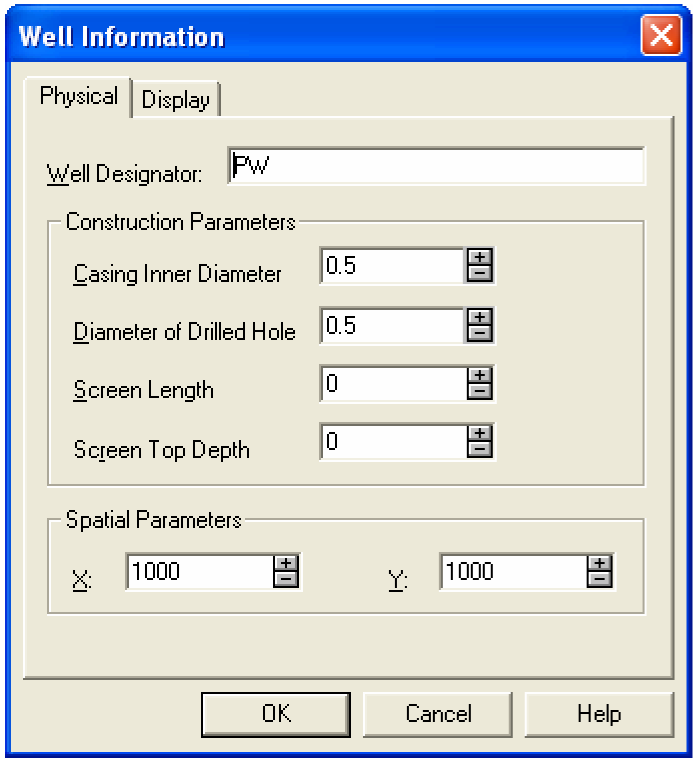
Click the ***Match Data*** view tab followed by the **Edit->Solution** menu. Set the analysis to “Theis, 1935 (Confined)”. Click the **Advanced** button to activate the *Advanced Solution Information* dialog box and click the **Calculate Initial Estimates** check-box to unset it. Accept the property sheet by clicking the **OK** button. Next, click the **Edit->Aquifer Test** menu to activate the *Aquifer Test Information* property sheet. Click the ***Pumping*** tab and set the *Variable Pumping Rate* checkbox. Click the ***Rates*** tab that was added to the property sheet and right click the mouse on the spreadsheet and click the **Insert** menu. Click the mouse in the first row of the Pumping Rate column and enter the value .547. Hit the **Return** key to add another line to the spreadsheet and enter 830. Hitting the **Return** or **Tab** keys would advance to the Pumping Rate column but 0 is the value we want so click the **OK** button.

Since data points had been modified with respect to weight and symbol, we need to set them back. Click the **Edit->Aquifer Test** menu and click on the **Well Data** tab. Change the weight on lines 35 through 39 back to 1 and change the symbol back to an X. Accept the property sheet by clicking the **OK** button. Finally, click the **Calc->Optimize** menu to optimize the solution. Click the ***Predicted*** view tab to review the results presented below. If the curve extends beyond the limits of the graph, click the **View->Clip** Data menu.

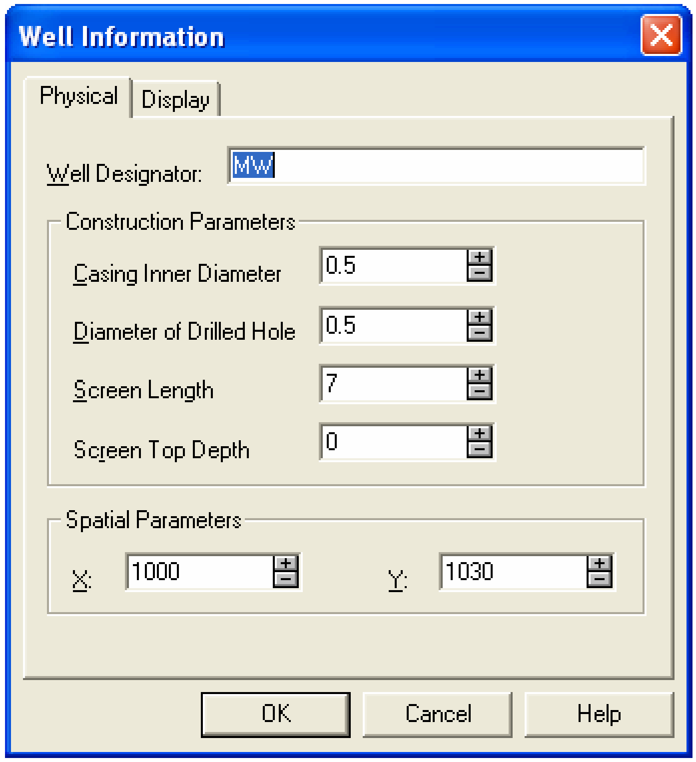


Using Wells

To set up for this analysis we must add wells to the site map to set up for a multiple well analysis. Click the ***Site Map*** tab on the view to activate the map. Click the **Add->Well** menu and left click the mouse somewhere on the map. The *Well Information* property sheet will be displayed. Enter the values as below.



Repeat the procedure to add the monitoring well and enter the values as below.

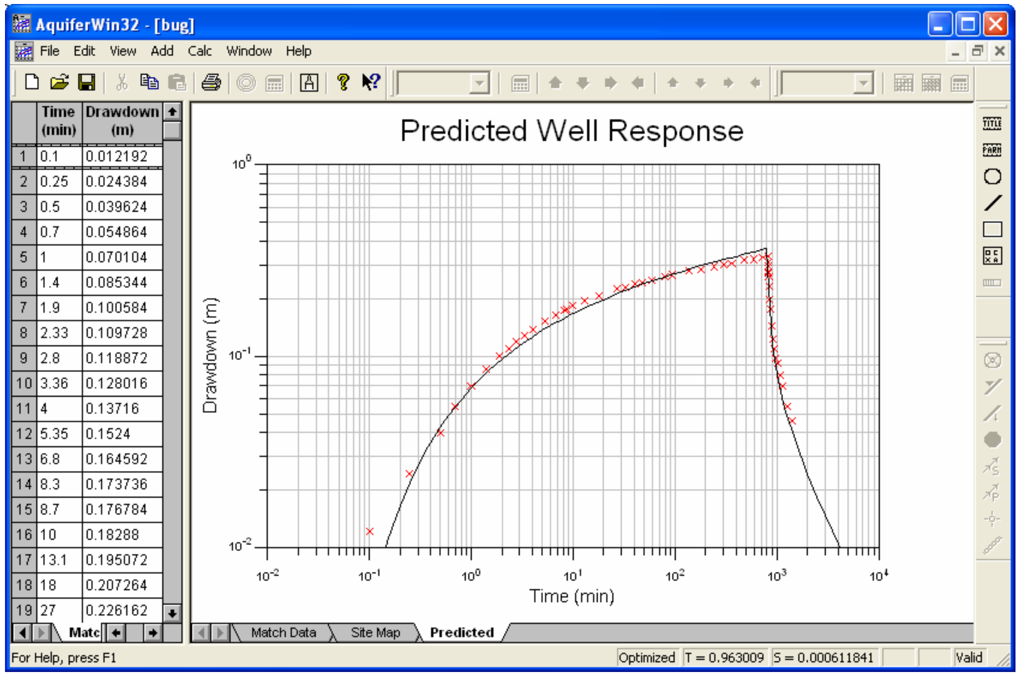


Click the ***Match Data*** view tab followed by the **Edit->Solution** menu. Set the analysis to "Theis, 1935 (Confined)". Click the **Advanced** button to activate the *Advanced Solution Information* dialog box and click the **Calculate Initial Estimates** check-box to unset it. Click the ***Parameters*** tab and unfix *Transmissivity* if it has been fixed by clicking on the push-pin check-box adjacent to the field. It may have been set to fixed by the previous linear regression analysis. Accept the property sheet by clicking the **OK** button. Next, click the **Edit->Aquifer Test** menu to activate the *Aquifer Test Information* property sheet. Click the ***Pumping*** tab and set the *Pumping Well* combobox to "PW". Click on the *Pumping Rates* spreadsheet and enter the pumping schedule. To add a line to the spreadsheet, hit the **Ins** key and change the *Pumping Rate* to ".547" and hit the **Tab** key to advance to the next line adding another data point. Enter a *Time* of "830" and a *Pumping Rate* of "0". Click the **Wells** tab and click the **Add** button to move "MW" from the *Available Wells* to the *Monitored Wells* list box. Click the **Well Data** tab to enter the time/drawdown data. Since the time/drawdown data is already in the ***Match Data*** tab of the spreadsheet from the previous analysis, right click the mouse on the spreadsheet and select the **Select All** menu item from the context menu. Copy the information using **Ctrl-C** from the keyboard or **Copy** from the context menu. Click the ***MW*** tab of the spreadsheet and paste using **Ctrl-V** or **Paste** from the context menu. Accept the property sheet by clicking the **OK** button.

Next, click the **Edit->Analysis** menu to activate the *Analysis Information* property sheet. Click the **Variable Pumping** radio button to use the pumping schedule we entered previously. Click the **Match Data** tab, select "MW" in the *Well Designator* combobox, click the **Include Well in Match Data** check-box and accept the property sheet using the **OK** button.

Now that the data has been entered, perform a manual match against the type curve ignoring the recovery data points to get a good initial estimate for the parameters.

Finally, click the **Calc->Optimize** menu to optimize the solution. Click the ***Predicted*** view tab to review the results presented below.



Analyzing Slug Tests

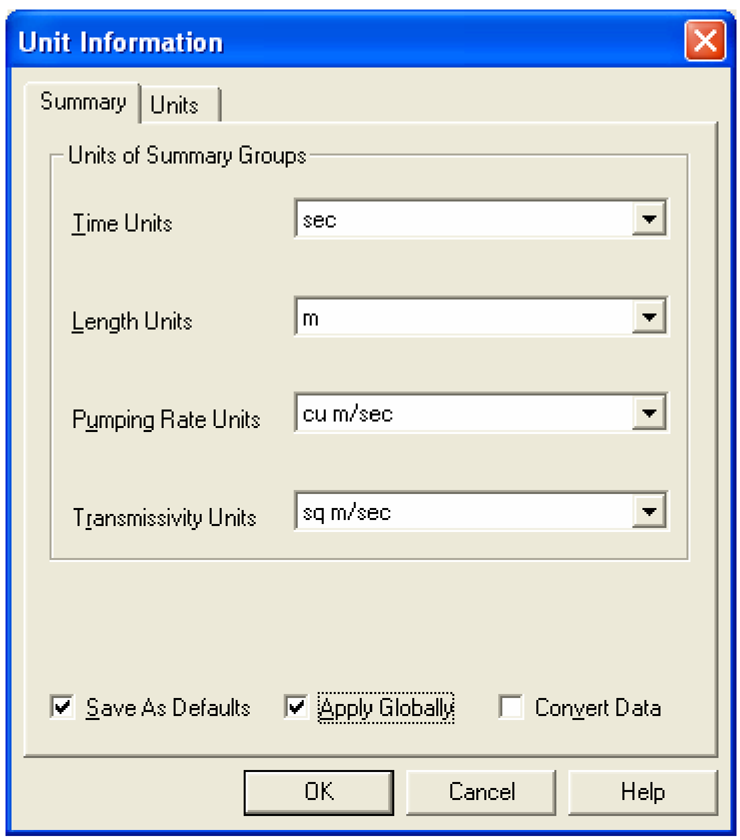
What you will learn:

- *analyzing a slug test*
- *using different slug test analysis techniques*
- *more about unit conversions*

Slug tests seem to be routine in many field investigations because they are simple and cost-effective to perform. If you are looking for a procedure to perform all slug tests, this tutorial may be the answer. The first step is to use the **File->New** menu to create a new document since the  button on the Standard toolbar defaults to an "AquiferWin32 Analysis" document. Select "AquiferWin32 Slug Test" from the list box and click the **OK** button. The default analysis for a slug test is the traditional linear regression version of Bouwer and Rice.

The next step is to enter the time and displacement data. The particular data set we will use is taken from Lohman (1979; page 29). The data represent recovery in a well under confined conditions.

To continue our theme of making life a little easier, the data is stored in the file *dawson.dat*. Before importing the data, however, activate the spreadsheet view by clicking the mouse in the column title part of the spreadsheet. Use the **Edit->Units** menu and set the values as below.



Accept the property sheet by clicking the **OK** button. Next, import the data by right clicking the mouse on the spreadsheet and selecting **Import** from the context menu. Locate the file *dawson.dat* and load in the data. Notice that the import has left the initial data value 1,1. Click the left mouse button on the line number column (leftmost column) and hit the **Del** key to delete the row. Now, click the mouse in the graph view and, resisting the temptation to use the **Calc->Optimize** menu, click on the **Edit->Aquifer Test** menu. At this point, skip the user fields and click on the ***Well*** tab. It is important to complete the well construction information on this tab because the analysis requires it. The field book says that: "the well is cased to 24 m below the top of the aquifer with 6 inch casing and drilled as a 15.2 cm open hole to a depth of 122 m and the theoretical initial displacement should be .56 m". Put a meaningful value in the *Well Name* field like "Well". Now enter the *Casing Inner Diameter* which is initially set to ".5" meters. If you are unsure of the units, move the mouse cursor over the *Casing Inner Diameter* field and wait. The units "m" will be displayed as a tool tip and the status bar will display "Casing Inner Diameter (m)".

Lets take a moment to learn about units and unit conversions. In this case, we have a field that wants meters and we have a value in inches. Let us assume that our calculator battery died. You now have several options. The obvious first option is to do the conversion manually and enter the value. A second option is to right click on the *Casing Inner Diameter* field, select the **Units** menu, change the units to inches, click the **OK** button and enter 6 into the field. The system doesn't care what units you use because it internally converts all data before performing calculations. You can go wild and use whatever units you want on a parameter by parameter basis. (This is not recommended if you are using Windows 3.x since the tool tip reminder as to unit values doesn't work!) For our purposes here, click the **Cancel** button to cancel the changes.

Click on the **Edit->Aquifer Test** menu again and feel free to fill in any user fields on the ***Test*** tab before clicking to the ***Well*** tab. After entering something meaningful in the *Well Name* field, enter "6" in the *Casing Inner Diameter* field. Right click on the *Casing Inner Diameter* field and select **Data Conversion** from the context menu. Click on inches in the *Length* radio buttons of the ***From*** tab of the embedded property sheet to indicate the units of the previously entered value. Click on the ***To*** tab and notice that the units to convert to are not editable and reflect the prevailing units of the parameter being edited. Now click the **Convert** button followed by the **OK** button. The value of 6 inches has been converted to meters and the field has been updated with the converted value.

To complete the data entry, type ".1524", "98" and "24" in the *Diameter of Drilled Hole*, *Screen Length* and *Screen Top Depth* fields. Finally, enter ".56" into the *Initial Displacement* field and accept the property sheet by clicking the **OK** button.

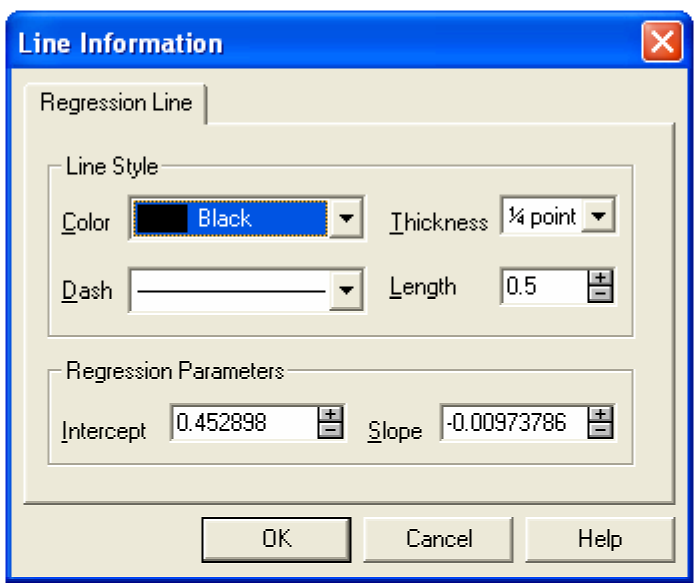
The next step is to click on the **Edit->Solution** menu and update any additional parameters that are required for the analysis. Since this is a linear regression solution, the *Convergence Information* on the ***Solution*** tab is disabled. Click on the ***Well Parameters*** tab and note the previously entered well construction parameters are being used and there is one additional parameter. The *Gravel Pack Porosity* defaults to ".3" which is fine. In fact, unless the water is recovering within the screen or open hole interval, this value is not used by the analysis. It is important to note at this point the custom check-boxes to the right of the fields. The lower check-box indicates whether the parameter is "linked" to a value entered at another section of the program. In this case, the first four parameters are linked to the values previously enter in the *Aquifer Test Information* property sheet. Should we choose to do so, we can unlink any or all of the values and enter an override value. This action will not change the linked value and the linked value can be restored by relinking the parameter. The upper check-box is not used in a linear regression analysis; however,

it determines if the parameter is free to vary in a curve match scenario. A linked value is always fixed so it has to be unlinked to be estimated by the program.

Click on the ***Hydraulic Parameters*** tab and change the *Aquifer Thickness* to “124”. If we wanted to simulate recovery within the well screen, the *Correction Type* should be set to “Recovery within Screen”. Now click on the ***Additional Results*** tab and notice additional parameters are present. The *Effective Casing Inner Diameter* and *Effective Screen Inner Diameter* are calculated values used by the analysis that differ from the *Casing Inner Diameter* and *Screen Inner Diameter* only if recovery is occurring within the screened interval. There are also three additional calculated parameters which are specific to this analysis. An attempt has been made to provide you with all the values you would need should you want to do the calculations on paper. Accept the property sheet by clicking on the **OK** button.

Now the time has come to calculate the optimized solution. For this analysis, the optimized solution is achieved via linear regression. Click in the graph view and click the  button on the Match toolbar or use the **Calc->Optimize** menu. Notice that a linear regression has been performed on the data and the Bouwer and Rice solution has been calculated. What a minute, this is supposed to be a confined analysis and the value for initial displacement should be .56 not the .453982 calculated by linear regression. Lets ignore for the moment that we know the solution is confined.

There are two primary ways to determine the initial displacement, H0, and the slope of the line. The first way is to manually set it. Move the mouse cursor over the linear regression line near the center of the graph. The cursor will change to the NSEW cursor (the one with arrows pointing up, down, left and right). Double click the left mouse button and the *Regression Line Information* property sheet is displayed.



Actually, this property sheet is the best way to set H0 so change *Intercept* to the value we know to be cast in stone, “.56” meters. Now if you are a veteran slug tester or if you read Bouwer and Rice, you know that the slope of the line and the well construction parameters are the only parameters that are involved in the calculation. How then do we proceed? Accept the property sheet by clicking the **OK** button. You now have a "best fit line" that doesn't hit any of the data points. (There are actually three lines on your screen; the outermost lines are actually part of the rectangle that surrounds the line and indicates that the line has been selected. If you

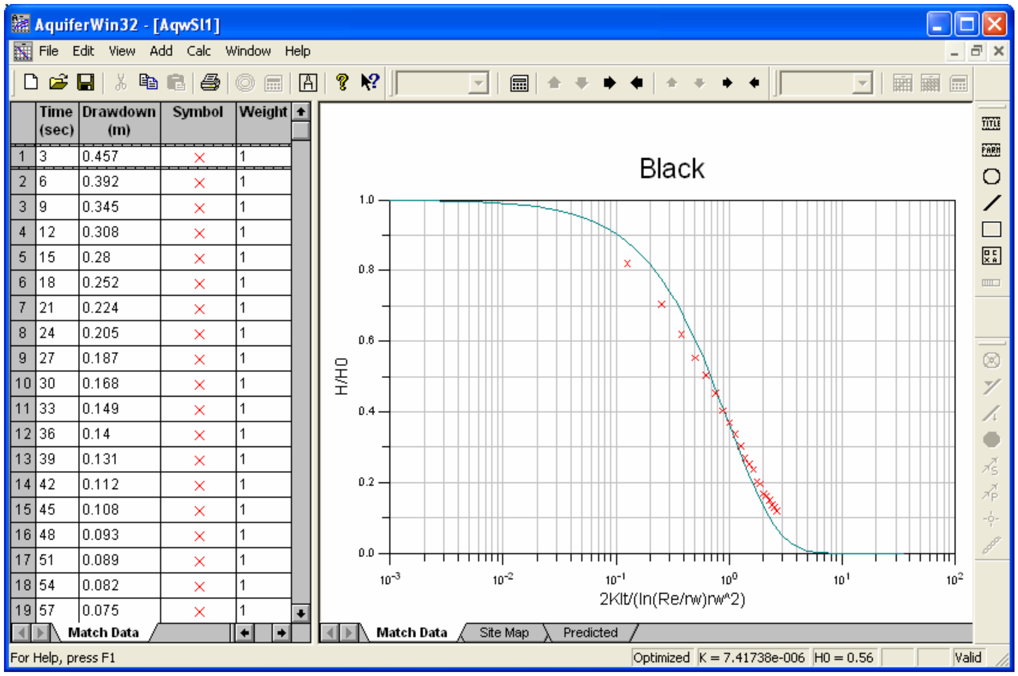
click the left mouse button outside of this rectangle the selection will disappear.) Now, move the mouse cursor over the rightmost section of the line. The cursor will change to an EW (arrow left and arrow right) cursor indicating that you can move it. Click the left mouse button, hold it down and drag the end of the line around until you are satisfied that you have generated an acceptable match to the data. Notice that as you drag the line around, the calculated value of hydraulic conductivity, K, changes too. If you were to move the entire line, the calculated value of K would not change because only the slope matters. Incidentally, the same procedure works for the left portion of the regression line so you never have to use the *Regression Line Information* property sheet as you did above; simply move the left portion of the line to reset the H0 (intercept) value. However you slice it, you have full control on the solution just as you would if you were doing it on paper.

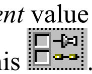
The second approach is for those people who are drawn to the optimize button. How can we change the "optimized" solution? Right click the mouse on the spreadsheet and select the **Columns** menu from the context menu. An Add/Remove list box pair is displayed showing that there are two additional columns available to you. Move both the weight and symbol columns from the *Available Columns* to the *Displayed Columns* by clicking the **Append** button twice and click the **OK** button. Where are they? Use the **View->Optimize** menu to optimize the spreadsheet so that all columns are visible and only as wide as they need to be**. (NOTE: You can also access these data from the *Well Data* tab on the Aquifer Test Information property sheet**). Change the weight on data rows 5 through 21 to a number of 100 or greater. For all linear regression analyses, a weight of 100 or greater removes the point from consideration during the linear regression calculation. You can also change the symbol displayed for these data by double clicking the mouse on the current symbol and changing it. This would give a visual indication of which data were ignored. In this case, H0 is calculated to be .516 and not the theoretical value of .56; however, it is not always possible to know the theoretical H0 and the data may be the only indicator as to its value. The important point is that you have lots of ways to perform the analysis.

Since manually changing the weights and symbols can be time consuming, you could drag select lines 5 through 21 and click the **Edit->Selection** menu. On the *Selection Edit Options* property sheet, click on the ***Weight*** tab. Click the **Set Value** radio button and enter 100 into the adjacent edit field. Now click on the ***Symbol*** tab, click the **Set Value** radio button, change the *Symbol* combobox to a triangle and accept the property sheet. This option is easier for bulk changes to the spreadsheet.

Now lets assume you are convinced that the Bouwer and Rice solution is appropriate. How would you know? Aquifer^{Win32} takes a step toward answering that question with a second Bouwer and Rice solution base on a curve match. Change the weights all back to 1 and all the symbols back to the same style if you changed them. You may also want to slide the splitter bar over to hide the *Weight* and *Symbol* columns. You could also remove the columns in an analogous manner to the one used to add them earlier.

Click on the **Edit->Solution** menu and select "Black, 1978 (Unconfined Aquifer)" in the *Analysis* combobox. If you look through the property sheet, you will notice that nothing has changed from Bouwer and Rice. Click the ***Hydraulic Parameters*** tab and click on the push pin check-box next to the *Hydraulic Conductivity* field if it is currently checked. The previous linear regression analysis may have initialized the value to fixed. Accept the property sheet by clicking **OK** and attempt to match the data to the curve using the arrow keys. Use the **Edit->Aquifer Test** menu to activate the *Aquifer Test Information* property sheet and click on the ***Well*** tab. Change the *Initial Displacement* to ".56" and try again. Notice that the data is automatically rescaled to the new value and doesn't match any better. Obviously, this data doesn't represent an unconfined aquifer response as shown below.



Another step to try is to click the **Edit->Solution** menu and go to the ***Hydraulic Parameters*** tab. Unlink and unfix the *Initial Displacement* value by clicking both of the check-boxes to the right of the field so it looks like this . Accept the property sheet by clicking the **OK** button. Now click **Calc->Optimize** button and look at the results. This process will optimize both T and H0 but doesn't result in a good match. The validity of this step is determined by how confident you are with the value of H0.

Since the unconfined solution did not give an acceptable match and because we know the response is confined, we will move on to the confined solution. Click on the **Edit->Solution** menu and select "Cooper, Bredehoeft & Papadopulos, 1967 (Confined Aquifer)" for a confined slug test analysis. Click on the ***Hydraulic Parameters*** tab and relink the *Initial Displacement* by clicking on the lower of the two check-boxes to the right of the field. Now click on the **OK** button. The system will calculate the 10 standard type curves for this analysis. Use the arrow keys to match the data to the third curve from the left. Hit the **Ctrl-T** key twice to select that curve (the currently selected type curve changes color), fine tune your match and look to the status bar for the answer.

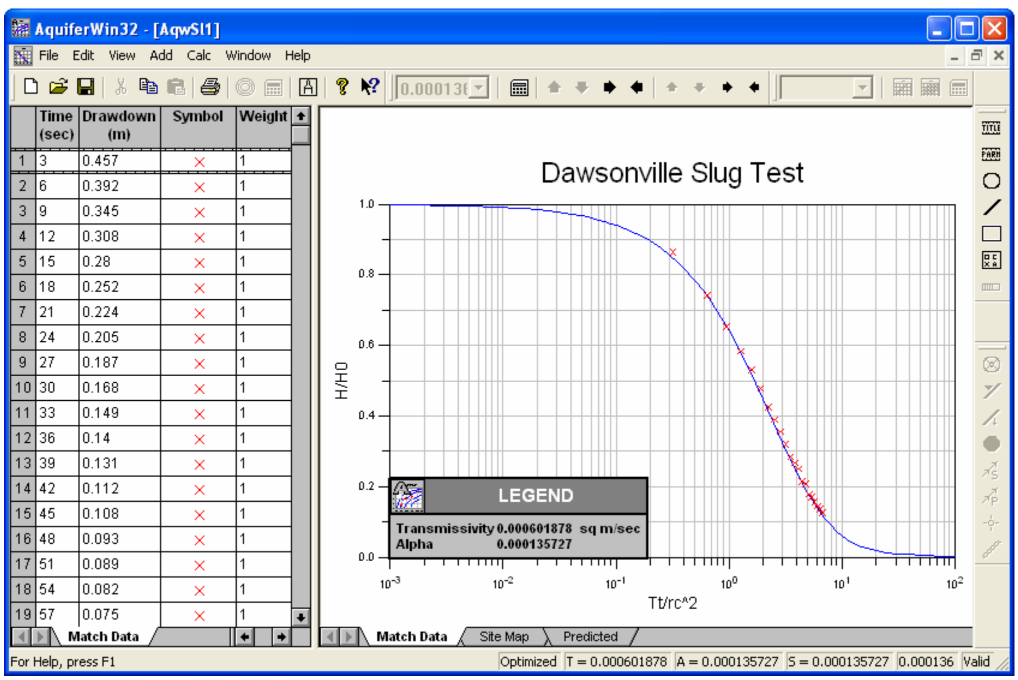
To get an optimized solution, go ahead and select the **Calc->Optimize** menu. Notice that now the optimized solution has matched to a curve that is not displayed. Click on the **Edit->Solution** menu and click on the ***Hydraulic Parameters*** tab and click on the little push pin check-box to the right of the *Alpha* field. This will "fix" the parameter and remove it from optimization. Set the value to .001 and click on the **Apply** button to optimize the match. This would essentially optimize the manual match we did earlier. Click on the little right arrow at the upper right of the property sheet to advance the tabs and click on the ***Statistics*** tab to quantify the "accuracy" of the curve match. The residual standard deviation is .004379. Now click on the ***Exceptions*** tab, select "Initial Displacement" in the *Parameter* combobox and click on the **Linked Value** check-box followed by the **Fixed Value** check-box; this tab provides an alternative way to link/unlink and fix/unfix parameters and also shows you where range information can be set for each parameter. It is sometimes necessary to enforce minimum and maximum parameter values to achieve an optimized match. In this case, they are not necessary so click on the **Apply** button.

Notice that the calculated value for *Initial Displacement* has changed to .547477 and a look at the ***Statistics*** tab reveals a residual standard deviation of .003467.

How do I optimize the solution for Alpha and get the curve on the graph? Go back to the ***Hydraulic Parameters*** tab and click on the push pin check-box next to Alpha and click on the **Apply** button. Now click on the ***Hydraulic Results*** tab to see the calculated values or look in the status bar. Go to the ***Curves*** tab and decide what curves you want on the graph. Remove all but one of the values from the *Curve Information* list box by selecting them and clicking the **Delete** button. Select the one remaining value and click on the **Edit** button. Change the value to ".000136", change the *Label Format* to "EXPONENTIAL" and change the *Precision* to "6" and click the **OK** button on the property sheet. Click the **OK** button on property sheet and there is your optimized answer. If you are not concerned with having the exact type curve represented on the graph you can skip this last step and use the ***Predicted*** view tab which displays the observed displacement data with the predicted displacement curve through the data.

The final step in any analysis is to prepare a graph for the report. The first thing to do is add a legend to the graph indicating what the symbols represent. Click on the **Add->Legend** menu and drag a rectangle starting at the lower left hand corner of the graph. The size does not matter but the starting point is the anchor point for the legend. After dragging a rectangle, the *Legend Wizard* will be displayed. Click the **Create custom legend** radio button and uncheck the Automatically Adjust and Locate Legend checkbox. Click the **Next** button three times to go to the ***Styles*** page; uncheck the **Symbol Sublegend** checkbox and click the **Finish** button.

Add a more meaningful title to the graph by right clicking on the graph and selecting **Graph** from the context menu. On the ***Graph*** tab, change the *Title* field to be "Dawsonville Slug Test" and click on the **OK** button. The resulting graph is displayed below.



Simulating Aquifer Tests

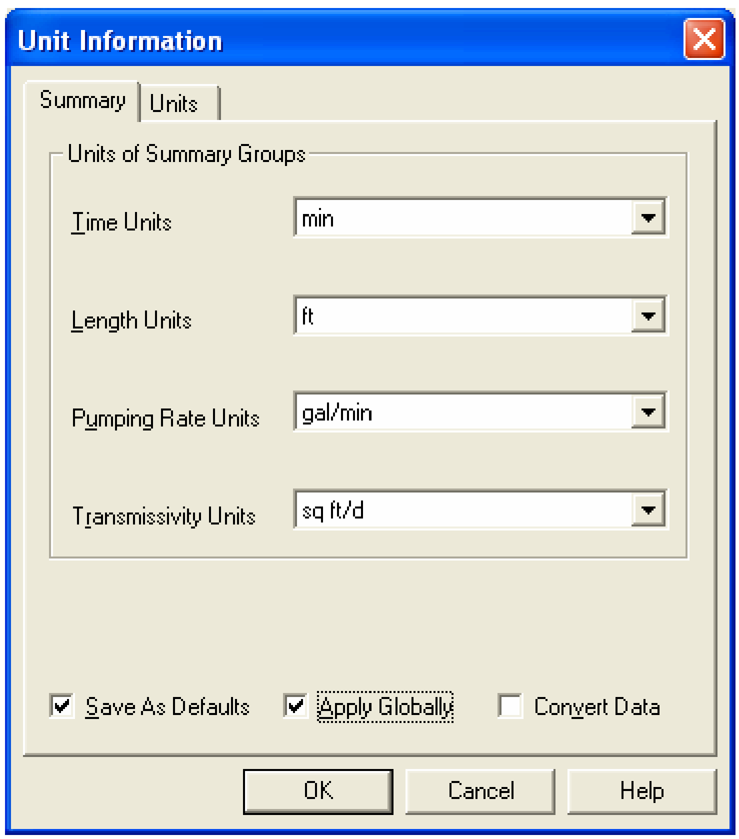
What you will learn:

- *simulating an aquifer test*
- *plotting simulated drawdown contours*
- *plotting simulated hydrographs at observation wells*

The *Professional* version of Aquifer^{Win32} can also be used to simulate pump tests. If you have an idea of the type of response and hydraulic properties of an aquifer, you can estimate the aquifer response to a given pumping rate. The response can be depicted as contour maps at given times and drawdown versus time graphs at any number of monitoring wells.

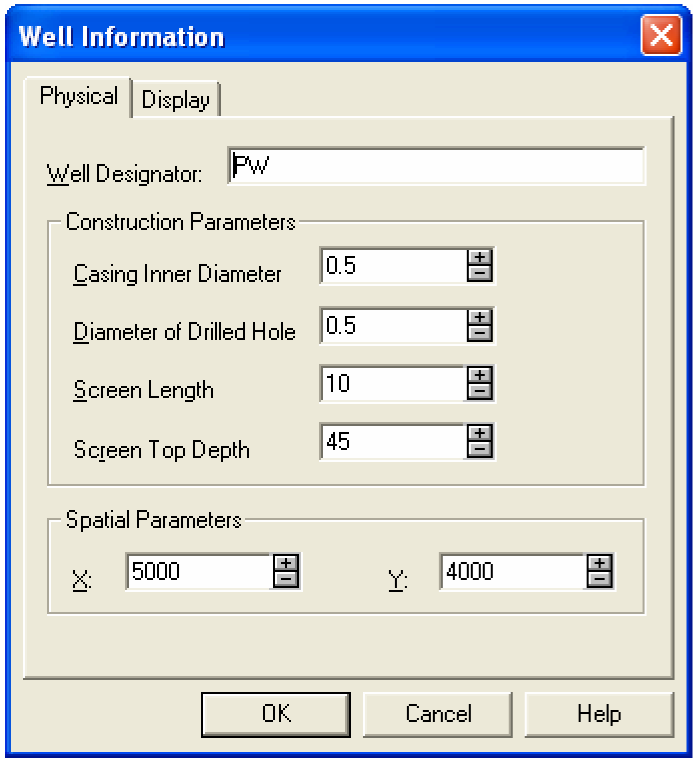
First, use the **File->New** menu and create an "AquiferWin32 Simulation" document. The only visible object on the ***Test Simulator*** view is an arrow. This arrow is the reference head arrow used to define the uniform flow field from which to subtract the drawdown predicted by the analysis. To edit the parameters click the **Edit->Reference Head** menu. We are not going to change any of the values because we are interested in a contour map of drawdown. Given this fact, click the **View->Reference Head** menu to hide the reference head arrow.

As with other tutorials, use the **Edit->Units** menu to set up the units to be used for the simulation. Edit the values as below.



Instead of using the **Add->Well** menu to add wells to the map, click on the **Edit->Map Items** menu to display the *Map Item Information* property sheet. Set the *Item*

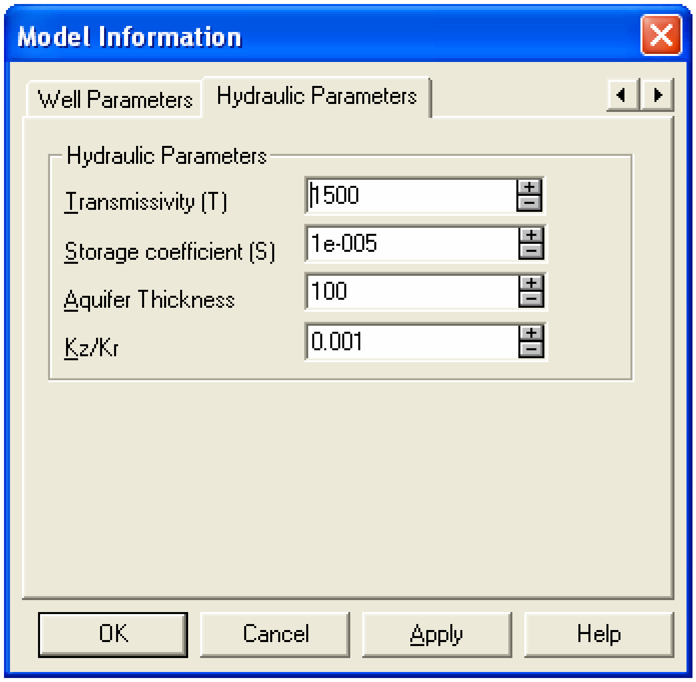
Type to “Well Object” and click the **Add** button. The *Well Information* property sheet is displayed to set the pertinent parameters for the well. Edit the *Well Information* property sheet as below.



Accept the property sheet by clicking the **OK** button and add two additional wells.

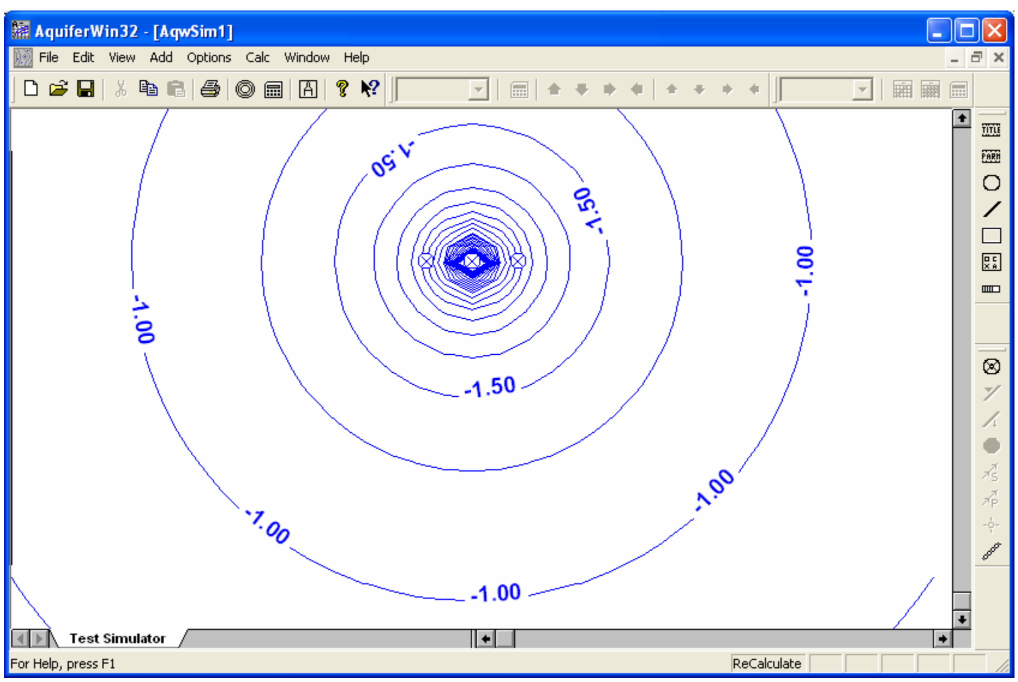
Well	Screen Length	Screen Top Depth	X	Y
MW1	3	45	5500	4000
MW2	3	97	4500	4000

Assume that the aquifer thickness is 100 feet, we have screened the pumping well in the center and one monitoring well at the top and one at the bottom. Accept the *Map Item Information* property sheet and the wells will appear on the map. Next, click on the **Edit->Model** menu and select “Hantush, 1961 (Confined Partial Penetration)” in the *Analysis* combobox. Select “PW” in the *Pumping Well* combobox and enter “2880” in the *Contour Time* field. Click on the ***Well Parameters*** tab and set the *Screen Length* and *Screen Top Depth* to be “10” and “45” for both the *Pumping Well* and *Monitoring Well*. Set the *Pumping Rate* to “25”. Click on the ***Hydraulic Parameters*** tab and set the values as below.



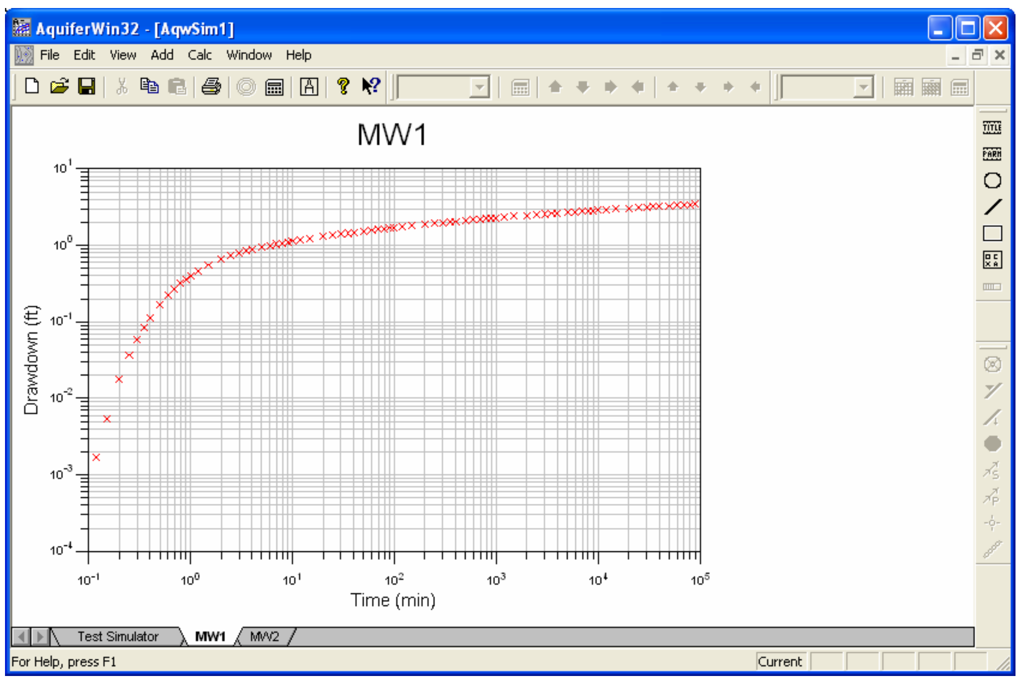
Click on the **OK** button to accept the values and click the **Calc->Recalculate** menu to calculate the simulation. The resulting contour map represents a projection of the drawdown expected in monitoring wells screened between 45 and 55 feet in response to a pumping well similarly screened pumping at 15 gpm for 2 days.

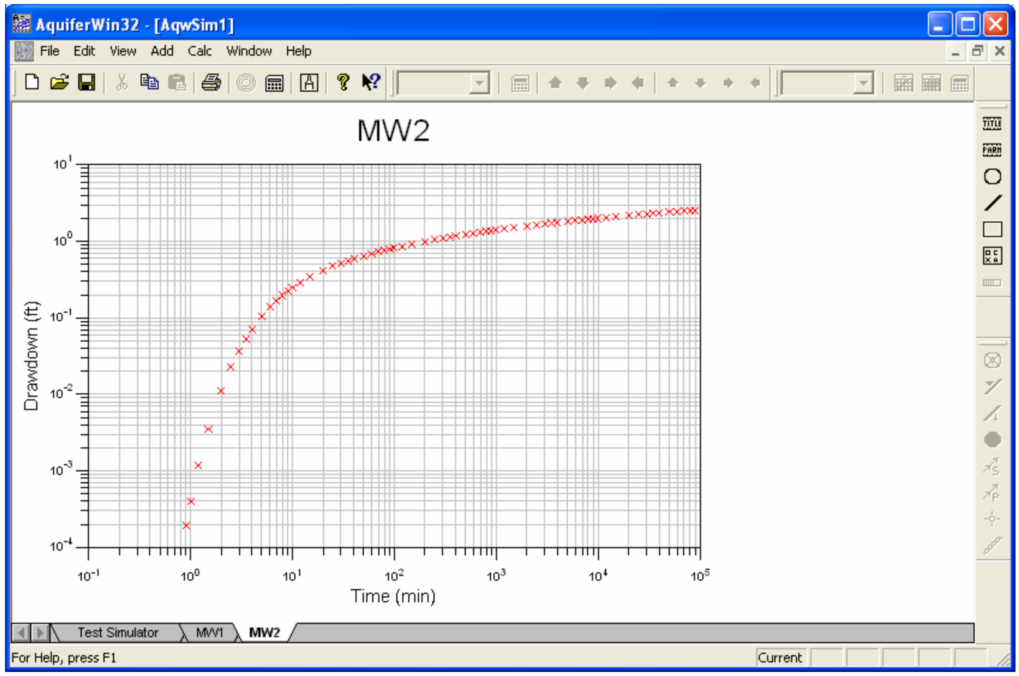
Next we will adjust the contour parameters. Click on the **Options->Contour->Parameters**, click the ***Levels*** tab change the *Minimum Level* to "-8.5" and the *Interval* to ".25". Click on the ***Labels*** tab and change *Distance between* to "2000". Click on the **Font** button and change the point size to "14" and change the *Font Style* to "Bold". Click on the **OK** button and click on the **Yes** button in response to the subsequent message box. Click the **View->Full->Screen** menu to adjust the map scale. The resulting window is shown below.



To generate drawdown versus time data for the two monitoring wells previously added to the map, click on the **Edit->Simulation** menu, click on the ***Wells*** tab and click on the **Add** button twice to move "MW1" and "MW2" from the *Available Wells* list box to the *Monitored Wells* list box. Click on the ***Well Data*** tab and notice that the spreadsheet has three tabs along the bottom. One tab for each monitored well and the ***Simulation Times*** tab. The values contained in the ***Simulation Times*** tab are used for all monitored wells. Click on the **OK** button to accept the property sheet and click the **Calc->Recalculate** menu to calculate the solution. To view the graphic results, click the **View->Well Data** menu to add two additional tabs to the view. Click on both of the new tabs to view the results. Click on the **View->Clip Data** menu to clip all data points not contained within the limits of the graph. The difference between the two graphs is caused by the effects of partial penetration.

Now we want to adjust the graph to display the same axis scales. Although we could change them individually, an easier way is to click the **Edit->Default Well Graph** menu. Click the ***X-Axis*** tab and enter ".01" for the *Minimum X* edit field and click on the **Automatic Minimum** check-box to unset it. Click the ***Y-Axis*** tab and click on the **Automatic Minimum** check-box to unset it. Click the **OK** button to accept the property sheet. In order to activate the defaults, click the **Edit->Simulation** menu and go to the ***Wells*** tab. Click the **Remove** button twice to remove the wells. Now add them back to the *Monitored Wells* list using the **Add** button. When we removed them the graph parameters were removed. Adding them, initializes graph parameters using the Default Well Graph parameters. Click the **OK** button to accept the changes. Click the **Calc->Recalculate** menu and the graphs are as presented below.



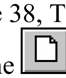


AquiferWin32 - [AqwSim1]
File Edit View Add Calc Window Help
MW2
Drawdown (ft)
Time (min)
Test Simulator
MW1
MW2
For Help, press F1
Current

Water Table Aquifers

What you will learn:

- *Using Neuman's Water Table Analysis Method*

This section outlines how to analyze an unconfined aquifer pump test. The particular data are taken from Lohman (1979; page 38, Table 14). The first step is to create a new analysis document by clicking on the  button on the Standard toolbar. This toolbar button creates a new "AquiferWin32 Analysis" document whereas the **File->New** menu presents the five potential document types.

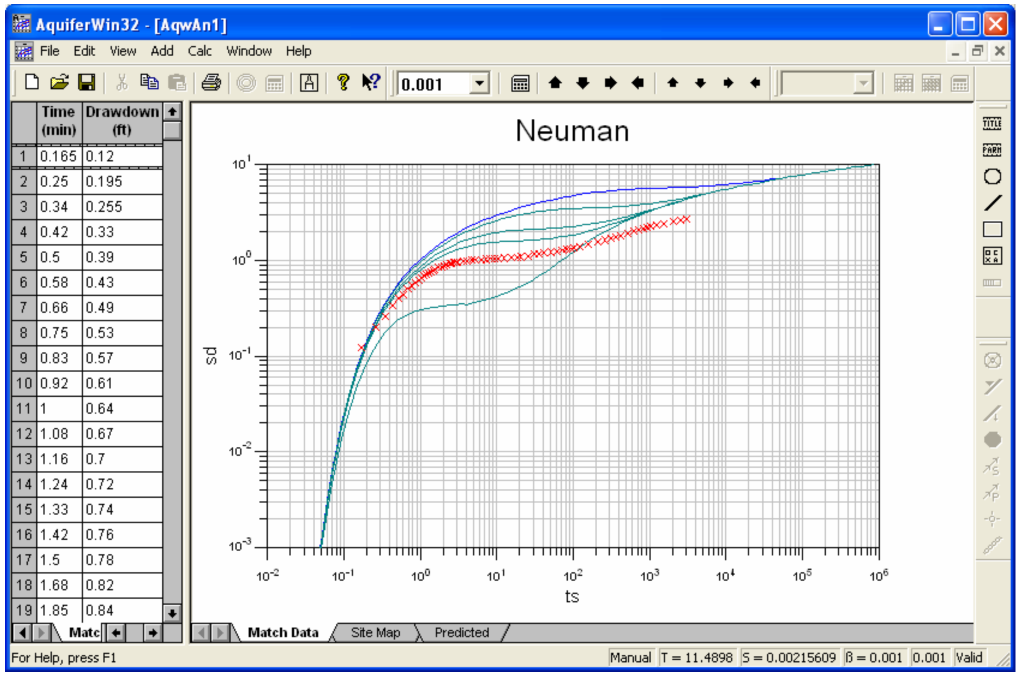
Once the document is created, the first step is to set the units to be used in the simulation. The **Edit->Units** menu displays a property sheet with two tabs. The first tab, ***Summary***, contains the four basic parameters and their respective units. The values represent either the program defaults or the values previously saved as default. Under normal circumstances, you would set the summary units to most closely approximate the units of the data collected. Set *Time Units*, *Length Units*, *Pumping Rate Units* and *Transmissivity Units* to "min", "ft", "gal/min" and "sq ft/min". Click the **Save As Default** check-box and the **Apply Globally** check-box. There is no need to click the **Convert Data** check-box because no data exist yet. Accept the property sheet by clicking the **OK** button.

Click on the **Edit->Aquifer Test** menu. The first tab of the *Aquifer Test Information* property sheet contains user-fields. These user-fields can be used to annotate any graph or map view and are also included in the hard copy report. Click on the ***Pumping*** tab and enter "PW" as the *Well Name* in the *Pumping Well* section and enter "1080" into the *Pumping rate* field. Enter "MW" as the *Well Name* in the *Monitoring Well* section and enter "73" in the *Radial distance* field. The remainder of the well construction information is not required since this analysis will not consider partial penetration.

To enter the well response data for the monitoring well, click on the ***Well Data*** tab. This spreadsheet has one tab along the bottom for the match data. Click on the row number column of the first data row to select it and hit the **Del** key to delete the data row. By now you probably have experience entering data into the spreadsheet using the keyboard so we will import the data from an ASCII file. Click the right mouse button within the spreadsheet to activate the context menu and select **Import** to import the data from the file *unconf.dat*.

After we accept the *Aquifer Test Information* property sheet by clicking the **OK** button, we are ready to select the analysis to use. Click the left mouse button in the graph view to activate its menu. Select the **Calc->Reset Data Offset** menu which is good practice when changing units without converting the data. Click the **Edit->Solution** menu, click on the *Analysis* combobox and select "Neuman, 1972 (Unconfined Aquifer)". Click on the ***Parameters*** tab and notice the *Radial distance* and *Pumping rate* have been updated based on our previous entries in the *Aquifer Test Information* property sheet. Set the *Specific Yield* to be two orders of magnitude higher than the value of *Storage coefficient*. Right click the mouse on the *Specific Yield* data field and select the **Sync->Calculated** menu Strictly speaking, this step is not necessary but will generate a better set of type curves. (**NOTE: The ratio of Storage coefficient and Specific Yield dictate the offset between the "Early" and "Late" portions of the type curves)**. Click on the ***Curves*** tab and set the contents

of the *Curve Information* list box to "0.001", "0.01", "0.05", "0.10" and "1.00". Click on the **OK** button and the screen should appear as below.

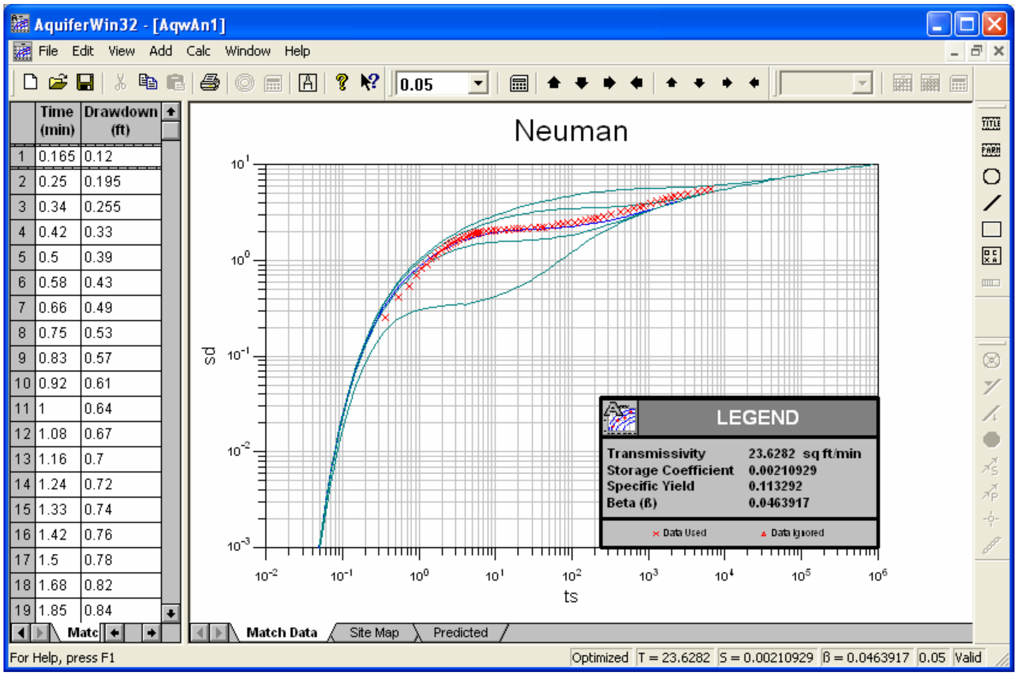


Click on the **Calc** menu and make sure there is a check mark next to the **Match Early Data** menu. If it is not, click on the **Calc->Match Early Data** menu. Move the data around using the arrow keys to perform a manual match of the early (left most) data points to the left side of the type curves. Use **Edit->Toggle Type Curve** or **Ctrl-T** to select the appropriate type curve. Look at the status bar and note the values for T and S which are representative of matching the early data (the initial confined response). Although matches are subjective, match to a T of approximately 20.3 and an S of approximately .0025 using the .05 type curve. Next, click the **Calc->Match Late Data** menu to match the late data. Notice that the data moves and the status bar T and S change. This is because both the Early and Late Matches are kept independently as if they were two separate monitoring wells. Repeat the procedure for the late (right most) data to determine T and S for the late data (the unconfined response). This time, match to a T of approximately 23.6 and an S of approximately 0.111 using the .05 type curve. As you toggle between the early and late solutions, you can see the difference in the value of T.

Now that you have made your two matches as prescribed by the analysis you have finished the traditional solution. To improve the presentation of the results, you can click **Calc->Recalculate Type Curves** to calculate a set of type curves consistent with the new *Storage Coefficient* and *Specific Yield* ratio. By recalculating the type curves, you will have to reset the selected type curve. This step gives you better type curves than the traditional ones in the literature because they have been calculated specifically to your solutions. Use **Ctrl-T** to toggle the type curve selection back to the .05 type curve.

If you want to get an optimized solution and eliminate the need for two separate matches, click **Edit->Solution**, click the ***Solution*** tab and click the **Advanced** button and click off the **Calculate Initial Estimates** check-box. The initial estimates calculation is not necessary because we performed a manual match. In some cases, this being one, the initial estimates calculation can cause the solution to become unstable. Click the **OK** button to close the property sheet and click the **Calc->Optimize** menu. To see the results of the optimization, click on the **Edit-**

>Solution menu and go to the ***Results*** tab. If you would like the exact type value of .0468 instead of .05, click on the ***Curve*** tab and change *Value* from .05 to .0468 and set *Precision* to "3". Since a curve value was changed, the type curves will automatically be recalculated. Add a legend by clicking the **Add->Legend** menu, dragging a rectangle on the screen and clicking the **Finish** button on the *Legend Wizard*. The optimized view is as below.



Step Test Analysis

What you will learn:

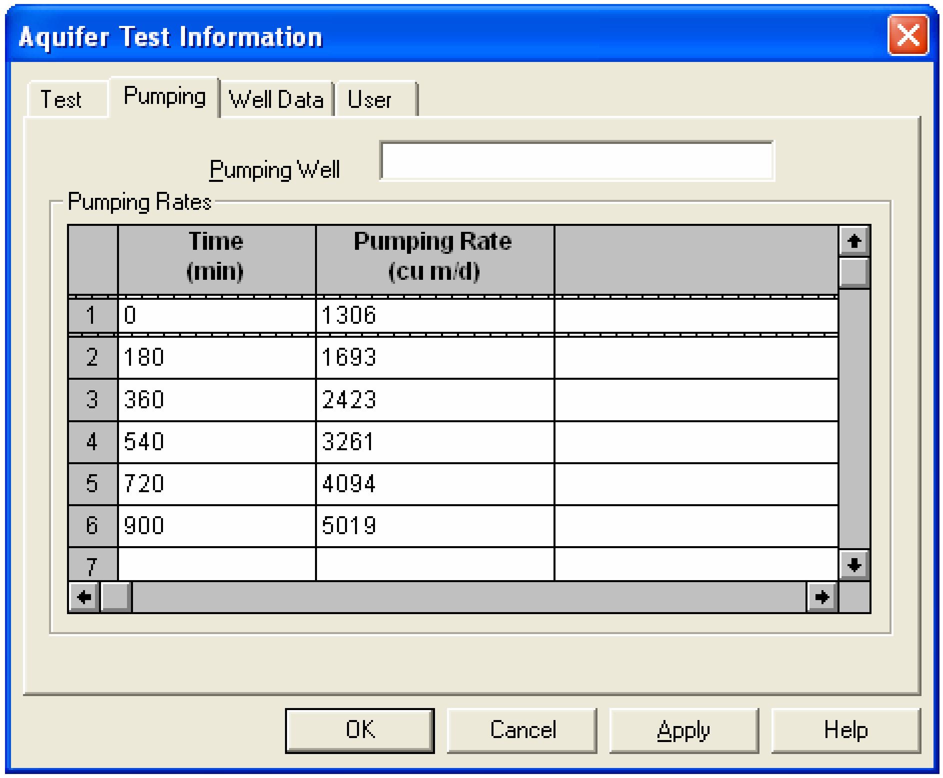
- *Using the Eden and Hazel method to determine well performance*
- *Automatic and manual matching procedures*

For this particular test, users are guided through the input and analysis of a data set taken from Kruseman and De Ridder (1994), page 204. These data are also used in an example file (*clarke4.aqp*) which can be used to check the analysis results. The data was taken from Clarke L. (1977), *The analysis and planning of step-drawdown tests.* Q. Jl. Eng. Geol, volume 10.

The first step is to set up a new file in which to input the step test data. Selecting the **File**->**New** menu displays the five types of documents or analysis types, including "AquiferWin32 Flow Model", "AquiferWin32 Analysis", "AquiferWin32 Simulation", "AquiferWin32 SlugTest" and "AquiferWin32" StepTest. Select "AquiferWin32 StepTest" and click the **OK** button. This action will bring up the default Eden and Hazel (Part 1) screen.

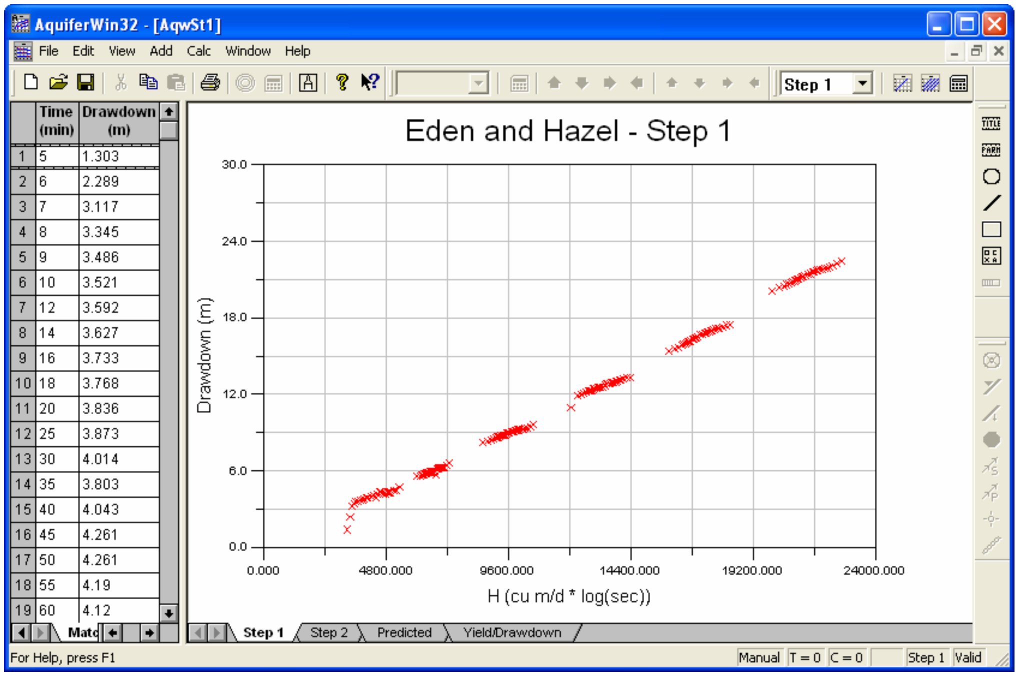
The next step is the specification of the units to be used in the input and analysis. It is recommended that these be specified at this stage, although as you know, the units can be modified after input. For this example, set *Time Units*, *Length Units*, *Pumping Rate Units* and *Transmissivity Units* to "min", "m", "cu m/d" and "sq m/d". Click the **Save As Default** check-box and the **Apply Globally** check-box. There is no need to click the **Convert Data** check-box because no data exist yet. Accept the property sheet by clicking the **OK** button.

Once the step test has been selected and the units of input chosen, the data input process can begin. You should be familiar by this stage with the options for inputting data and be well versed in all of the methods. Since this is a large data file, it is recommended that the file *clarkedd.dat* be imported as an ASCII file using the **File->Import** menu option. This file contains only the drawdown/time data and when imported the file should contain 174 lines. It is again important to remove the initial default value from the first row of the time/drawdown column. Next enter the pumping rate and time data. This is entered by selecting the **Edit->Aquifer Test** menu, clicking the ***Pumping*** tab, right click the mouse on the spreadsheet and select the **Insert** menu to create a row of data; now, enter the data on the table as shown below.

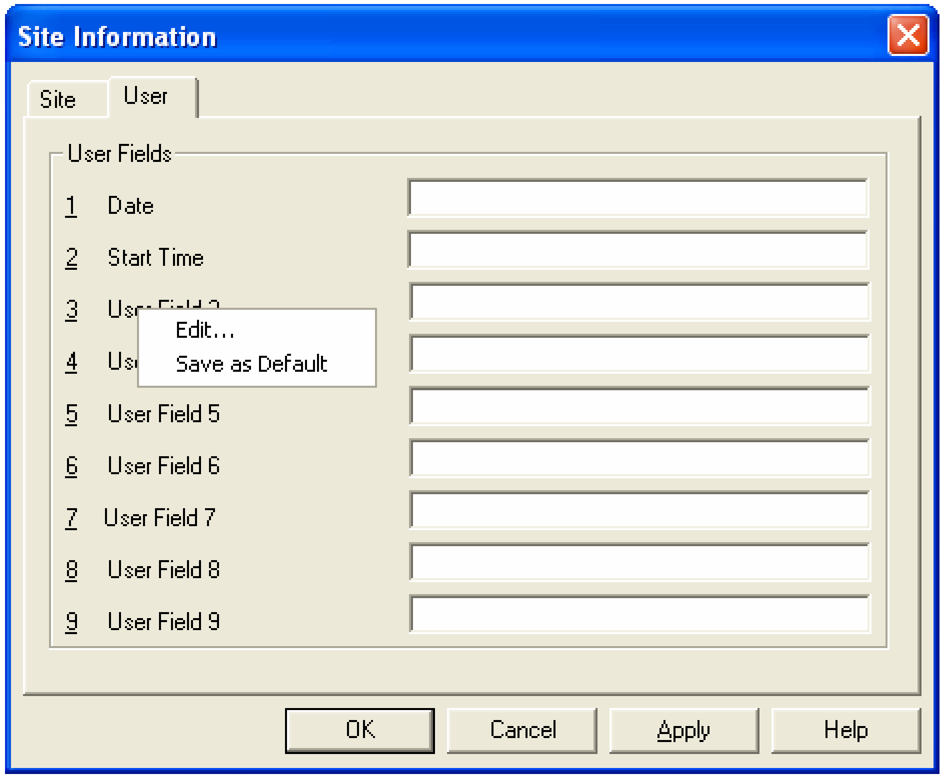


Accept the property sheet by clicking the **OK** button. Normally, the graph axes will automatically rescale to match the data; however, if you now look at the graph, the data may appear to be missing. This is not due to its abduction but to the fact that the axes will have to be rescaled to allow the display of this data. In this particular case the maximum on these linear scaled axes should be set such that the maximum value for x is 24000 and the maximum value for y is 30. Click in the graph view and activate the **Edit->Graph** menu. Click the ***X-Axis*** tab and enter “24000” in the *Maximum X* field and click the **Automatic Maximum** check-box to unset it. Click the ***Y-Axis*** tab and enter “30” in the *Maximum Y* field and click the **Automatic Maximum** check-box to unset it. When new data sets are to be analyzed the drawdown or y-axis can be scaled according to the maximum value of drawdown encountered. In the case of the x-axis it is suggested that the initial scale be set to a high value to display the whole data set and that it is scaled down to increase resolution of the data at an appropriate lower value of H as desired.

At this point it is worth mentioning data weighting. Some of the early data in the data sets will not be very useful in the analysis. If it is desired that some of this data be excluded from the analysis, go to the **Edit->Aquifer Test** menu, ***Well Data*** tab which can be used to change the weighting of some data points. If the weight on a point is set to 100 or greater, it is then after excluded from the analysis. A change in the symbol type can be used in conjunction with this function to allow the user to distinguish excluded data on the graph. The resulting graph is shown below.

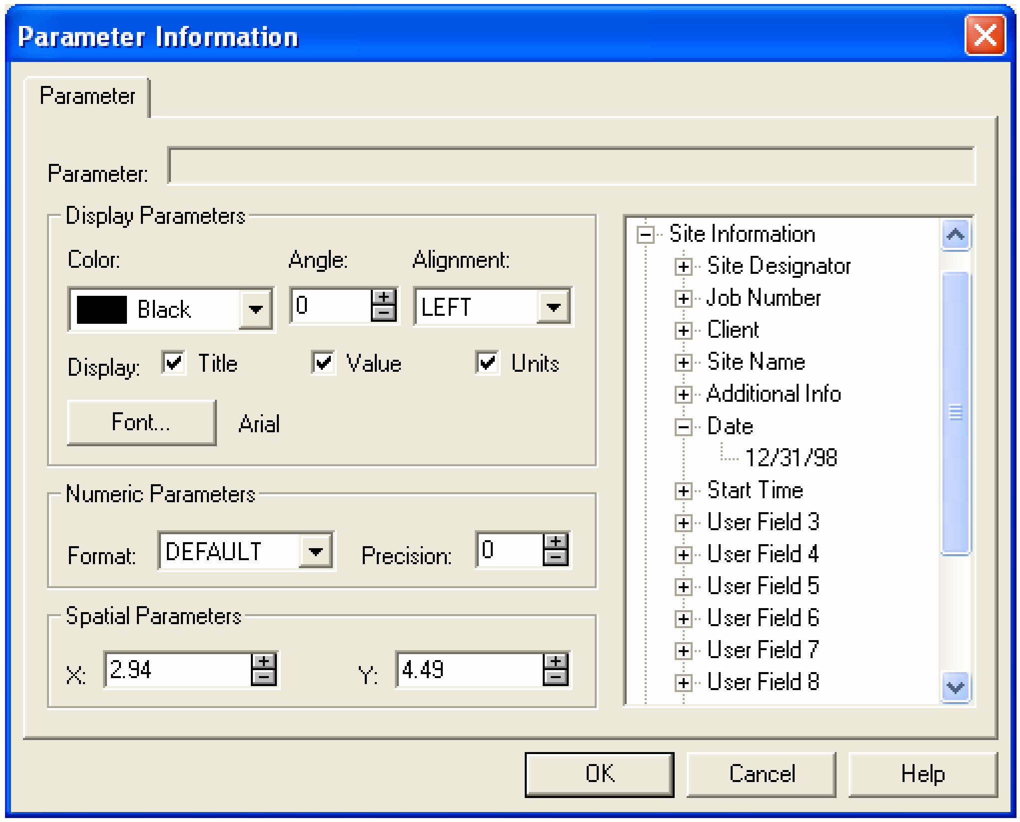


Now that these initial data have been entered the test is ready to run. However it is worth noting that there are a number of extra inputs which are specified in the property sheet. The **Edit->Site**, **Edit->Aquifer Test** and **Edit->Analysis** menus activate property sheets that can be used for the input of details which can be later displayed in the form of title boxes, headers, footers, parameter boxes etc. The details can be entered into the defined boxes or one of the **"**User Fields**"** in the ***User*** tab of each property sheet. User fields are customizable parameter fields which allow input of a wide range of preferred data or text items. Click on the **Edit->Site** menu to bring up the *Site Information* property sheet. If you click on the ***User*** tab, you will see that each field has a default name. This default name can be changed easily by following the following instructions. Right click the mouse on the title for *User Field 1* to activate the context menu and select the **Edit** menu. Enter "Date" into the *User Field Title* field and click **OK** to accept the dialog. Repeat the procedure to set *User Field 2* to "Start Time". If you activate the context menu for *User Field 3* the property sheet would resemble the one below.

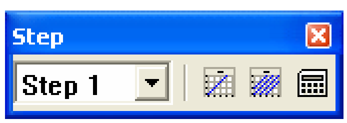


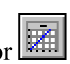
Enter “12/31/98” into the newly identified *Date* field. If you also activate the **Save as Default** menu for any of the fields, the system registry will be updated and future Aquifer^{Win32} sessions will use the value.

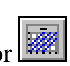
In addition to customizing the property sheets, this information can then be accessed in the parameter objects to enhance graphical output. Accept the property sheet by clicking the **OK** button. Click the **Add->Parameter** menu and click the left mouse button within the graph. Drilling down into the tree list you will see the newly created fields as below.



There are several ways to analyze the data and obtain a solution. These are briefly described in turn below. The Step toolbar below shows the buttons to be used for the step analyses. The buttons from left to right are: a combobox for choosing the active step which is used in conjunction with both the manual match and the single regression analysis; the button for performing single linear regression based on the active step; the multiple regression button; and the optimization button. There are analogous menus on the **Calc** menu.



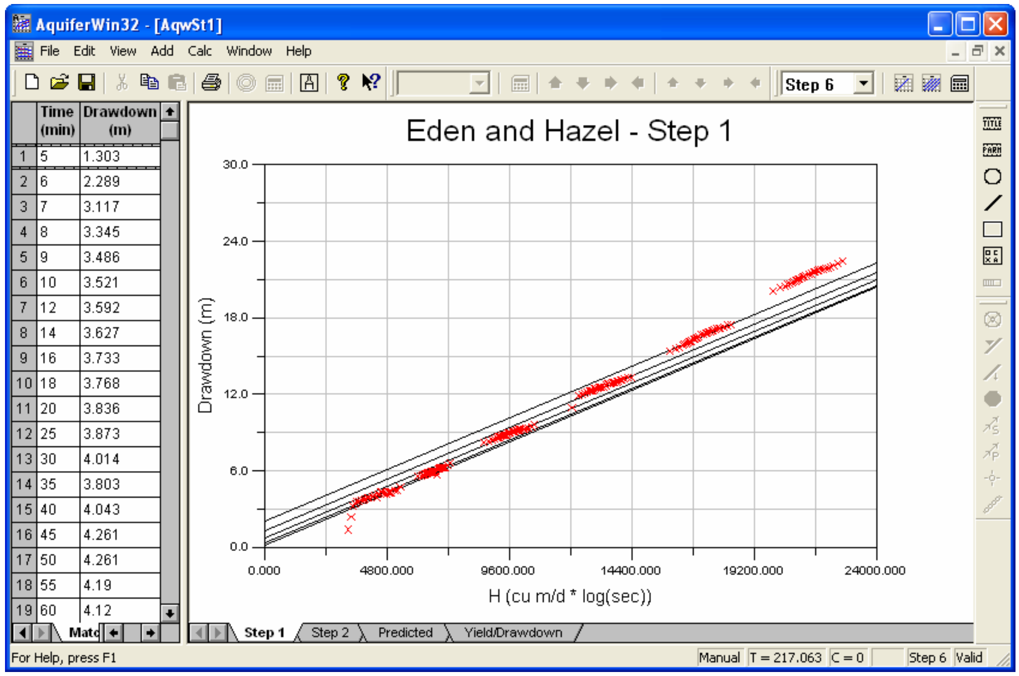
The **Calc->Linear Regression–>Single Step** menu or  toolbar button calculates a linear regression line based on the active step. In this case, the data for the first step or pumping period is used. Since this analysis requires that the slope of all regression lines for all steps be equal, the remaining steps are calculated by specifying the slope and then finding the intercept of the best fit line. The step is chosen either by using the toolbar combobox shown above, or using the **Edit->Toggle Step** menu or **Ctrl+D** accelerator key. Analysis Step 2 is required to complete the analysis.

The **Calc->Linear Regression->Multiple Step** menu or  toolbar button performs a linear regression on all steps combined to find the best slope across all steps and the individual intercepts for each step. Analysis Step 2 is required to complete the analysis.

The **Calc->Optimize** menu or  toolbar button performs the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to calculate an optimized solution.

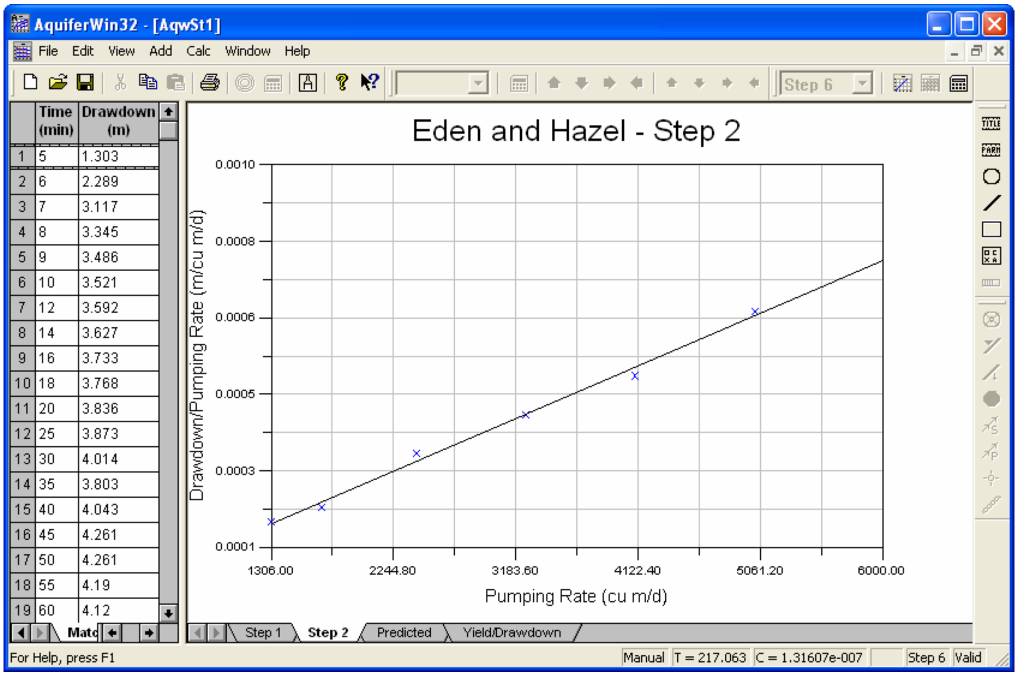
Analysis Step 2 is not required to complete the analysis. Furthermore, parameters can be fixed and/or constrained during the optimization process. This is described in more detail in previous sections.

In this instance, change the active step to Step 6 and will use the **Calc->Linear Regression->Single Step** menu to calculate a resulting in a Transmissivity of 217 sq m/d. The resulting graph is shown below.



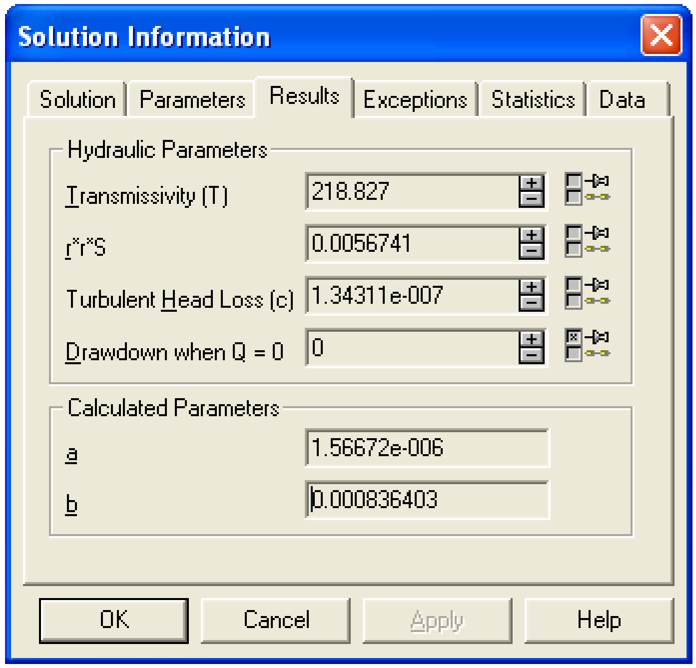
To complete the analysis, click on the ***Step 2*** tab of the graph view. Once again, the data points, which were calculated based on the results of Step 1, may not be visible if autoscalling has been deactivated. Regardless, click the **Edit->Graph** menu, click the ***X-Axis*** tab and change the *Maximum X* to "6000" and click the **Automatic Maximum** check-box to unset it. Click the ***Y-Axis*** tab and change the *Maximum Y* to ".001" and click the **Automatic Maximum** check-box to unset it. Also, change the *Precision* to "4". Finally, click the ***Graph*** tab and change the X-Origin to "1.1" so the axis label is not clipped.

To complete Step 2, click the **Calc->Linear Regression->Single Step** menu or toolbar button. The result of this step is to calculate the Coefficient of Turbulent Head Loss as 1.31607e-007 sq d/m^5. The units of Coefficient of Turbulent Head Loss may be different and can by clicking the Edit->Solution menu, clicking the ***Parameters*** tab, right click the mouse on the edit field, select the **Units** menu, set the values appropriately, click the **Convert Data** check box and click the **OK** button. Whether you use the **Calc->Linear Regression->Single Step** or the **Calc->Linear Regression->Multiple Step** in Step 1, Step 2 remains the same. If you had used the **Calc->Optimize** menu, Step 2 would be done automatically. The resulting graph is shown below.

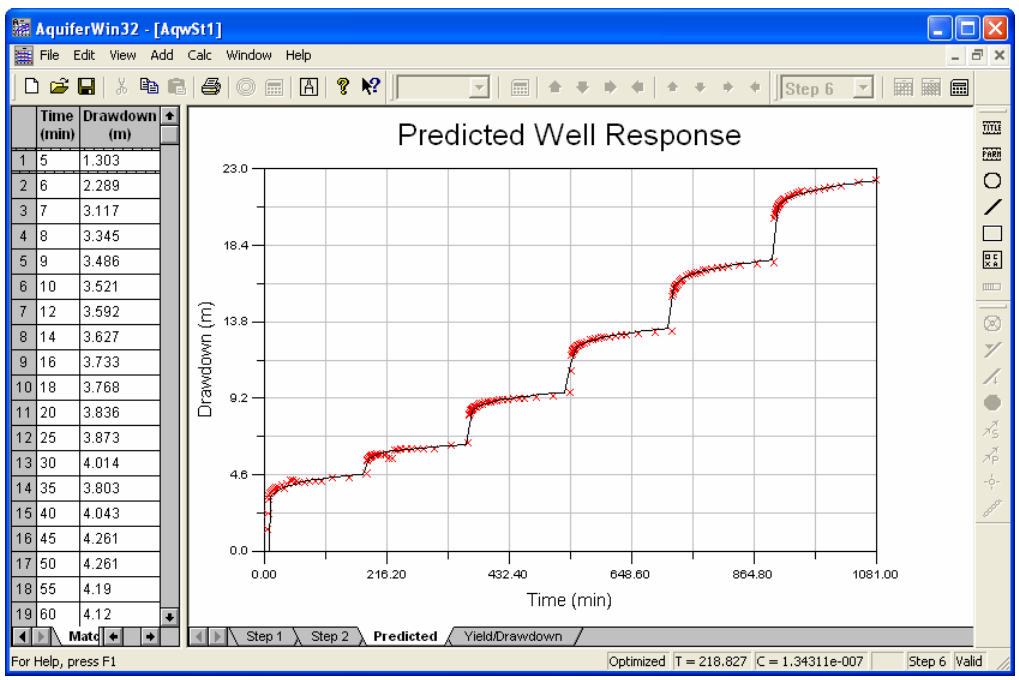


Changes to the solution can be made using manual methods; the lines can be moved in two ways. First the lines can be adjusted as with other linear regression analyses discussed previously. For analysis Step 1, set the active step using the Step toolbar combobox to activate the appropriate regression line. To change the intercept of a line, move the mouse over the line until you get a NSEW (arrows pointing up, down, left and right) cursor, click with the left mouse button and drag the line to a new position. To change the slope of the line, move the mouse cursor over either end of the line until you get the EW (arrows pointing left and right) cursor, click the left mouse button and drag the endpoint to a new position. In analysis Step 1, all regression lines will be updated based on the new slope.

Now, click the **Calc->Optimize** menu to calculate the optimized solution. The values for T and C are 219 sq m/d and 1.34311e-007 sq d/m^5 respectively. Click the **Edit->Solution** menu and the ***Results*** tab to display the property sheet below.

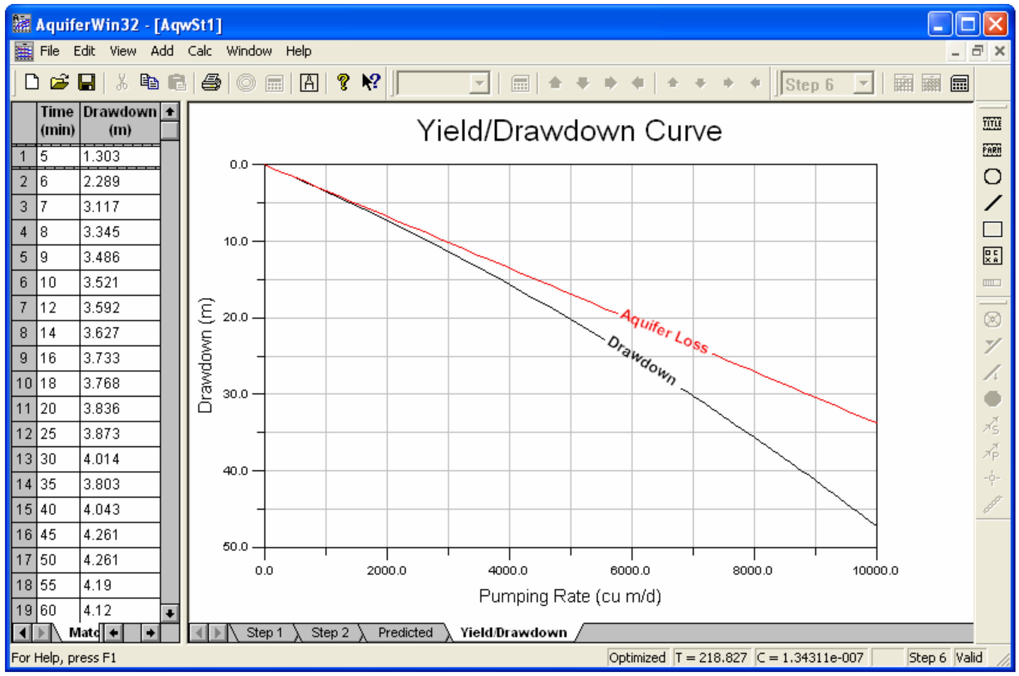


An alternative view of the "goodness of fit" is contained in the **Predicted** view tab. The graph, shown below, was adjusted from the default by changing the axes to linear, changing the labels, etc. It is a useful interpretive aid and allows you to establish the accuracy of the fit of the results by comparing observed and predicted drawdown versus time.



Another graph presented in the ***Yield/Drawdown*** tab predicts the drawdown created by pumping at a given rate and over a specified time. The time is currently taken to be the average of the time of each step, equal to 180 minutes in this example. The drawdown line (shown in black on the graph) of the yield/drawdown curve shows the total drawdown/yield relationship for the given time, while the second line shows the effect of aquifer loss (shown in red on the graph) giving the hypothetical relationship

for a 100% efficient borehole. The curve is useful in evaluating the extent of turbulent losses. The enhanced graph is shown below.



Analytical Flow Model

What you will learn:

- *performing an analytical flow model*
- *plotting simulated drawdown contours*
- *plotting simulated hydrographs at wells*
- *setting up a transient simulation with variable pumping rates*
- *automatically calibrating the model to observed target heads*
- *adding a legend to the contour map with a color flood bar*

The Modeling version of Aquifer^{Win32} can also be used to perform analytical flow models or analyze multiple pumping well aquifer tests. The analytical elements supported by a specific model are specified in the model specific documentation and help file. In general, all models support wells and targets which is what we will use in this tutorial. Additionally, some models support steady-state calculations, some support transient calculations and some support both.

First, use the **File->New** menu and create an "AquiferWin32 Flow Model" document. The only visible object on the view is an arrow. This arrow is the reference head arrow used to define the uniform flow field from which to subtract the drawdown predicted by the analysis. The first thing to do is set up the units by clicking the mouse within the map and then using the **Edit->Units** menu. Select "min" for *Time Units*, "ft" for *Length Units*, "gal/min" for *Pumping Rate Units* and "sq ft/d" for *Transmissivity Units*. Click the **Save As Defaults** and **Apply Globally** check-boxes and accept the property sheet by clicking the **OK** button. It is important to save your selections as the defaults because these values are used whenever new items are created within the document.

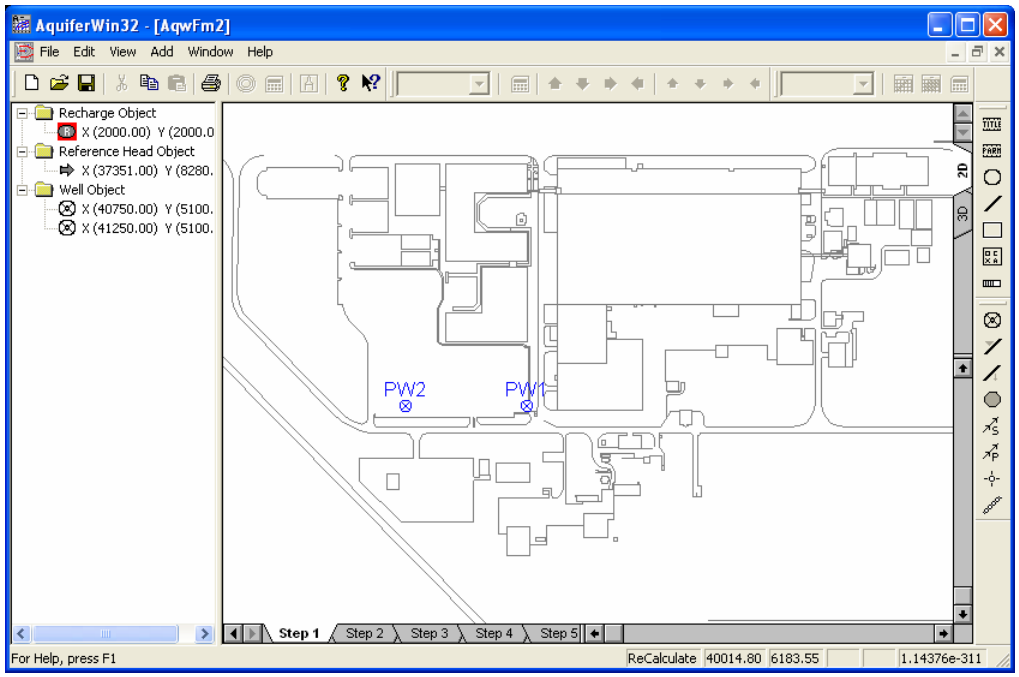
Next, we need to define the map window within which you are going to work. In this case, click the mouse within the map window and use the **File->Map** menu to import the file *example.map*. This file represents a large site; however we are going to focus our attention on a subset of this map. Click the **Options->Map->Window** menu and drag a rectangle on the map. On the *Map Window* dialog that is displayed, enter "40000" for *Origin X*, "4200" for *Origin Y*, "2000" for *Height* and "3000" for *Width*. Accept the dialog by clicking the **OK** button. When asked to adjust the contour window to the new map windows, select the **YES** button.

Now, we need to define the initial heads for the model. To edit the parameters click the **Edit->Reference Head** menu. Set the value of *Head* to be "200", *Gradient* to be ".001", *Angle* to be "135", *X* to be "37351" and *Y* to be "8280". This reference head location is not located on the map window we are using but it serves to calculate the correct initial head distribution for this simulation.

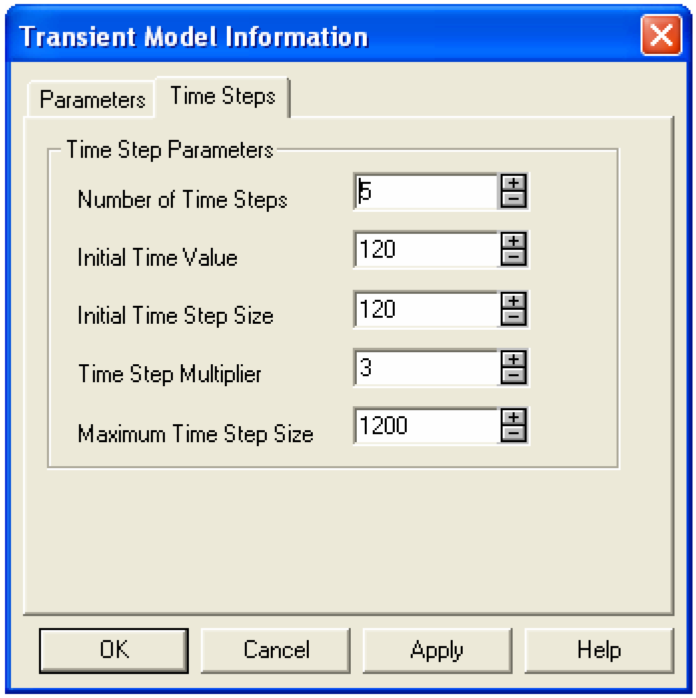
Instead of using the **Add->Well** menu to add wells to the map, click on the **Edit->Analytic Elements** menu to display the *AnalyticElement Information* property sheet. Set the *Item Type* to "Well Object" and click the **Add** button. The *Well Information* property sheet is displayed to set the pertinent parameters for the well. On the *Well Information* property sheet set *Well* to "PW1", *X* to "41250", *Y* to "5100" and click the **Transient Pumping Rate** radio button. Click the **Display** tab and click the **Display Label** check-box. Now click the **Transient** tab and import the

file *pw1gpm.dat* to set the pumping schedule for this well. Accept the property sheet by clicking the **OK** button. Repeat the procedure to enter PW2 with an x-coordinate of 40750, a y-coordinate of 5100 and import the pumping schedule in file *pw2gpm.dat*. Accept the *Analytic Element Information* property sheet by clicking the **OK** button.

Now that we have set up two wells with variable pumping rates, click the **Edit->Flow Model** menu and set the solution to "Theis, 1935 (Confined)". Click the ***Parameters*** tab and set *Transmissivity* to "600", the *Storage Coefficient* to ".005", the *Aquifer Thickness* to "100" and the *Porosity* to ".3". The aquifer thickness and porosity parameters are used during particle tracking but not during calculation of hydraulic head. Accept the property sheet by clicking the **OK** button and the view should appear as below.

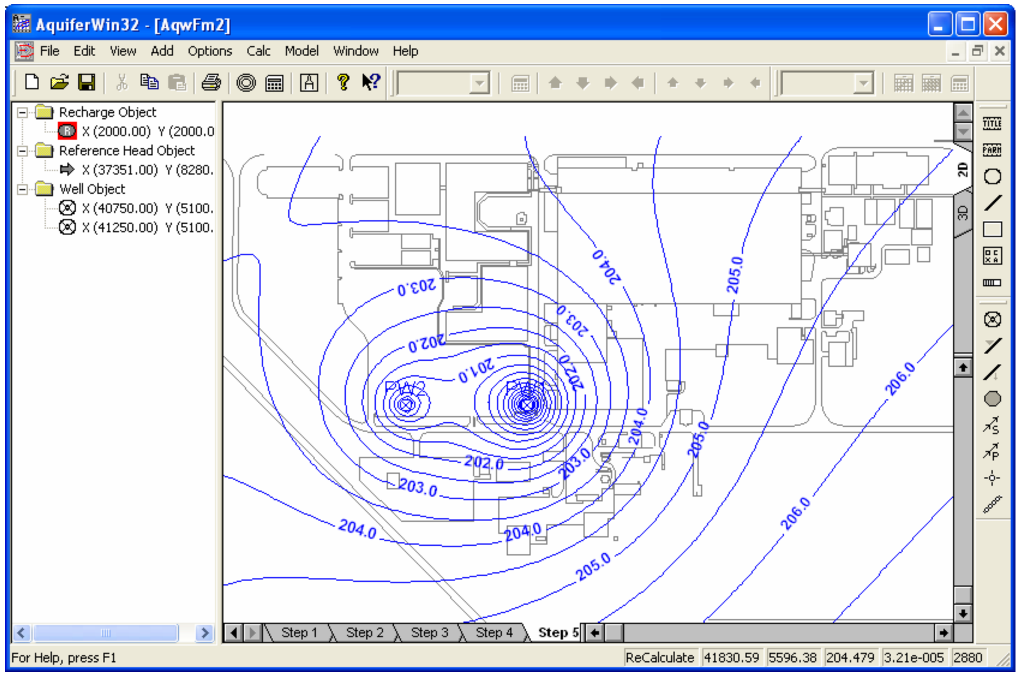


Notice that the view tabs have changed to ***Step 1***, ***Step 2***… This reflects that fact that this solution does not support direct calculation of steady state conditions and the calculation was automatically changed to transient. Click the **Options->Transient** menu to set up the time steps for the simulation. Click the ***Time Steps*** tab and fill it out as below.



Accept the property sheet and you will be asked if you would like to recalculate the model. Click **Yes** to proceed with the recalculation. Click the **Options->Contour->Parameters** menu and set *X Nodes* to "100", *Y Nodes* to "100". Go to the ***Levels*** tab and set *Minimum Level* to "203" and *Interval* to ".5". Now go to the ***Labels*** tab and click on the **Font** button. Set the *Size* to "10" and the *Font Style* to "Bold". Also set the value of *Distance between* to "500" and the *Precision* to "1". Accept the property sheet by clicking **OK**. Click the **Yes** button to recalculate and click the **Yes** button to propagate the changes to all time steps.

The model is now calculated generating 5 view tabs representing the 5 time steps previously defined. The rightmost status bar cell displays the time represented by the contour map in minutes. The contour map for Step 5 is displayed below.



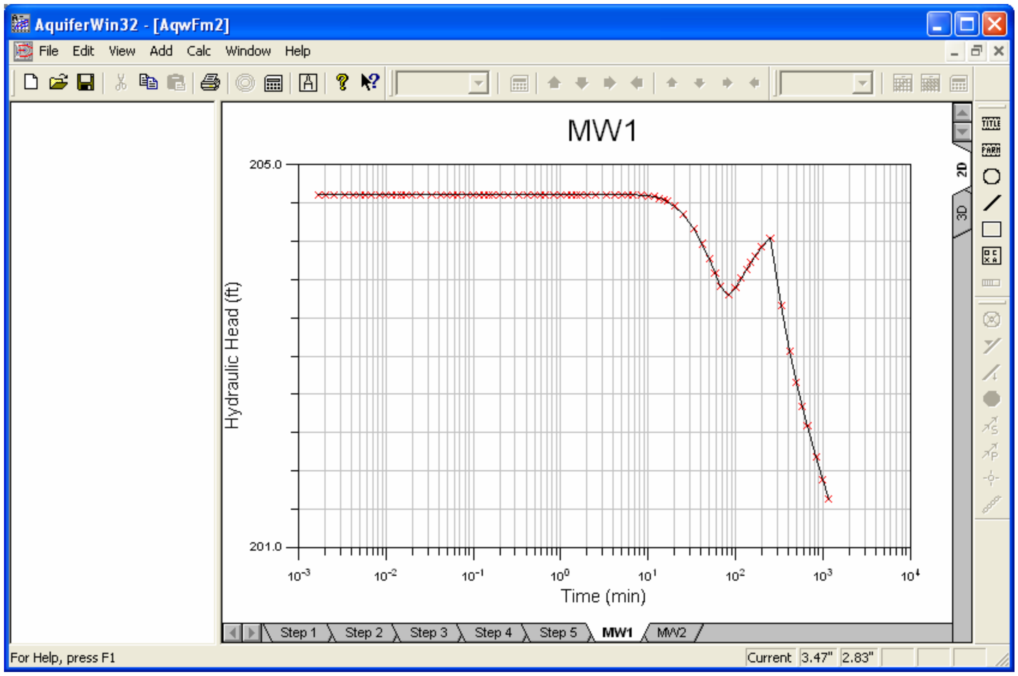
The next step is to calibrate the model to observed heads. Click the **Add->Target** menu and add a target between the two wells. Set the *Target Designator* to “MW1”, *X* to “41000”, *Y* to “5300” and click the **Transient Target Head** radio button. Next, click the ***Transient*** tab and import the file *mw1.dat*. Click the **Add->Target** menu and add another target between the two wells. Set the *Target Designator* to “MW2”, *X* to “41000”, *Y* to “5100” and click the **Transient Target Head** radio button. Next, click the ***Transient*** tab and import the file *mw2.dat*.

Now that we have set head targets to calibrate the model we can check how well our initial estimates of parameters match. Click the **Edit->Simulation** menu, click the **Well Data,** using the context menu activated by the right mouse click on the spreadsheet, use the **Select All** menu followed by the **Delete** menu. Now import the file *simtimes.dat* to import the times at which we want to calculate predicted heads. The default values do not really work in this example because we want increased time resolution in the time periods during which pumping rate changes are occurring. Go to the **Wells** tab, click the **Add** button twice to move MW1 and MW2 from the *Available Wells* list to the *Monitored Wells* list. Accept the property sheet by clicking the **OK** button. No click the **Calc->Recalculate** menu to recalculate the model. To view the graphs for the monitored wells, click the **View->Well Data** menu. Click the left-pointing arrow in the lower left corner of the map view to slide the tabs and reveal the ***MW1*** and ***MW2*** tabs.

Go the **MW1** tab and click the **Edit->Graph** menu, click the **Y-Axis** tab and change the *Axis Type* radio button to **Arithmetic Scale** and the *Axis Numbers Format* to “FIXED”. Also, change the Y-Axis label to “Hydraulic Head” because our model is set to calculate hydraulic head instead of drawdown. Repeat the procedure for the **MW2** tab.

Notice that the observed and predicted heads do not match very well based on the initial estimates of the parameters. To quantify the match, click on a time step tab and use the **Calc->Target Statistics** menu to activate the *Target Statistics Information* property sheet. The residual standard deviation is .5. Now we will perform an optimization to estimate parameter values that best match the target values. Click the **Calc->Recalculate** menu followed by the **Calc->Optimize Model** menu. Now take a look at the **MW1** and **MW2** view tabs to see the visual difference

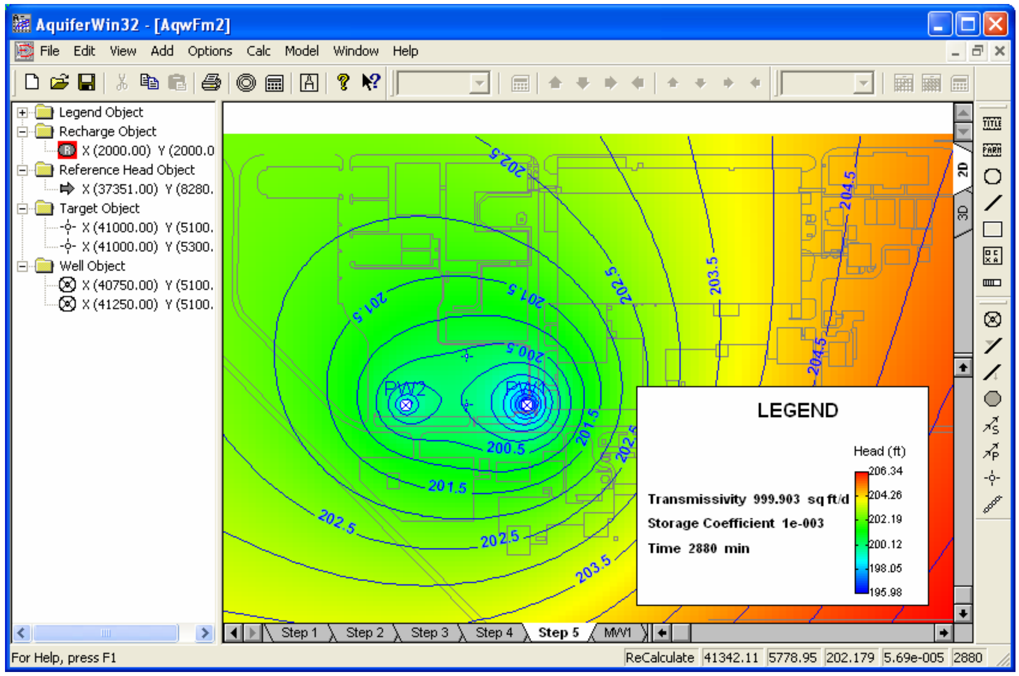
and click the **Calc->Target Statistics** menu to quantify the match. In this case, the standard deviation is .0004 and represents a significantly better calibration.



After autocalibration, the next logical step is to enhance the graphical output. Click on the **Step 5** view tab to display the hydraulic head contour map for time step 5. Click the **Add->Legend** menu and drag a rectangle in the lower right corner of the map. We are creating a custom legend so click the **Create Empty Legend** radio button followed by the **Finish** button. Double click on the legend just created and set *X1* to "41700", *Y1* to "4275", *X2* to "42900" and *Y2* to "5175". Click the ***Contents*** tab and set *Type* to "Solid" then change to the ***Items*** tab. Set *Item Type* to "Legend Object" and click the **Add** button. Set *X1* to "900", *Y1* to "50", *X2* to "950" and *Y2* to "550". Click the ***Items*** tab and set the **Automatic Maintenance of Color Flood Legend** check-box. Basically, we are reserving space within the legend to display a labeled color bar for a color flood. Accept the property sheet by clicking the **OK** button. Set the *Item Type* to "Title Object" and click the **Add** button. Set *X* to "900", *Y* to "600" and *Title* to "Head (ft)". Click the **Font** button and set the *Size* to "8". Accept the dialog and property sheet by clicking the respective **OK** buttons. Add another title with an *X* of "500", *Y* of "750", *Title* of "LEGEND", and set the font to 12 point bold. Change the *Item Type* to "Parameter" and click the **Add** button. Set *X* to "50", *Y* to "400" and set the list control to "Transmissivity" under "Model Parameters". Click the **Font** button and set the *Size* to "8" and the *Font style* to "Bold". Accept the *Parameter Information* property sheet by clicking the **OK** button.

Add another parameter and set *X* to "50", *Y* to "300" and set the list control to "Storage Coefficient" under "Model Parameters". Click the **Units** check-box to unset it. Set the *Format* to "EXPONENT". Click the **Font** button and set the *Size* to "8" and the *Font style* to "Bold". Accept the *Parameter Information* property sheet by clicking the **OK** button. Add a final parameter and set *X* to "50", *Y* to "200" and set the list control to "Time" under "Contour Parameters". Click the **Font** button and set the *Size* to "8" and the *Font style* to "Bold". Accept the *Parameter Information* property sheet by clicking the **OK** button. Now accept the *Legend Information* property sheet by clicking the **OK** button.

Now that a legend is displayed, click the **View->Display Color Flood** menu and the resulting contour map is displayed below.



WinFlow Solver

Getting Started

The WinFlow tutorial 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 tutorial provides several snap-shots of the WinFlow screen to show you what your screen should look like. In addition, the final tutorial data files are provided so that you may skip most of the following steps but still get a feel for how WinFlow works.

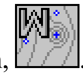
The tutorial tells you to select menu items and a short-hand method is used to indicate what menu items are to be selected. Thus, the menu items are listed in order of selection and are separated by a "->" symbol. For example, instead of writing the following text:

Select **File** from the main menu and then select **Open** from the pull-down menu. The following shortened version is adopted in this manual:

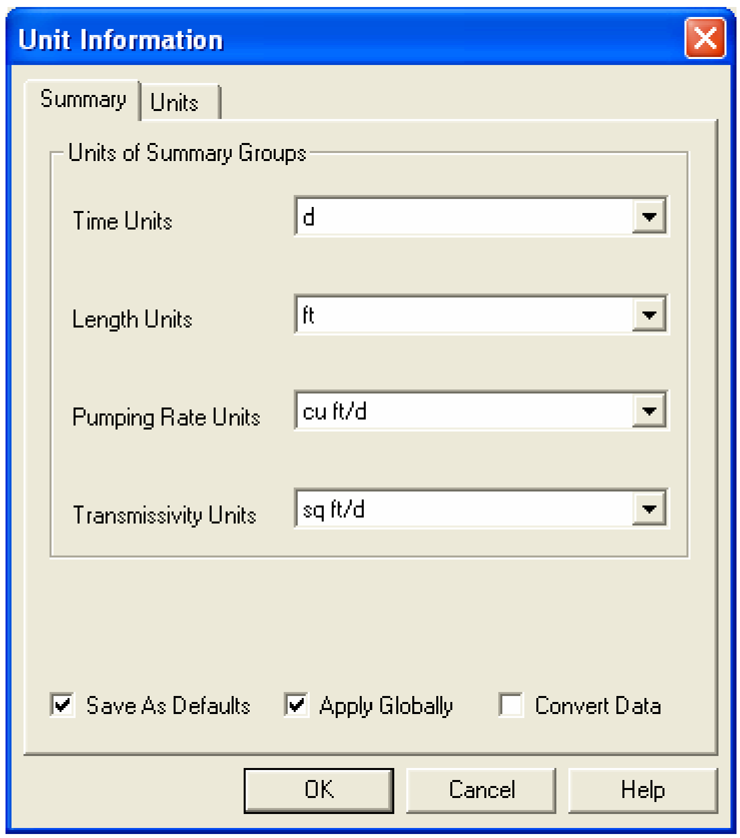
Select **File->Open**

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 succesion. 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

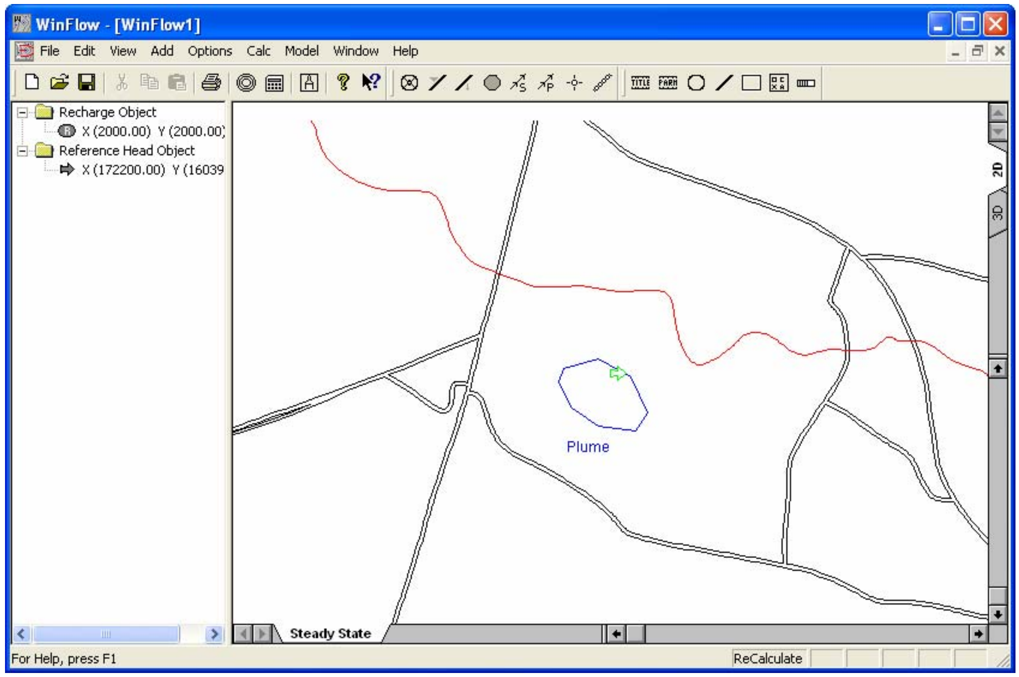
Start the WinFlow program by double-clicking on its icon, . You should see a simple menu displayed above a blank window. A new model is started by selecting **File->New** or by pressing the  button on the Standard toolbar below the menu. Now you can start to design the model. Note that the model view represents two windows separated by a vertical bar. This is refered to as a split window. The menus in WinFlow are context sensitive in that each view has different menu items. Click anywhere within the right view, the map view, to activate its menus.

The first step is to identify the units you wish to use. Click the **Edit->Units** menu to activate the *Unit Information* property sheet and set the values as below:

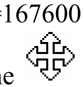


We are using a consistent set of units as you would have done in WinFlow 1.0; however, you can use whatever units you wish. We have selected the **Save As Defaults** checkbox so that these units are used by default. If these are the units you always wish to use, you will not have to edit them again because future models will automatically use them. We have also selected the **Apply Globally** checkbox so that all values throughout the entire model are changed. Finally, we have not selected the **Convert Data** checkbox because no real data has been entered yet. In the event you wanted to change the units on an existing model, selecting this checkbox would automatically convert all the parameters to the new set of units.

The next 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 tutorial. You will import the map called PLUME1.MAP. Select **File->Map** and choose PLUME1.MAP in the standard Windows file dialog. Your screen should now look like the screen shown below.

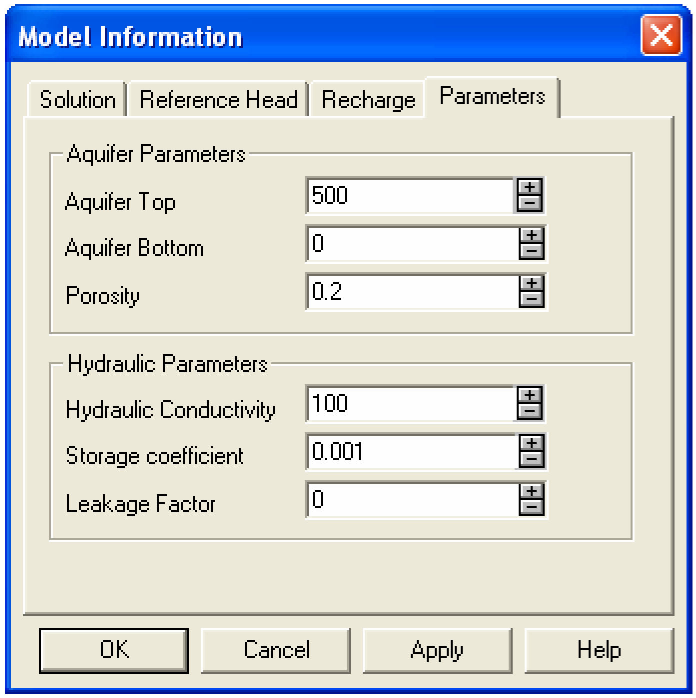


Notice the green arrow in the center of the screen, which denotes the location of the "reference head". The reference head is actually located at the center of the base of the arrow. The reference head is a point in the aquifer where the head is fixed during the simulation and cannot change except in transient simulations. 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.

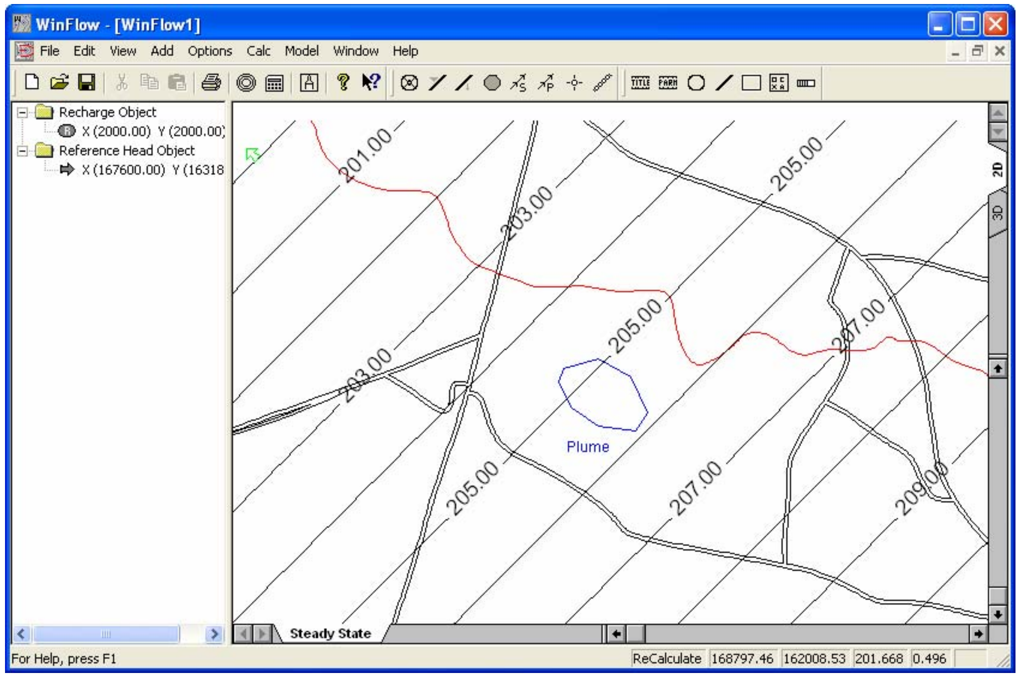
When you release the mouse button you can double click and will see a property sheet that lists data corresponding to the reference head. Check the coordinates listed in the dialog and make sure the coordinates are close to the ones given above. Next, change the hydraulic head to 200, the hydraulic gradient 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->Model** to activate the *Model Information* property sheet. The default solution uses the WinFlow solver and the convergence information parameters are all disabled; this is because the WinFlow solver does not support auto-calibration. Click on the **Parameters** tab and change the values as below:



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.). Once you have made the changes, click the **OK** button.

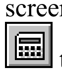
You have now set up the most important parts of the WinFlow model! To see the results, simply select **Calc->Recalculate** or click the  button. You should see a series of contours that are straight lines, representing the uniform regional hydraulic gradient. You will also note that the contour interval is not a round number. 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**. Click on the **Levels** tab, change the minimum contour level to 199, the contour interval to 1.0 ft, and set the color to Black. Click on the ***Labels*** tab and set the color to Black. 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 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 down. WinFlow automatically attaches the file extension ".wfl" to the file name.

Adding Features

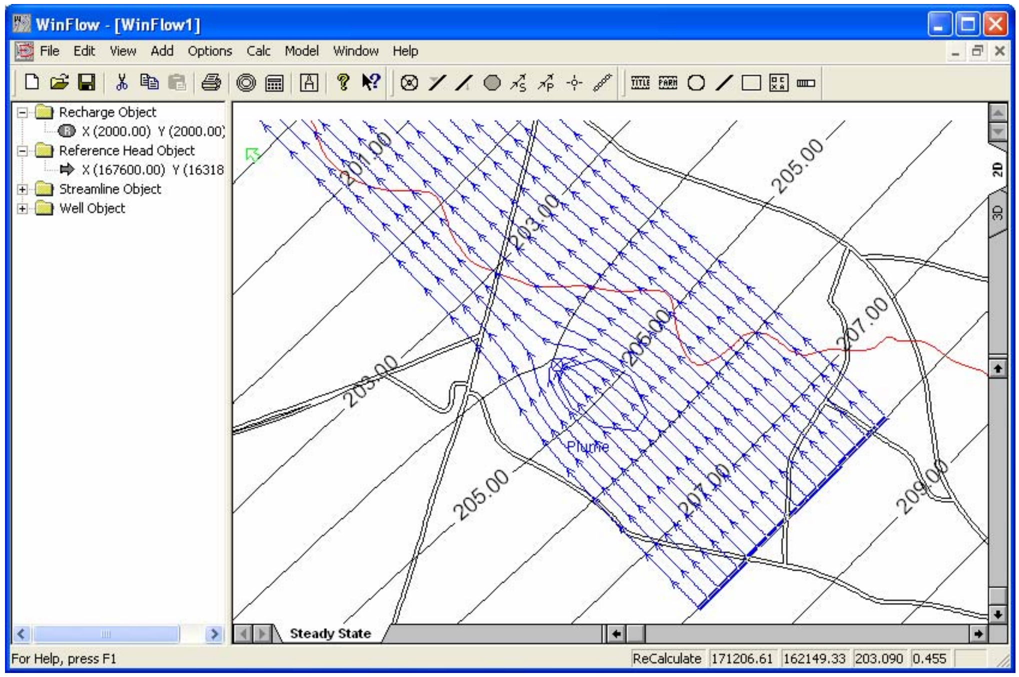
Most models are more complex that the one you just created. Usually, you would like to analyze the effects of pumping wells or other features. In the next section, you will add a pumping well to the model and evaluate the capture zone of the well by adding streamlines. You will also learn how to change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

You should see a blue 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 property sheet for the well will be displayed; enter a name of PW1 and 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->Streamline->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, the *Line Cluster Information* property sheet 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->Trace** and click on the **Display Arrows** checkbox. Next,

click on the ***Streamline*** tab and change the distance between arrows to 500 ft and the arrow size to 100.

Your screen should look like the one shown below.

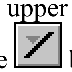


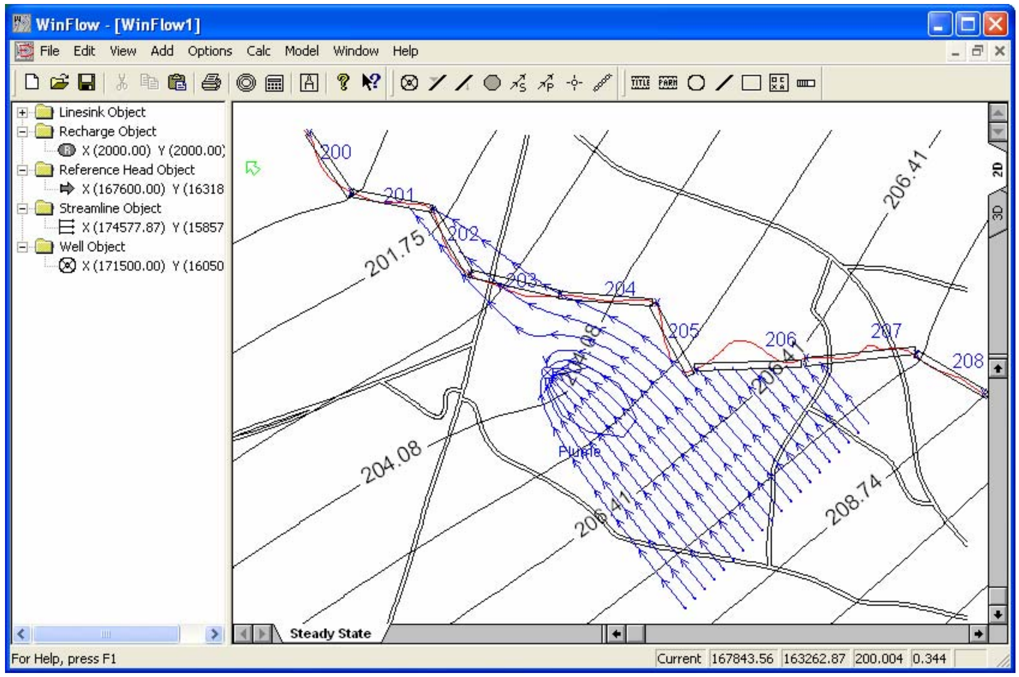
Note that the capture zone of the pumping well does not quite cover the entire plume area. You can increase the amount of pumping from the well by moving the cursor over the well until you see the  cursor. Now, double-click on the well and the *Well Information* property sheet appears. Change the pumping rate to 50,000 ft3/d and recalculate the model. The capture zone should now cover the entire plume.

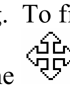
This is a good time to save the current model. Select **File->SaveAs** and enter the name EX2. Click the **OK** button to save the file.

Now let's examine other aspects of the map. The curved single line on the map (red in color) 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** and choose PLUME2.MAP. Since you have imported a new map, the model coordinates have been reinitialized so recalculate the model and reset the minimum contour value and contour interval as before. This new 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 property sheet 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 below.



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 *Linesink Information* property sheet shows the flow rate on the ***Calculated*** tab. 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 should now save the linesink model as EX3.WFL. **Select File->SaveAs** just as you did before. 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 Horizontally** and both models will be displayed at the same time. You can resize the windows and move the model windows around.

Where to Go from Here

The WinFlow tutorial has introduced you to the basics of creating a model. There are many other features, however, that have not been covered. You should see both the Getting Started Using the WinFlow Solver and Concepts chapters for an overview of some of the remaining features. WinFlow Assumptions covers some

hints on how to use WinFlow to solve real-world problems and answers common questions.

WinTran Solver

The WinTran Solver tutorial, described below, introduces you to the most important features of this software in a step-by-step example. You will be given very specific instructions illustrating how to use the WinTran Sovler 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 for the steps described.

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:

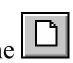
*Select **File** from the main menu and then select **Open** from the pulldown menu*

the following shortened version is adopted in this manual:

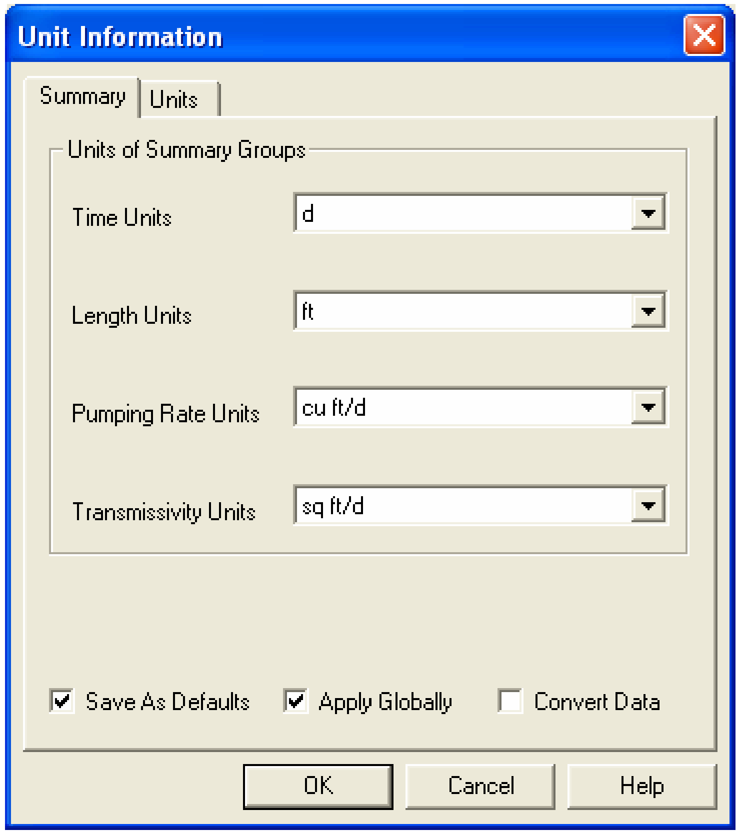
Select File->Open

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 left mouse button twice in rapid succesion. 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

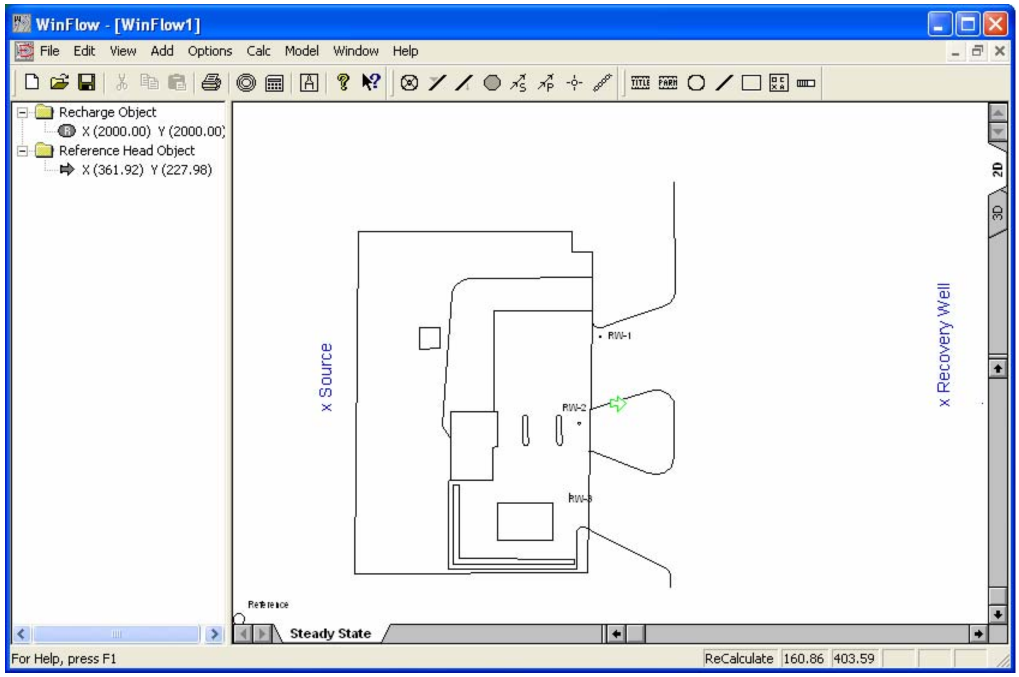
In WinFlow, start a new flow model document clicking the  button on the Standard toolbar or select **File->New**. In Aquifer^{Win32}, click the **File->New** menu and select "AquiferWin32 Flow Model".

The first step is to identify the units you wish to use. Click inside the map view to activate its menus. Click the **Edit->Units** menu to activate the *Unit Information* property sheet and set the values as below:



We are using a consistent set of units as you would have done in WinTran 1.0; however, you can use whatever units you wish. We have selected the **Save As Defaults** checkbox so that these units are used by default. If these are the units you always wish to use, you will not have to edit them again because future models will automatically use them. We have also selected the **Apply Globally** checkbox so that all values throughout the entire model are changed. Finally, we have not selected the **Convert Data** checkbox because no real data has been entered yet. In the event you wanted to change the units on an existing model, selecting this checkbox would automatically convert all the parameters to the new set of units.

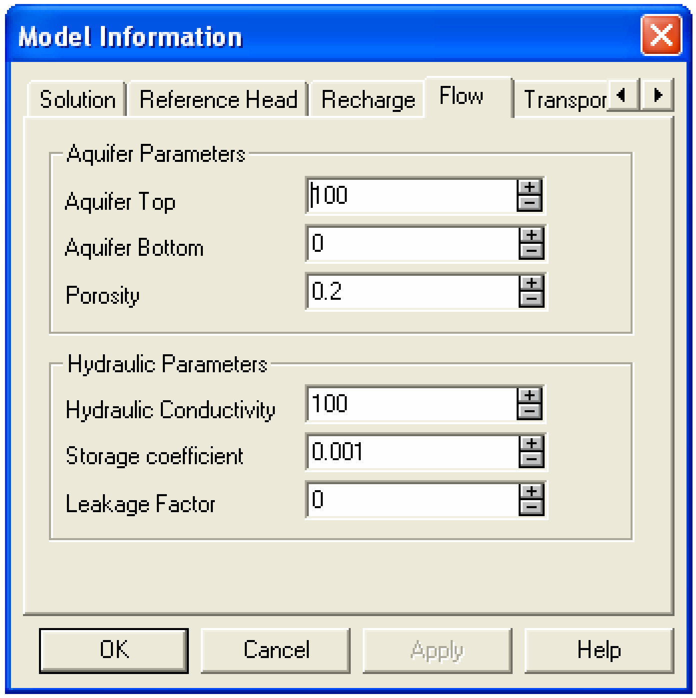
The first step in most models is to import a base map that will serve as your reference point for the model. An example of a base map has been supplied. You will import the map called SITE1.MAP. Select **File->Map** and choose *site1.map* in the standard Windows file dialog. Your screen should now look 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 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 main 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.

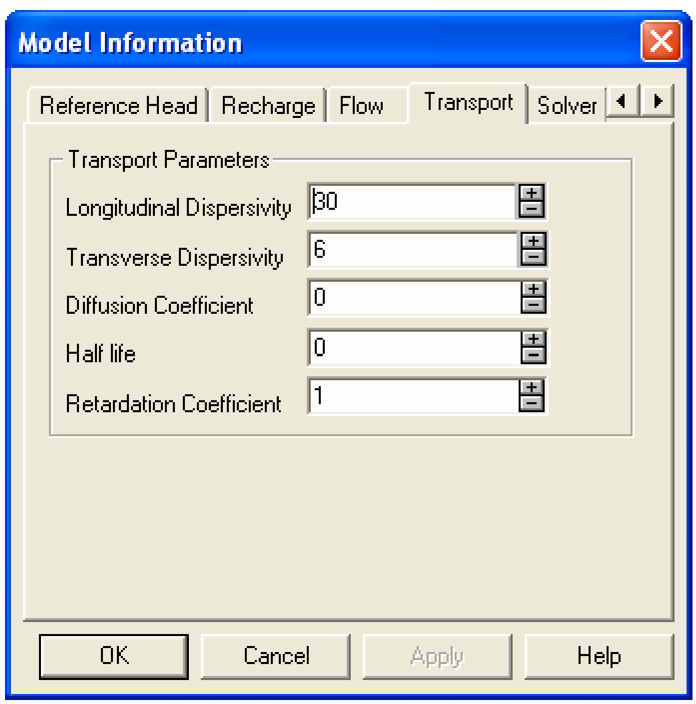
Next, place the cursor over the reference head (green arrow) and double-click the left mouse button. You will see a dialog that lists data corresponding to the reference head. Check the coordinates listed in the dialog and make sure the coordinates are close to the ones given above. Then, change the hydraulic gradient to 1.0e-02 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->Model** menu, change the solution to "WinTran 1.0 Compatible", click the ***Flow*** tab and set the aquifer properties to the values identified below:

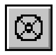


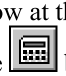
WinTran automatically determines whether an aquifer is confined or unconfined by comparing the computed water level to the aquifer top elevation. The aquifer is unconfined when the head is below the top of the aquifer and it is confined if the head is above the top of the aquifer. In this example, the aquifer is unconfined, so you will make the top of the aquifer very high by setting the top elevation to 100 ft (e.g., a large distance above the reference head value of 25 ft.).

The next step is to edit the transport parameters by selecting ***Transport*** tab. Change the longitudinal dispersivity to 30 ft and the transverse dispersivity to 6 ft as below. Click the **OK** button when you are done.



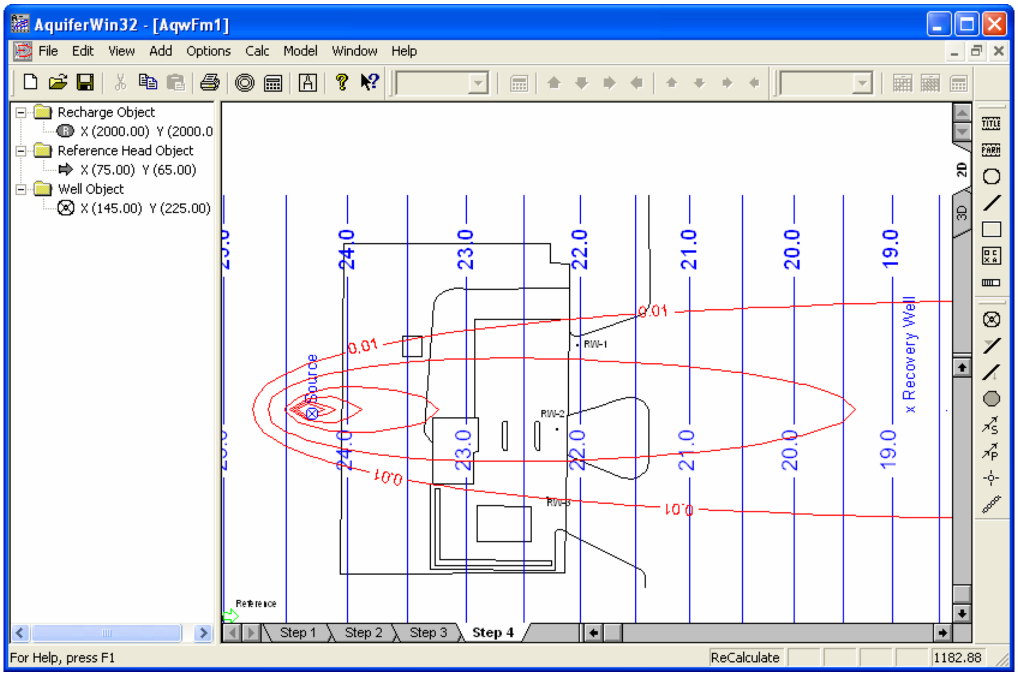
The default is a steady state flow model so we need to change to a transport model by clicking the **Model->Transport** menu. This will give access to the transport related parameters when adding well. Finally, we will change the time-stepping parameters for the transport simulation. Select **Options->Transient** from the menu. Change the number of time steps to 30, Contour Every to 10, Initial Time Value to 0, Initial Time Step Size to 1, Time Step Multiplier to 1.2 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 Well Designator of Well, enter a pumping rate of -1 ft^3/d (the negative sign denotes an injection rate). Next, click the ***Transport*** tab and enter a concentration of 100. Now, click the **OK** button. Note that the word “ReCalculate” appears in the status bar. It is there to remind you that the contours on the screen may not be representative of your current model.

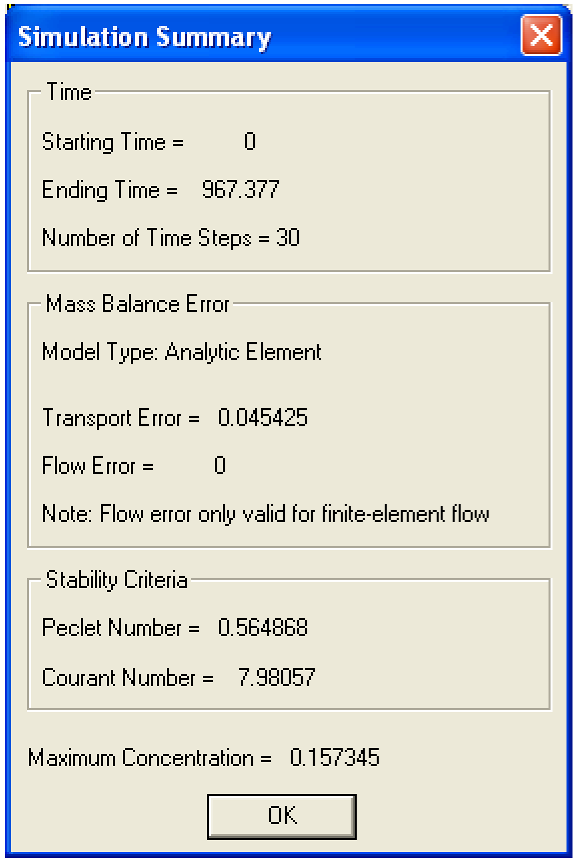
You have now set up the initial WinTran model! Click the **Model->Steady State** menu to only simulate flow at this point. To see the results, simply select **Calc->Recalculate** or click the  button. You should see a series of contours that are straight lines, representing the uniform regional hydraulic 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.

Click the **Model->Transport** model again to run the transport model. Before running the transport model, change the contouring options for both the flow model and for the transport model. Select **Options->Contour->Parameters** the menu, click the ***Levels*** tab, change the Minimum Level to 18.0 ft and the Interval to 0.5 ft. Click the ***Labels*** tab, change the Precision value to 1 Next, click the ***Transport*** tab, set the Min Log Cycle to -2 to use a starting log cycle concentration of 0.01. Set the value of Per Cycle to 5 to use 5 contours per log cycle. Unlike flow, where head is computed on a regular contour interval, concentration is contoured using log intervals. Click the **Font** button and set the font size to 10. Finally, set the Distance between to 200, set the Color to Red and click the **OK** button when you are done. Click the **Yes** button on subsequent dialogs to propogate the changes to all time steps and recontour.

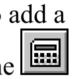
Now select **Calc->Recalculate** and both the flow and the transport model will run. You will first see the flow model computed, followed by a progress dialog showing the percentage complete for the transport simulation. In one of the property sheets above, we told WinTran to contour the results every 10 time steps. You also specified a total of 30 time steps for the simulation. You should four view tabs labeled Step 1 through Step 4. Step 1 represents time of 0 and initial concentration conditions. In this case, the initial concentrations are 0 so only the flow contours are displayed. Click the Step 4 tab which represents time step 30 as below.



During the calculation process, you should also see information displayed on the status dialog including the mass balance error. This value should be small (0.01 percent in this case). Additionally, the Peclet and Courant Criteria for the transport simulation are displayed. Generally, the Peclet number should be less than 2 and the Courant number less than 1. In this case, the Courant number is larger than 1, but the mass balance is good (less than 10 percent) so we will accept the results. This information is also available by clicking the **Calc->Simulation Summary** menu as below.

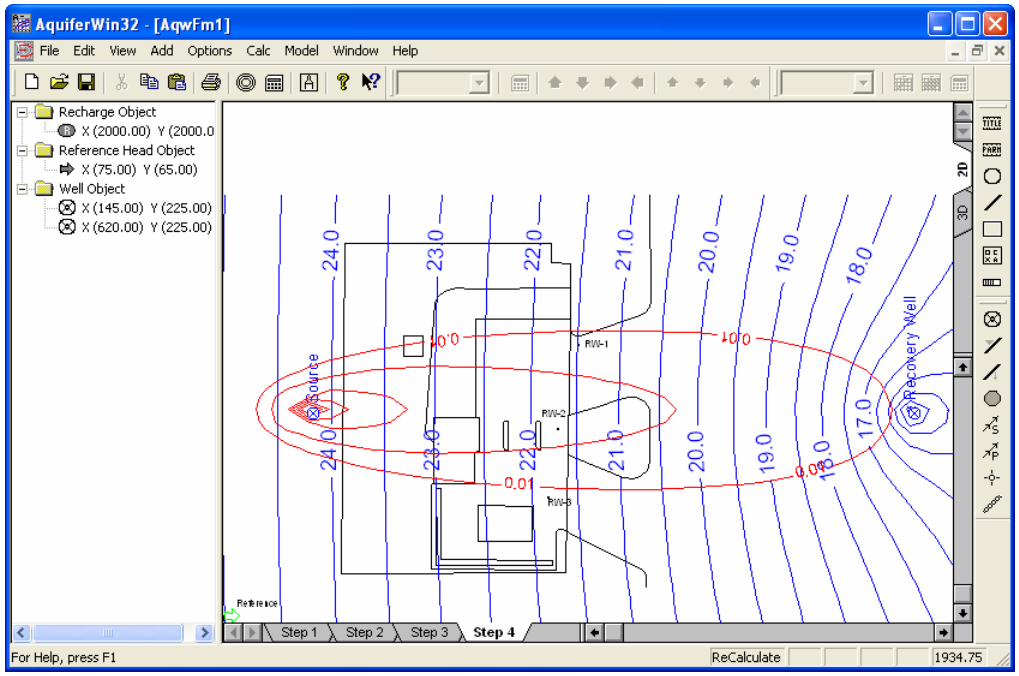


Simulating Remediation

The next step in the tutorial 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. Enter RW as the Well Designator , change the Constant Pumping Rate to 10,000 ft^3/d and click the **OK** button.

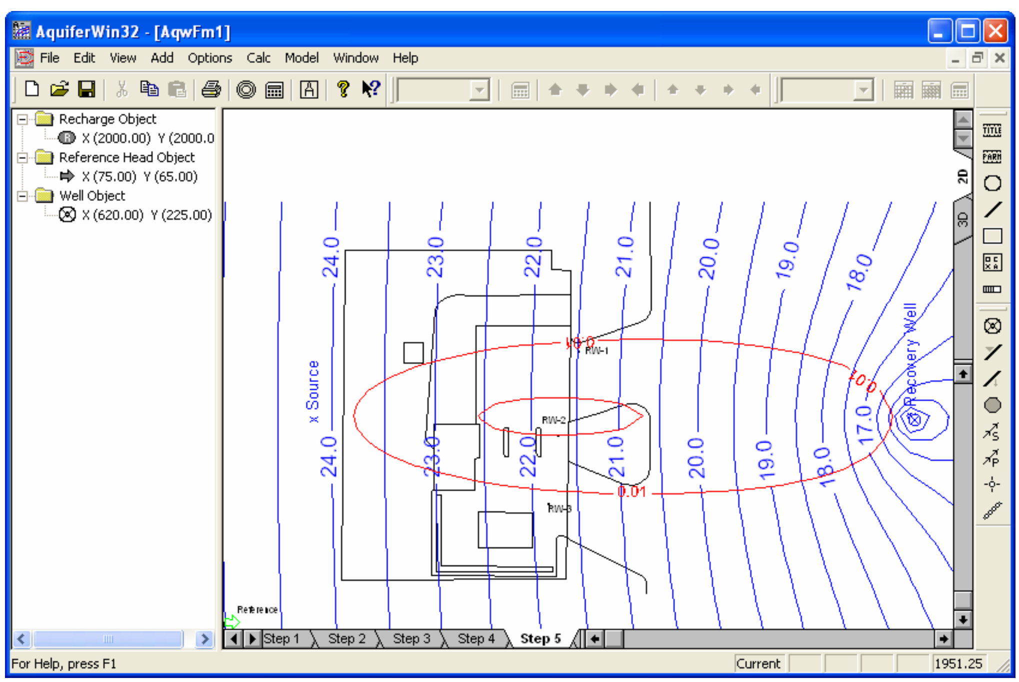
You will now save the concentrations computed at the end of the last simulation as initial concentrations for the remediation scenario. By default, WinTran assumes that initial concentrations are zero everywhere in the aquifer. You may override this default by selecting **Calc->Restart** from the menu.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



As a final exercise, 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 intial conditions and select **Calc->Recalculate** to see the results. Notice that the plume disappears quickly. To get a better picture of the cleanup, select **Calc->Transient**, change the Number of Time Steps to 10, Contour Every to 2 and Maximum Time Step Size to 10. Select **Calc->Recalculate** and 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. You should now be familiar enough with WinTran to use the model on many of your projects

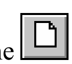
Concepts

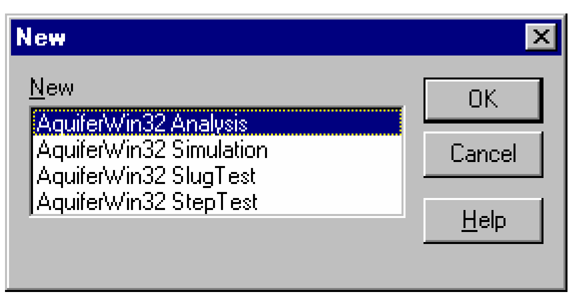
Introduction

This chapter covers basic operating features and concepts for Aquifer^{Win32}. It is a good idea to read this chapter after completing the tutorial to clarify or reinforce the concepts learned during the tutorial exercises. Many of the sections covered below are very important to a thorough understanding of how Aquifer^{Win32} works. We encourage you to read on!

Document Types

Aquifer^{Win32} creates five types of documents or files. These include **Analysis**, **Simulation**, **Slug Test**, **Step Test and Flow Model**. The **Analysis** document is used when analyzing pumping test data. A **Simulation** document is used to simulate an aquifer test and display contours of head or drawdown over an optional base map. The **Slug Test** document is used to analyze slug test data and the **Step Test** document is used to analyze variable pumping tests or step tests. The **Flow Model** document is used to perform analytical flow models. The Modeling version of Aquifer^{Win32} can create and edit all five document types. The Professional version excludes **Flow Model** documents. The Standard version works with **Analysis** and **Slug Test** documents and the Slug Test version can only work with **Slug Test** documents.

To start a new aquifer test, simply click the  button on the Standard toolbar or select the **File->New** menu. Selecting **New** from the **File** menu displays five types of documents or analysis types, including (1) "AquiferWin32 Flow Model, (2) "AquiferWin32 Analysis", (3) "AquiferWin32 Simulation", (4) "AquiferWin32 Slug Test" and (5) "AquiferWin32 Step Test" as shown below.



Clicking the  button on the Standard toolbar creates an **Analysis** document in the Standard and Professional versions and a **Slug Test** document if you are only licensed for the Slug Test version of AquiferWin32. To create a **Flow Model**, **Simulation**, **Slug Test** or **Step Test** document, you must select the **File->New** menu.

Entering Time/Drawdown Data

You can enter the time and drawdown data for an analysis into AquiferWin32 in three ways:

(1) cut and paste data from a spreadsheet application such as Microsoft Excel,

(2) import data from a comma delimited ASCII file, or

(3) manually type the data into the AquiferWin32 spreadsheet.

To paste the data into the AquiferWin32 spreadsheet, simply go into your spreadsheet (Excel or Lotus, etc.), open the spreadsheet file, drag a selection box around the two columns of data (time and drawdown), and copy the data to the clipboard. Next, click on the left side of line 1 in the AquiferWin32 spreadsheet. Type **Ctrl-V** or click the  button on the Standard toolbar to paste the data into AquiferWin32. The default first point is still in the spreadsheet, however, so go to the last line in the AquiferWin32 spreadsheet, click on the line to highlight the data, and hit the **Del** key.

During paste operations, the data is assumed to be in Windows™ text clipboard format with **Tab** characters separating data values and a newline character at the end of each data row. During cut and copy operations this same format is produced. Additionally, the order of the values in the clipboard corresponds to the order of the columns displayed on the spreadsheet. Since the number of and order of columns on the spreadsheet can be adjusted make sure the data matches the spreadsheet into which it will be pasted.

To import the data from a text file, click on the first line in the spreadsheet to highlight that row. Select **File->Import** and find the file using the *File Open* dialog. AquiferWin32 will tell you how many lines were imported. As with pasting from the clipboard, there will be one extra line to be deleted. In this case, however, you will delete the first line in the AquiferWin32 spreadsheet. Furthermore, the data values in the file must be separated by a comma character and each row of data must be terminated with a newline character. The issue of column order versus data order is the same as with cut, copy and paste.

You may also enter the data manually. Simply click on the first cell in the spreadsheet and start typing the data. The first column is for time and the second column is drawdown. Using the **Tab** key or the **Enter** key advances to the next column. When editing the last column, these keys advance to the first column of the next row, creating new data values in the next row if you are currently editing the last row of data.

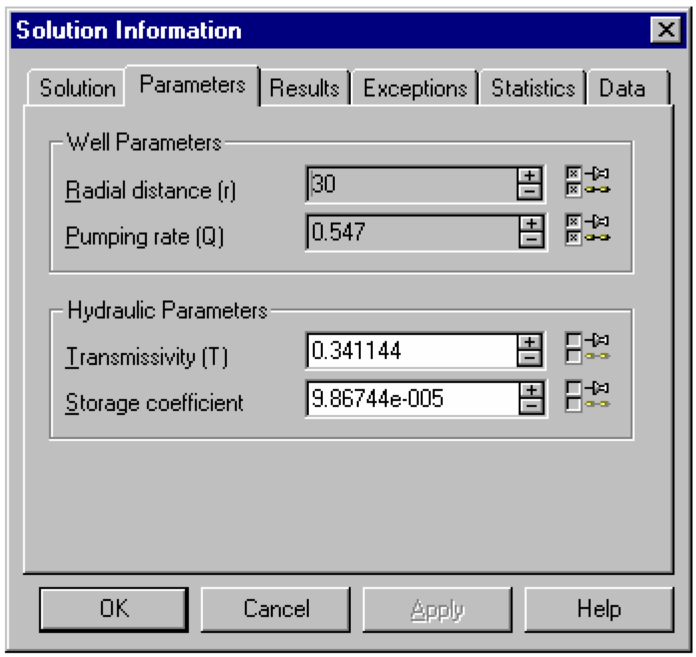
Analysis Parameters

An analysis parameter is a data value that is either required for or calculated by the currently selected analysis. Parameters can be numeric values such as *Transmissivity*, *Radial distance* and *Pumping rate*. They can also be enumerated (specific values from a predefined list) such as *Block Type*. Since analysis

parameters are the central theme to any aquifer analysis application, it is essential to understand how to set and manipulate them.

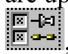
A list containing all parameters for those analyses currently supported by $Aquifer^{Win32}$ would exceed 60 items. Obviously, the user desiring to perform a simple Theis analysis should be sheltered from the 50+ extraneous parameters in order to avoid confusion. Further, this application will routinely undergo enhancement to include additional analyses; in a typical application, this would exacerbate the problem by adding additional parameters over time. Our solution to the problem is to encapsulate each analysis into a separate Dynamic Link Library and use the *Solution Information* property sheet to dynamically alter the user-interface based on the selected analysis. In this manner, the user need only deal with those parameters dictated by any given analysis and the addition of a new analysis need not change the main application.

The *Solution Information* property sheet is where most of the parameters are edited. As previously stated, this property sheet is dynamic in that it presents a list of available analyses for the particular document type and adjusts the tabs of information in the property sheet based on the selected analysis type as shown below.



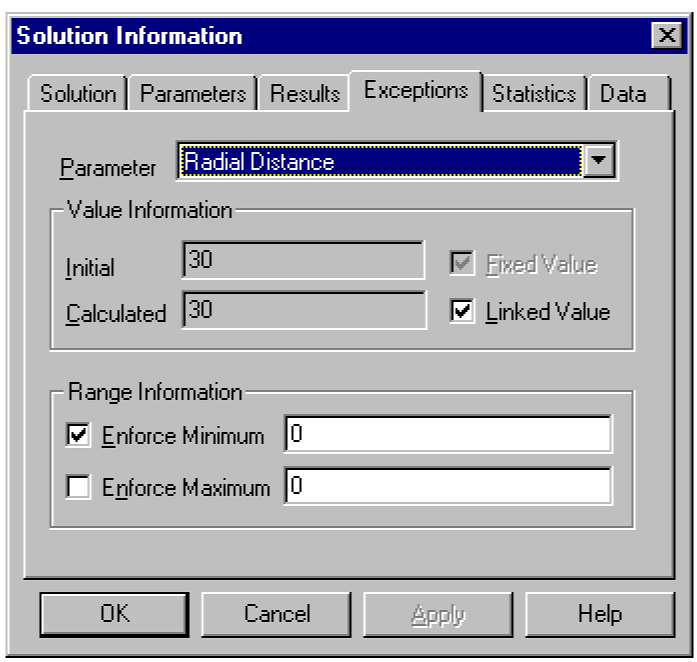
In addition to other tabs, the *Solution Information* property sheet presents two groups of tabs relating to the specific parameters for the analysis. The first group contains the initial guess for each parameter while the second group contains the calculated result for each. There can be any number of tabs for each group depending on how many parameters are involved in the selected analysis. In the case of a "Theis, 1935 (Confined)" analysis, there is one ***Parameters*** tab and one ***Results*** tab. In this simple case, the ***Parameters*** tab and ***Results*** tab are visually identical; however, they have different functionality.

The parameter fields in the ***Results*** tab are all "read-only" such that the values can be selected and copied to the clipboard but not modified. In some cases, parameters calculated by the analysis only appear on the ***Results*** tab. Such calculated parameters are provided in case the user would like to verify calculations based on the appropriate equations.

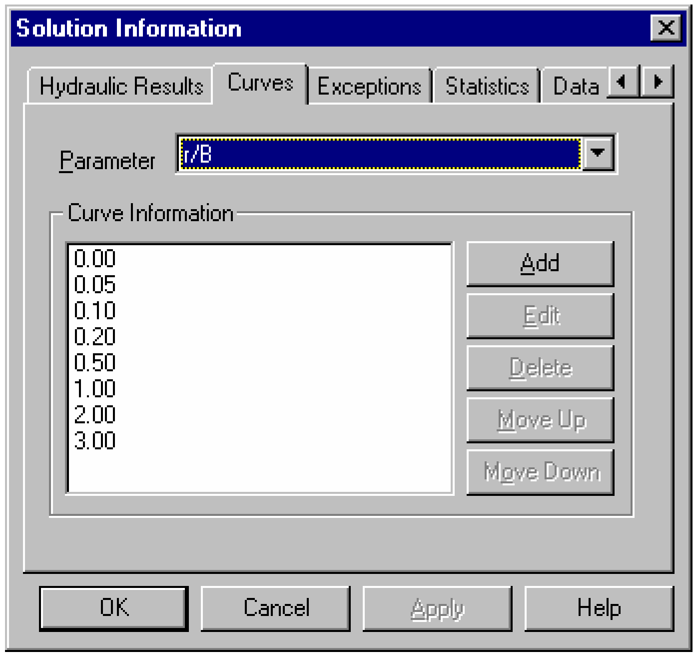
Fields on the ***Parameters*** tab are, in most cases, editable. The parameter values represent the initial value to be used during the optimization calculation; these values are updated automatically when a manual match is performed. A special control, , appears to the right of most parameters. This control contains two separate check-boxes each of which can constrain adjacent parameter value. Although not applicable to all parameters, the lower check-box controls whether a parameter is "linked". A linked parameter gets its value and units from a value either specified or calculated in some other property sheet. An example is *Radial distance* which can derive its value from one of two different places. In the case of a simple analysis, one that does not have well objects defined in the ***Site Map*** view, the value for *Radial distance* is linked to the value entered into the ***Pumping*** tab of the *Aquifer Test Information* property sheet. In the case of a multiple well analysis the value for *Radial distance* is calculated based on the distance between the selected pumping well and the monitoring well. In either case, you can click on the check-box to unlink the parameter. After is has been unlinked, the value and/or units can be changed. If it is subsequently linked, the value and units will revert back to those of the value to which it is linked. This concept allows you to manipulate the parameters during an analysis more easily.

The upper check-box controls whether a parameter is "fixed" during optimizations. Any parameter that is fixed will not be allowed to change during the optimization calculations. This flexibility is unique to Aquifer^{Win32} and allows optimization on parameters that are typically held constant. An example would be *Aquifer Thickness* in a partially penetrating well analysis. The physical thickness may not always be representative of its hydraulic impact or may no even be known. You could allow it to vary within a specified range with Aquifer^{Win32} to get a better match.

Should you desire to constrain a given parameter with respect to its minimum and/or maximum acceptable values during an optimization, the ***Exceptions*** tab is where to do it. In reality, this step may be required to get the solution to converge. The ***Exceptions*** tab contains a combobox listing all parameters involved in the solution. Once a specific parameter is selected, you can dictate how the optimization calculations treat it. As shown below, you can enforce a minimum value and/or a maximum value. You can also fix, link and specify the initial value as an alternative to the ***Parameters*** tab.



Another noteworthy tab within the *Solution Information* property sheet is the ***Curves*** tab as shown below for the "Hantush, 1964 (Leaky Confined Partial Penetration)" analysis. This tab only appears when the selected analysis allows for multiple type curves. The *Parameter* combobox in this case has numerous parameters including; r/B, Screen Length, Depth to Screen Top, Aquifer Thickness, Kz/Kr, Monitor Depth to Screen Top, Monitor Screen Length. Using this flexibility, you can generate sets of type curves to quantify the impact of varying any one of these parameters with the rest held constant.



Well Construction Information

It is very important to understand that all depths referenced in well construction information are relative to the top of the aquifer.

Aquifer[Win32] allows entry of many parameters that apply to one or more analyses. In order to adopt a paradigm that applies to all available analyses, some conventions were adopted out of necessity. One of the most important conventions deals with well construction information.

Generally, an attempt has been made to limit the parameters that are presented to the user to those pertinent to the currently selected analysis. Well construction information is the most notable exception.

In Aquifer[Win32], whether you are performing a simple analysis or you have defined wells, you must provide the well construction information whether it is pertinent to the currently selected analysis or not. **Well construction parameters such *as Screen Top Depth* and *Maximum Water Depth* are relative to the top of the aquifer and not the ground surface. In the case of an unconfined aquifer, the top of the aquifer is the water table.**

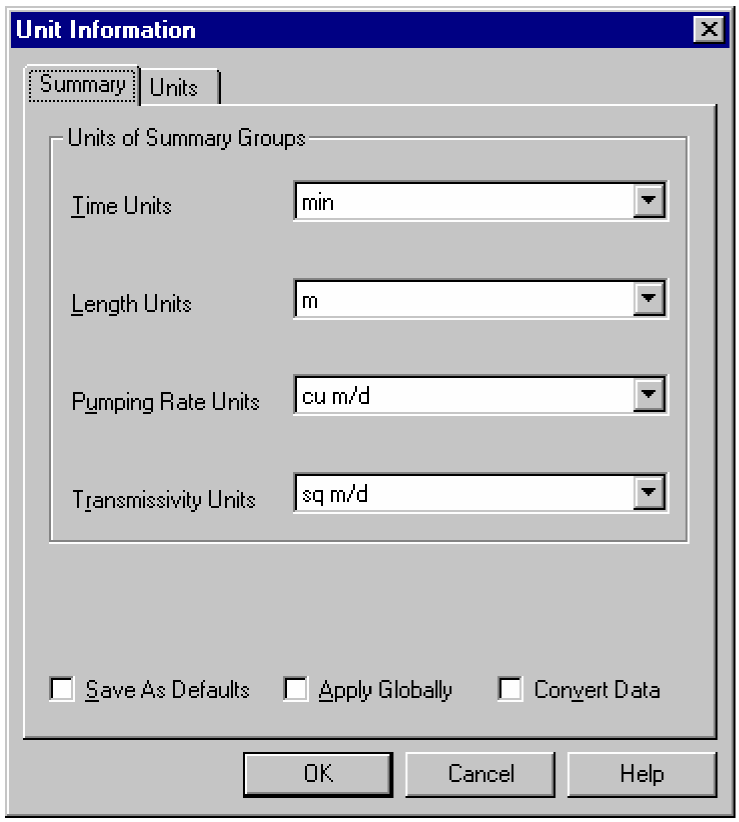
If you are unsure whether a particular analysis uses any of the well construction parameters, refer to the *Solution Information* property sheet or the respective journal article detailing the analysis. If the parameter appears in the *Solution Information* property sheet, it is used and will likely be "linked" to the information entered into the *Well Information* property sheet or the *Aquifer Test Information* property sheet.

Units

*The easy way to deal with units is to select **Edit->Units** right after creating a new document. Choose the units you will use, click on the **Save as Default** check-box, and click **OK** to save these units as your defaults.*

Aquifer^{Win32} allows the ultimate flexibility when it comes to setting units; however, it is imperative that one understands the paradigm. Every parameter contained in Aquifer^{Win32} has units that can be expressed in terms of three generic "units"; these are length (L), time (T) and volume (V). It is a good idea to use a consistent set of units for your analysis, but Aquifer^{Win32} does not require this. You may have drawdown in meters, time in seconds, transmissivity in gallons per day per square foot, pumping rates in gallons per minute, etc. You may also set up your default units that will be used whenever you run the Aquifer^{Win32} program.

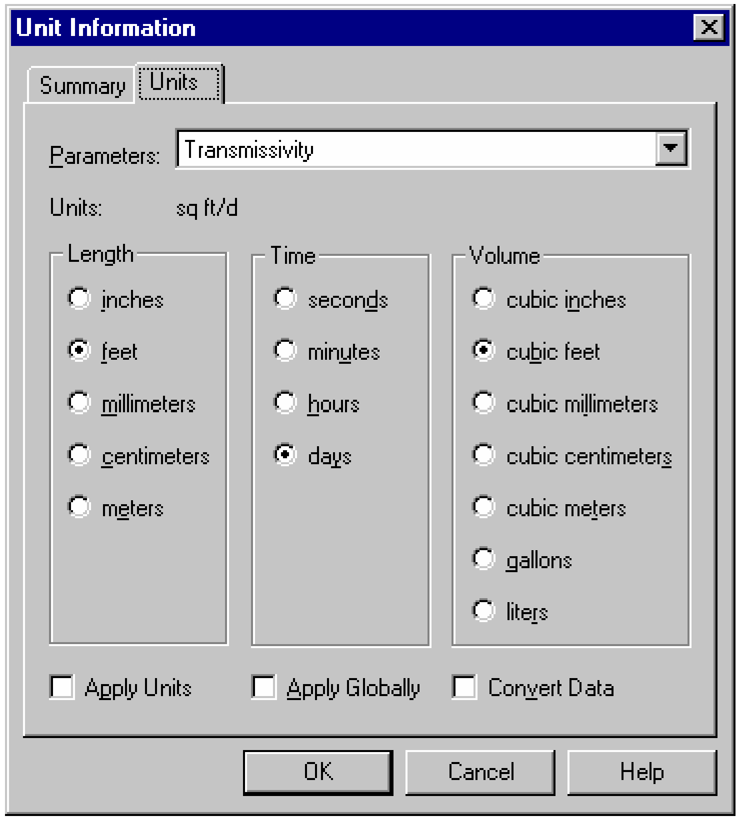
Selecting the **Edit->Units** menu displays the *Unit Information* property sheet with two tabs, as shown below. The first tab is labeled ***Summary*** and contains the four basic parameters and their respective units. The values represent either the program defaults or the values previously saved as default. Under normal circumstances, you would set the summary units once and save them as your defaults. If you always use the same types of units in your analyses, then you never need to change these again. Click the **Save As Default** check-box to do this and then click **OK**.



The **Convert Data** check-box is checked when you want Aquifer^{Win32} to convert the existing data from the current set of units to a new set. The **Apply Globally** check-box is used to convert data for all wells in the current document. If this box is not checked, the data would only be converted or have units set for the currently selected well.

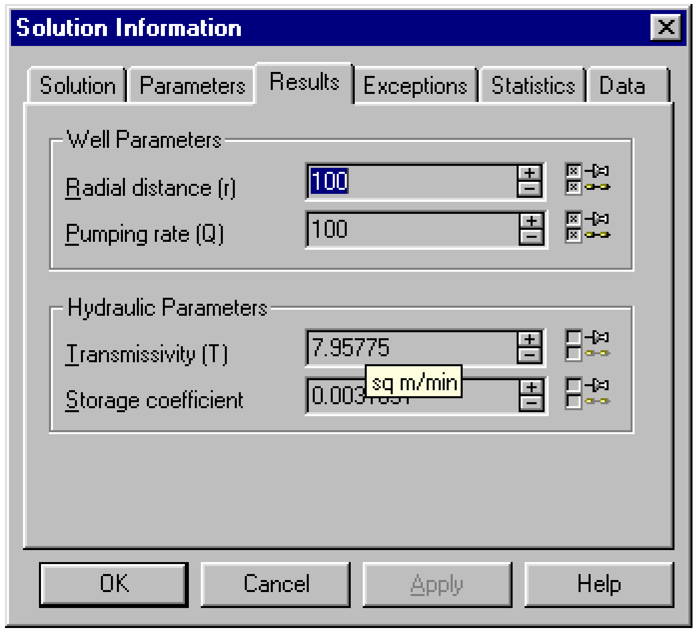
The ***Units*** tab allows specific parameters to be assigned units. As mentioned above, every parameter contained in Aquifer^{Win32} has units that can be expressed in terms of three generic "units"; these are length (L), time (T) and volume (V) with the possible values organized into three sets of radio buttons. When a parameter is selected in the *Parameter* combobox, the appropriate radio button sets are activated. Simply select

the desired unit from each radio button set. One of the most complicated parameters, Transmissivity, can be expressed in dimensionless units as V/T/L. If you select gallons, days and feet for V, T and L the resulting units would be gallons per day per foot (gal/d/ft). If you select cubic feet, days and feet, the resulting units would be square feet per day (sq ft/d). (**NOTE: If changes are made to both tabs of this property sheet, the values on the Summary tab take priority)**. Accept the property sheet by clicking the **OK** button.

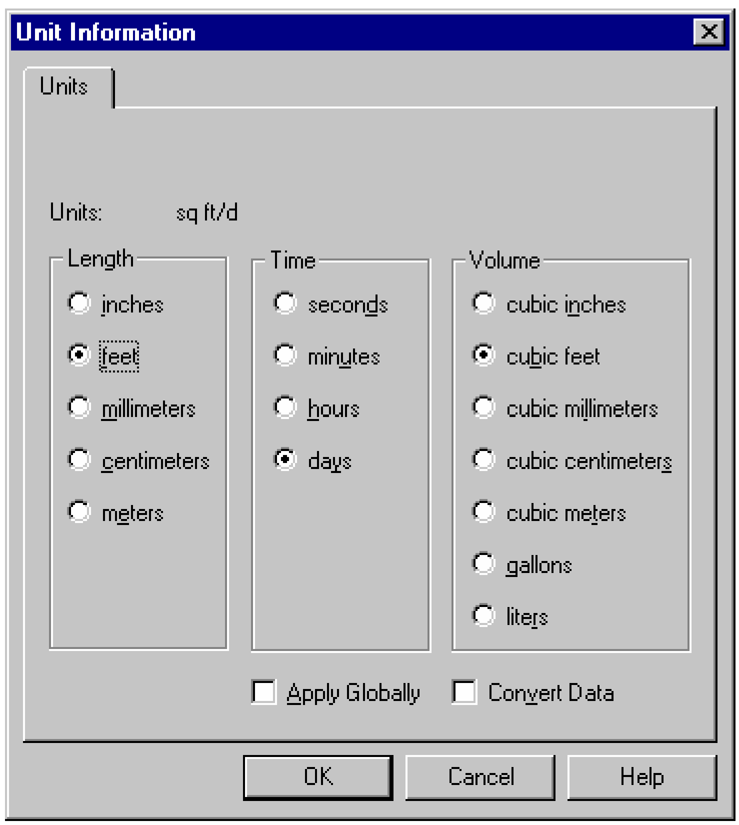


Display of units in tooltips does not work in Windows 3.1. Call Bill Gates to ask why.

Aquifer[Win32] makes it easy to track what units are assigned to each data item in the program. When viewing a data field on a property sheet, simply place the mouse cursor over the field and wait a second. A tooltip will be displayed that lists the current units assigned to that field. The following is an example where the cursor was over the *Transmissivity* field of the *Solution Information* property sheet. The tooltip reads “sq m/min” which is square meters per minute.

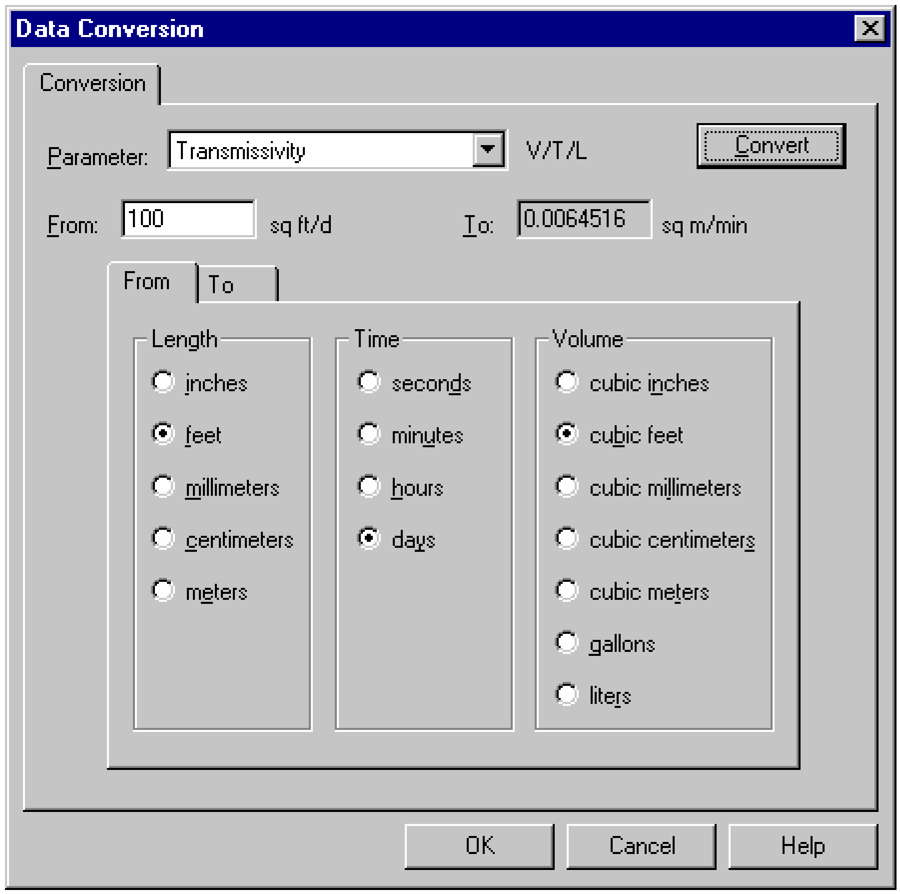


You can also reset the units applied to any data type using a similar approach. With the mouse cursor hovering over a data field, click the right mouse button and choose **Units** from the context menu. You will see the following property sheet that is similar to the one shown above.



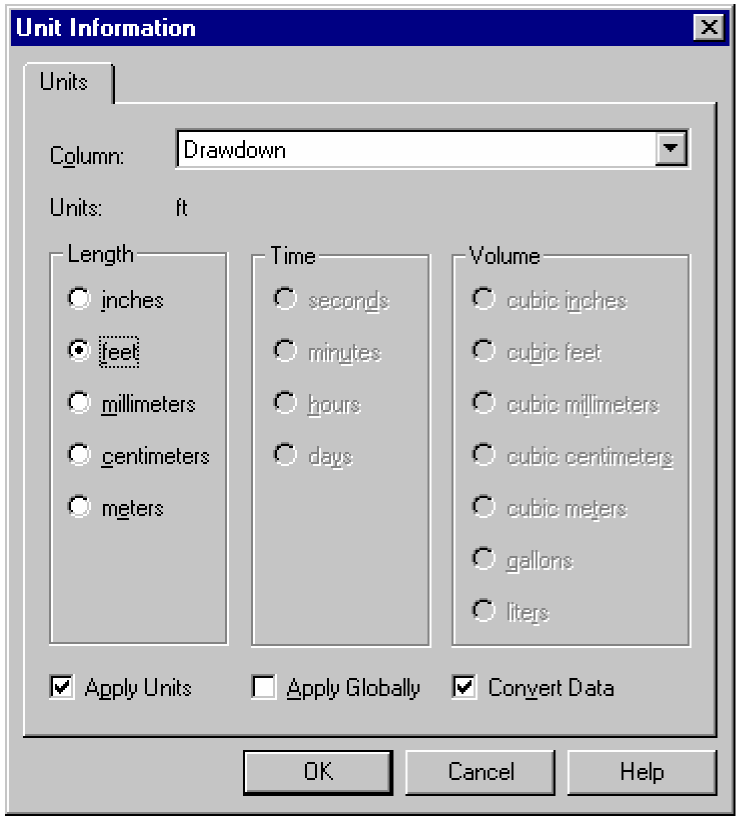
Set the units that you want for this data item and check the **Convert Data** option to automatically convert from the current units to the new ones. Choose the **Apply Globally** option if you have more than one observation well in this data set and you want to affect them all.

Aquifer[Win32] also has a couple of ways of accessing a unit conversion calculator. This calculator does not change the units used in your analysis but can be used to take data in another set of units and transform it to your set of units. For example, suppose you want to set an initial guess for transmissivity in the *Solution Information* property sheet. This is the same property sheet as shown above with units of sq. m/min. You have a rough estimate for transmissivity in ft^2/day but need to enter it here in m^2/min. You can right-click on the *Transmissivity* field and select **Data Conversion** from the context menu. Another property sheet is displayed as shown below. You simply enter your T value in ft^2/day and choose "feet" for *Length*, "days" for *Time*, and "cubic feet" for *Volume* and click the **Convert** button. You now have a value in m^2/min. Click **OK** to enter this value on the property sheet. This example is shown below.



Note that you cannot change the *To* field here because you are converting a value to your set of units. You can access a more generic data converter by selecting **Calc->Data Conversion** from the main menu. In this case, you can enter both the *To* and *From* units. Using this calculator does not change anything in Aquifer[Win32]; it is simply provided to help with unit conversions.

A final type of data conversion is related to columns of the spreadsheet view. You may right click on any column of the spreadsheet and select **Column Conversion** from the context menu. Enter the new units for the column of data and check the **Apply Units** and **Convert Data** check-boxes to recalculate the data in the new set of units. For multiple wells, use the **Apply Globally** option as well. An example of the column conversion property sheet is shown below.



Manual Curve Matching

Aquifer^{Win32} is designed with the philosophy that manual matching of the data to type curves is preferable to automated matching. When you learn about aquifer test analysis in seminars and college courses or through studying popular texts on the subject (see Fetter, 1994, page 220 for example), this is the technique that is taught. Aquifer^{Win32} is different from most other aquifer test analysis software because it provides a means of displaying multiple type curves and you choose the values of the type curves. Aquifer^{Win32} also provides you with the value of the match point you have selected.

When performing a manual type curve match, you use the arrow keys on the keyboard (up, down, left, right) to move the data over the type curve(s). As the data move, the estimates of Transmissivity and Storage are updated on the status bar. If you have defined a legend on your plot, the parameter values are also updated on the legend. You may also move the data by clicking the left mouse button over the arrows on the toolbar. The toolbar large arrows move the data a greater distance than the small arrows. The arrow keys on the keyboard are equivalent to the large arrows on the toolbar. You may hold down the shift key when pressing the arrow keys to simulate the smaller arrows on the toolbar; in this manner, you can make fine adjustments to the match.

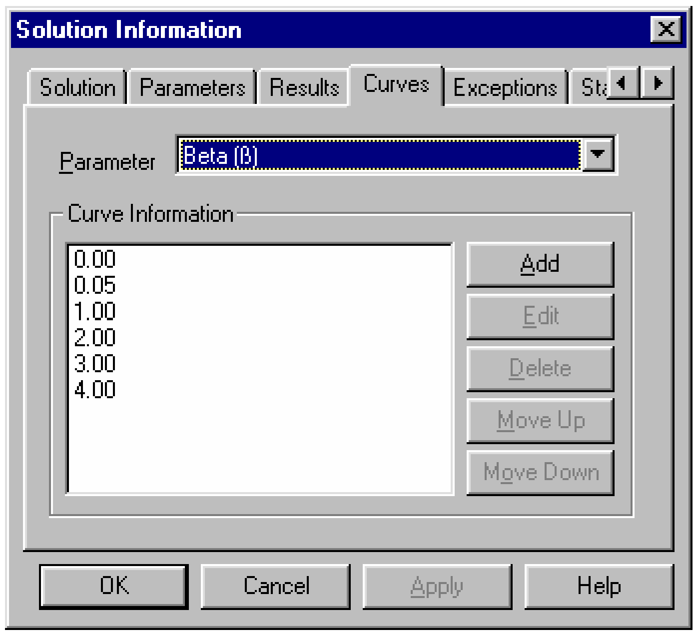
Type Curve Methods

The basic idea is to move the data so that they match as closely as possible to the type curve that has been selected. In the case of single type curve methods (e.g., Theis), obviously there is only one curve. For many others, though, there are an

infinite number of curves that could be chosen. Aquifer^{Win32} has a default set of type curves that closely correspond to those in the original paper that presents the technique. You may add or subtract from this number or define your own values.

As you move the data over multiple curves, the parameter estimates change based upon the currently selected curve. The selected curve is drawn in a different color than the other curves (default is blue). To change the selected curve, you may either select the curve value from the combobox on the toolbar (immediately to the left of the calculator button), select **Edit->Toggle Type Curve**, or press **Ctrl-T** on the keyboard. If you move the data over the newly selected curve and press the calculator button to optimize the solution, Aquifer^{Win32} will use the selected type curve data for initial guesses for the new solution.

You change the type curves that are displayed by selecting **Edit->Solution** and pressing the ***Curves*** tab. An example is shown below for one of the Hantush leaky solutions:



In some cases, there is more than one value in the *Parameter* combobox. In the case of "Neuman, 1974 (Unconfined Partial Penetration)", valid values in the *Parameter* combobox are "Beta (β)", "Screen Length", "Depth to Screen Top", "Aquifer Thickness", "Kz/Kr", "Monitor Screen Length", and "Monitor Depth to Screen Top". When constructing type curves, the initial guess parameters are used except for the parameter selected in the *Parameter* combobox; the specific values for this value are listed in the *Curve Information* list box. As you can see, you have great flexibility when constructing type curves.

Straight-Line Methods

Straight-line methods are different from type curve methods in that the data are not moved with the cursor keys or the arrows on the toolbar. In this case, you must move the line or change the slope of the line using the mouse. To move the line up or down without changing the slope, move the mouse over the center of the line until the cursor changes to a four-arrow cursor. Now hold the left mouse button down and drag the line up or down. To change the slope, move the cursor to either end and wait until the cursor changes to an east-west cursor. Hold the left mouse button

down and drag the end of the line to change the slope. You can also double-click on the regression line to enter the slope and intercept directly.

Automated Curve Matching

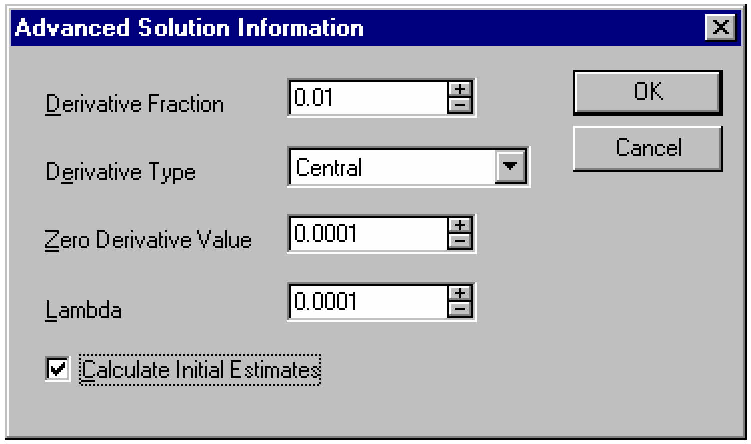
Aquifer^{Win32} uses the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to find the best statistical match between the field data and the type curve you have chosen. Click the button on the Match toolbar or choose **Calc->Optimize** from the menu to implement the automatic match. You will see T (transmissivity) and S (storage coefficient) displayed on the status bar at the bottom of the Aquifer^{Win32} window as each iteration of the match is finished

You may view detailed results of the nonlinear least-squares match by selecting **Edit->Solution** from the menu. The first tab, ***Solution***, on the property sheet displays the type of analysis you have performed. Other tabs include:

Parameters	The initial guesses for parameter values
Results	Optimized parameters computed by nonlinear least-squares
Exceptions	Unlink or unfix parameters and set enforced minimum and maximum parameter values
Statistics	Match point data and statistical measures
Data	Values calculated for each type curve

The Marquardt nonlinear least-squares procedure will estimate any parameter you choose. You may fix (or hold constant) any of the parameters, however, by selecting **Edit->Solution**, choosing the ***Parameters*** tab, and using the icon. If the little pushpin is not checked, then the parameter is free to vary during the optimization. If you click the pushpin so that it looks like this , then that parameter is fixed and will not vary. The most common parameter to fix in the Hantush method, for example, is the value of r/B, which defines what type curve you are matching. For example, suppose that you have a type curve on your screen with a r/B value of 0.05 and you want to have Aquifer^{Win32} compute the best values of T and S for this given r/B value. You would enter a value of "0.05" for the *r/B* field on the *Solution Information* property sheet and make sure the button looks like .

The automatic procedure will not always converge to a valid solution the first attempt. You may get an error message stating that the numerical solution has become unstable. This means that the Marquardt procedure could not come up with an improved estimate for the pertinent parameters. One problem with the Marquardt method is that your initial guesses for parameters must be fairly close to the "right" answer before it will work properly. You may also get a message after the error dialog that asks if you would like to turn off the option to automatically estimate initial guesses for parameters. Select **Yes** if you see this dialog. The default behavior is to calculate an initial guess for T and S using a straight-line method. Although this works well for some analyses, it doesn't work for all of them. This dialog controls the **Calculate Initial Estimates** check-box from the *Advanced Solution Information* dialog show below. This dialog is accessed from the **Advanced** button on the ***Solution*** tab of the *Solution Information* property sheet.



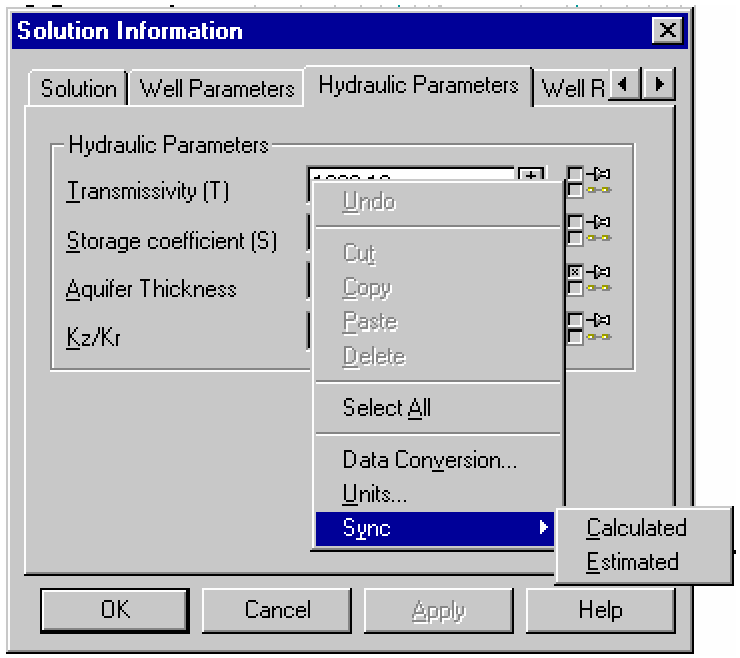
If you have convergence problems, this dialog is the first place to go. Turn off **Calculate Initial Estimates** by clicking on the check-box to remove the check mark. Now, go back and try a manual match making sure to select the type curve you are matching to. Also, make sure you have fine-tuned the match after you have selected the curve. Merely selecting a type curve does not update the initial guess of the type parameter; this is only done when you perform a manual match.

If optimization fails again, you should increase the value of *Lambda* by several orders of magnitude. With higher values of Lambda the solution is more likely to converge but the precision of convergence is diminished. If you set *Lambda* to a value of 1 or 10 and you are still can't achieve convergence, go to the *Solution Information* property sheet and look at the ***Results*** tabs. One or more of the parameters are likely diverging and have become invalid. A likely candidate might be the parameter represented by the type curves. Go to the ***Exceptions*** and start enforcing minimum and maximum values. The more you constrain the solution, the more likely it will converge.

Whether you are having convergence problems or not, it is a good idea to perform a manual match. Take a look at the data points that don't fit the curve; you have made a conscious decision that these points are of less importance than the ones that more closely match the curve. This being the case, set a higher value of weight on these points so that the optimization procedure can ignore them as well.

Manipulating the information on the *Advanced Solution Information* dialog, fixing and/or constraining parameters, changing data weights and initial guesses are the four areas where you can influence the convergence. Sometimes it takes a good bit of trial and error to achieve convergence, especially if the data deviates substantially from theoretical. Also remember that a manual match is just as good or maybe better because you are using you professional judgement when analyzing the data. The numerical algorithm can only minimize residuals.

During the course of optimization, you may wish to update the initial estimates of parameters to correspond to the values recently calculated. This would allow future optimizations to converge faster. Although you could manually copy/paste the values between the results and parameters pages, a context menu is available to perform this task. Find the field in the parameters or results set of pages, right click the mouse on the data field and select the **Sync->Estimated** menu to copy the calculated value into the estimated value to used in subsequent optimizations.



Manual Parameter Estimation

Some of the aquifer test analysis methods use a family of type curves that can be dependent on a number of parameters. In these cases, it can be somewhat difficult to come up with initial estimates that will allow the solution to converge. Also, it is sometimes desirable to change parameters and see what affect they have on the predicted well responses to try and determine better initial estimates or assess sensitivities of specific parameters. In order to enable this functionality, the **Apply** button on the *Solution Information* property sheet acts differently depending on the view from which it was activated. If the Curve Match view is active, the **Apply** button will perform an optimization. If the Predicted view is active, it will simply recalculate the predicted curve. Additionally, if a parameter is changed with the Predicted view active that would affect the match point, the new match point is calculated and the Match view will be updated; this is useful if it is desired to see the match point for given values of T and/or S without randomly moving the data around until you get close to the desired values.

The procedure is to activate the Predicted view, click the **Edit->Solution** menu to display the *Solution Information* property sheet, locate the parameter you want to change in the parameters set of tabs, change the value, use the **Sync->Calculated** context menu and click the **Apply** button. The predicted curve will be recalculated and displayed visually demonstrating the impact this parameter change has had on the match. Additionally, the ***Statistics*** tab can be used to quantitatively determine how well the predicted curve matches the observed data.

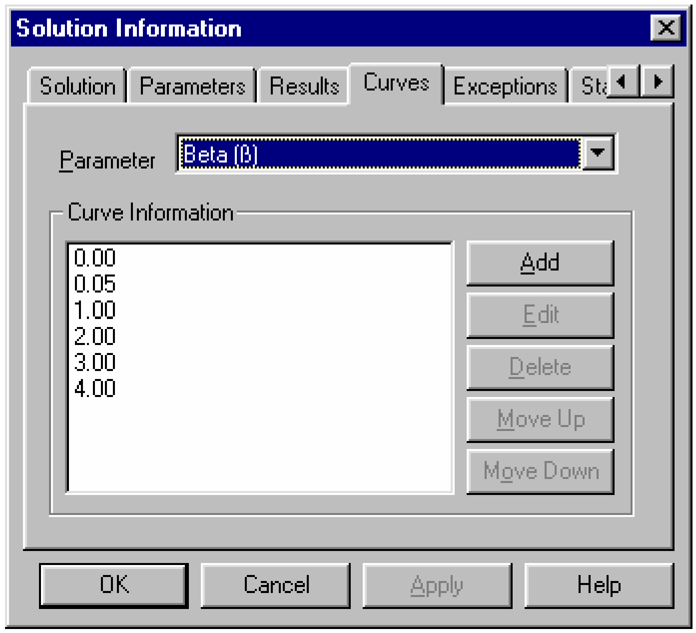
Multiple Type Curves

Many of the aquifer test analysis methods use a family of type curves. In this case, your goal is to not only fit the data to a curve but to also choose the best curve. Aquifer^{Win32} is designed to help in this type of analysis by displaying a family of type curves for these methods. You may choose how many curves are displayed and may

use this feature to prepare sets of master type curves for use in analyzing data in the field.

If you are familiar with other aquifer test software, you will see that Aquifer^{Win32} has a slightly different philosophy or focus. Whereas most other aquifer test analysis packages rely heavily on the automatic matching procedures, Aquifer^{Win32} is designed to be more like the visual or manual matching techniques advocated by most hydrogeology text books (See Fetter, 1994 for example) and by ASTM.

You change the type curves that are displayed by selecting **Edit->Solution** and pressing the ***Curves*** tab. An example is shown below for one of the Hantush leaky solutions.

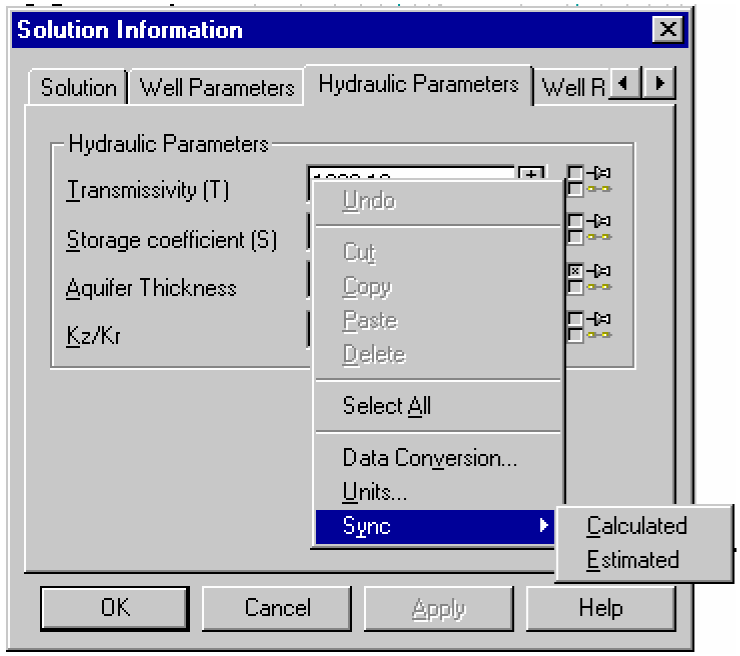


In some cases, there is more than one value in the *Parameter* combobox. In the case of “Neuman, 1974 (Unconfined Partial Penetration)”, valid values in the *Parameter* combobox are “Beta (β)”, “Screen Length”, “Depth to Screen Top”, “Aquifer Thickness”, “Kz/Kr”, “Monitor Screen Length”, and “Monitor Depth to Screen Top”. When constructing type curves, the results parameters are used except for the parameter selected in the *Parameter* combobox; the specific values for this value are listed in the *Curve Information* list box. As you can see, you have great flexibility when constructing type curves.

In cases where a number of parameters affect the type curves, optimization of parameters can invalidate the type curves by changing the value of a parameter that was used in calculating the type curves. You can identify this condition by looking at the last pane in the status bar when the type curves are displayed. It will contain a value of "Valid" or "InValid" indicating if the type curves displayed are valid for the current set of parameters. If they are not, use the **Calc->Recalculate Type Curves** menu to recalculate them.

To directly control the parameter values used in the type curves, you will have to update the parameter values on the results set of pages in the *Solution Information* property sheet. These values are not directly editable but can be changed by using the **Sync->Calculated** menu on the edit control context menu. Find the field in the parameters set of pages as you would to adjust the initial guess for a parameter; enter the value, right click the mouse on the data field and select the **Sync->Calculated**

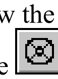
menu to copy this estimated value to the calculated value which will be used in subsequent calculations of type curves.

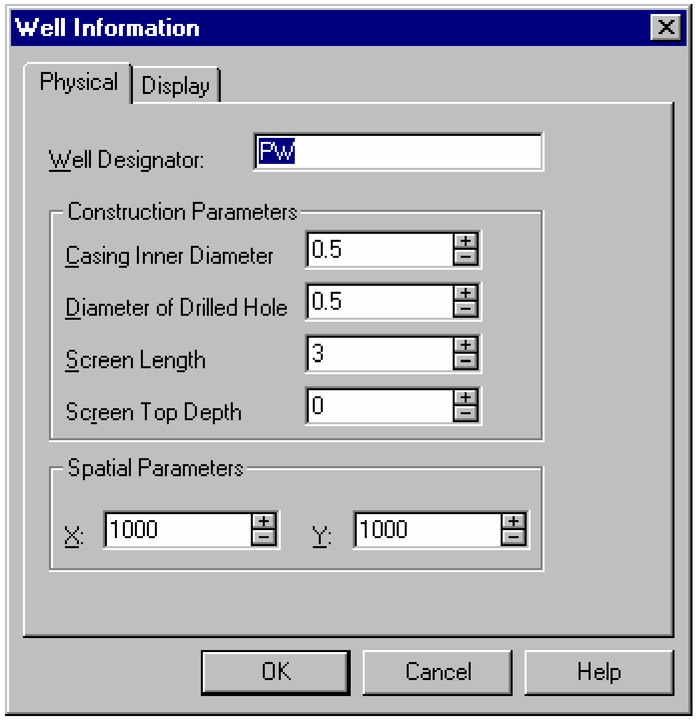


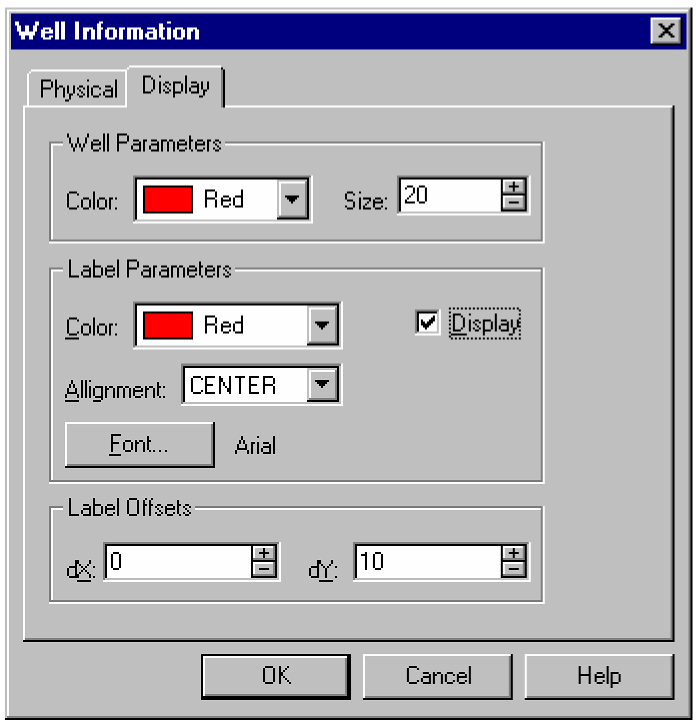
Data Setup for Multiple Well Analyses

Aquifer^{Win32} has advanced features that require additional effort to implement and understand but provide enhanced capabilities for those who choose to spend the time. Three high level concepts are contained in Aquifer^{Win32} which relate to analysis of multiple wells. In reality, they can be looked at as the three preliminary steps to analyzing the data from an aquifer test. The three concepts are Site, Aquifer Test and Analysis; each will be discussed in this section.

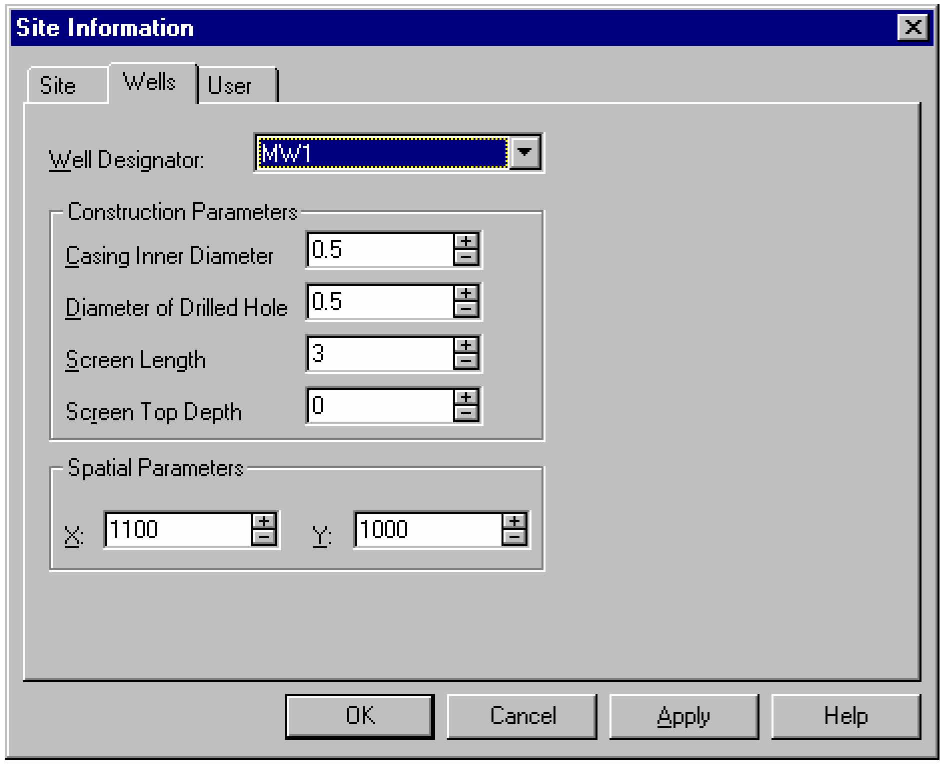
Site

The site is nothing more than the collection of wells present at a given field location. The ***Site Map*** view is the graphical view of the site and the *Site Information* property sheet, accessed via the **Edit->Site** menu, is used to view the detail. Wells are added to the ***Site Map*** view using the **Add->Well** menu or the  button on the Analytic toolbar. The *Well Information* property sheet is then presented and should be filled in. The ***Physical*** tab contains the well construction information and the ***Display*** tab contains parameters controlling how the well will be displayed on the ***Site Map*** view.





The *Site Information* property sheet, accessed via the **Edit->Site** menu, has three tabs of information. The ***Site*** tab contains predefined user fields that can be used and the ***User*** tab contains nine additional user fields that can be customized to meet your needs. These user fields are available to Parameter elements which can be used in headers/footers or to annotate views. The third tab, ***Wells***, is used to edit individual well objects. The *Well Designator* combobox contains all wells defined for the site.

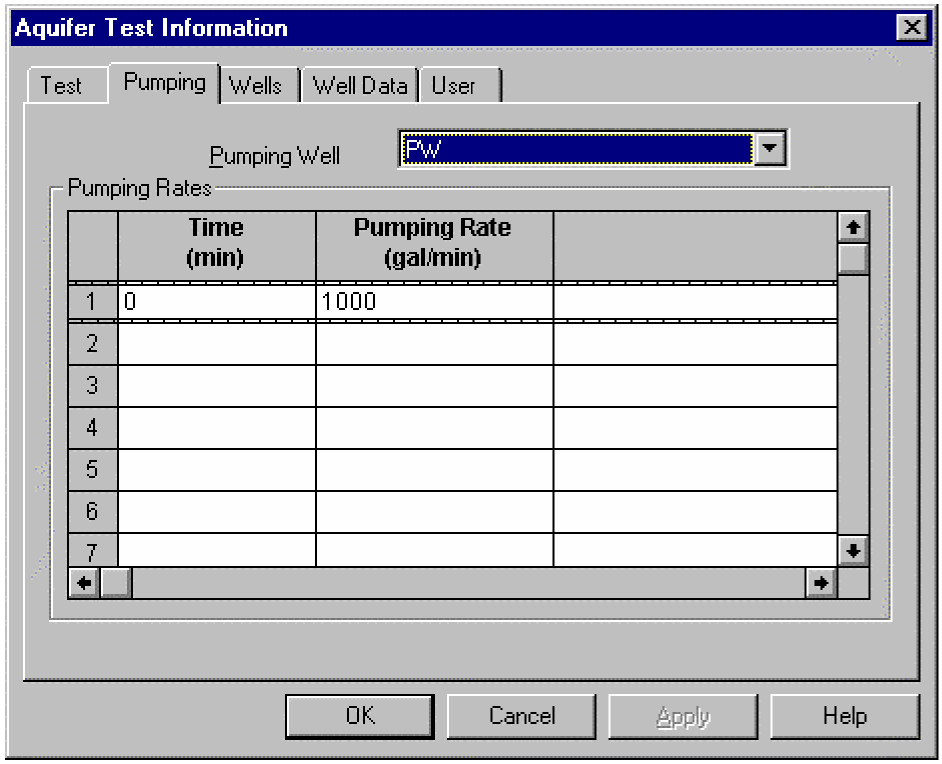


Although the ***Site Map*** view can be fully implemented by loading a basemap and entering all wells, the minimum requirement is to enter the pumping well and all monitoring wells you wish to analyze. An approach that seems appropriate would be to load a basemap and add all the wells displaying their designations. This file can then be saved to serve as a template for future pump tests at the site. When a specific pump test is performed, reset the size and/or color of the pumping well and wells monitored during the pump test. The resultant map would likely be incorporated into your report.

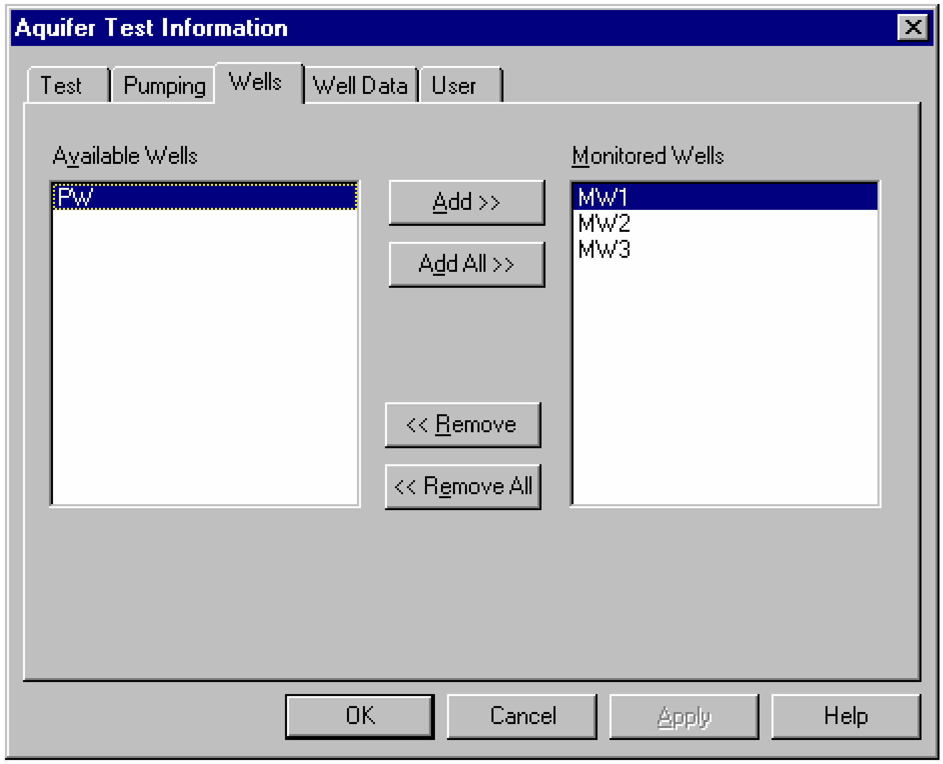
Aquifer Test

Having defined a site, the next step is to define the aquifer test. The idea behind the aquifer test is to characterize the pump test in terms of the pumping well, pumping schedule and raw monitoring well responses. These data are compiled in the *Aquifer Test Information* property sheet which is accessed via the **Edit->Aquifer Test** menu. Like the *Site Information* property sheet, the *Aquifer Test Information* property sheet has two tabs for user fields. In addition, it has three additional tabs to define the pump test that was performed. Again, this data is designed to be the raw field data with AquiferWin32 serving as the repository for the data.

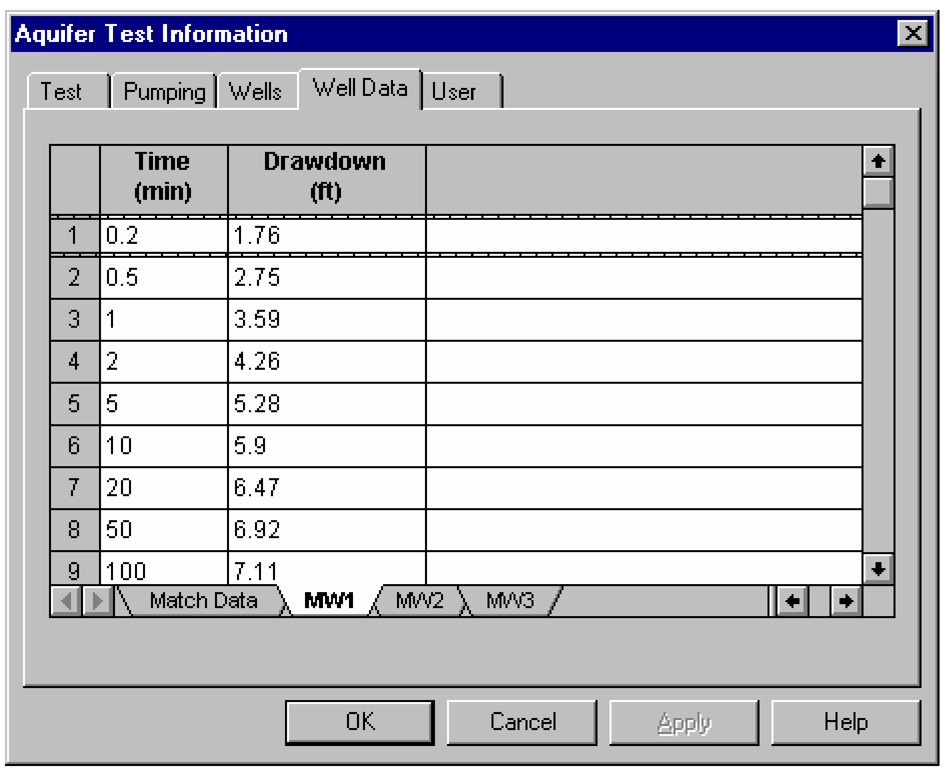
The first thing to do is define the pumping well and the pumping schedule. As shown below, the ***Pumping*** tab is used for this purpose. The *Pumping Well* combobox contains a list of all wells defined in the site. Set the selection to the well that was pumped during the test. Also, enter the pumping schedule in the *Pumping Rates* spreadsheet starting from a Time value of 0. With the exception of Step Tests, AquiferWin32 does not support variable pumping rates during the analysis phase; however, you can enter the entire pumping schedule here and decide what rate to use in the Analysis phase.



The ***Wells*** tab is used to select which wells were monitored during the pump test. Move the wells that were monitored from the *Available Wells* list to the *Monitored Wells* list using either the **Add** button or using drag and drop. Once a well has been moved to the *Monitored Wells* list, it is enhanced to include data storage for the time/drawdown data and other parameters required for analysis. A new tab is added to the spreadsheet in the ***Well Data*** tab to store the time/drawdown data. It is important to note that the process of removing a well from the *Monitored Wells* list will delete all the additional data which cannot be recovered.

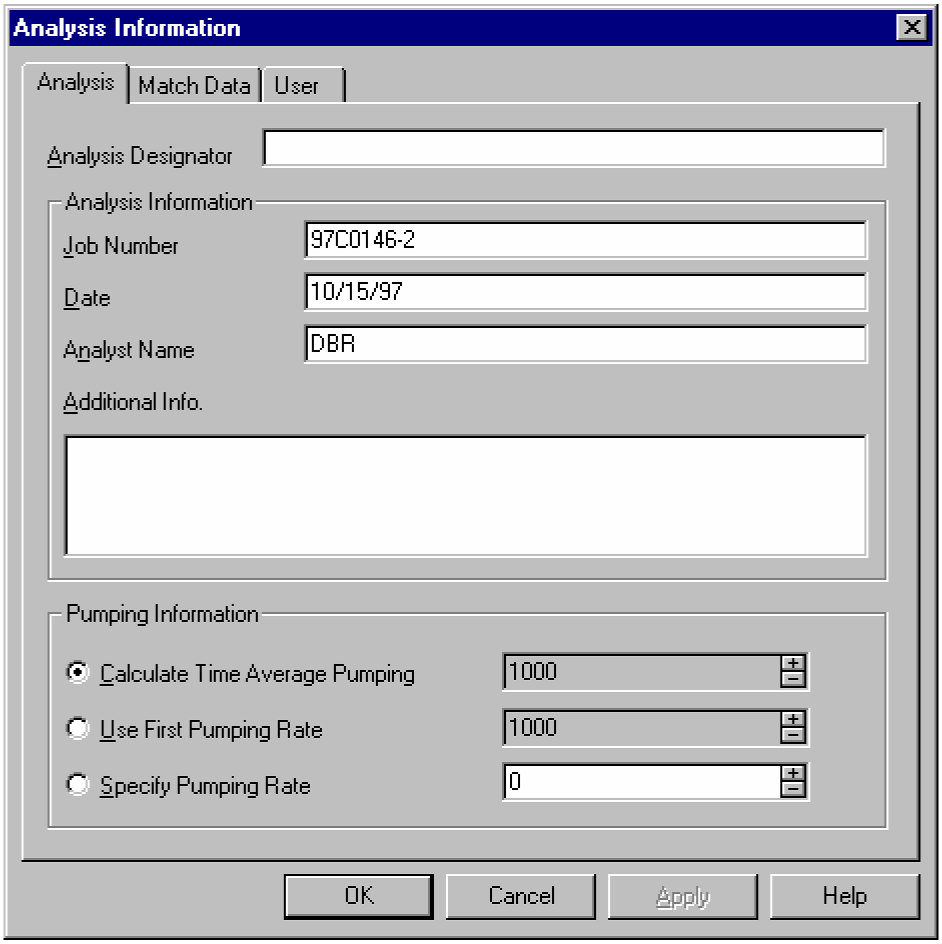


The ***Well Data*** tab is where the time/drawdown data is entered. Ignoring the ***Match Data*** tab for the moment, enter the time/drawdown data for each well by clicking on the appropriate spreadsheet tab. This is intended to be the raw data from the field which will be clipped and/or transformed in the Analysis phase. Again, by storing the raw data, you can use the Aquifer^{Win32} document as a repository for information.

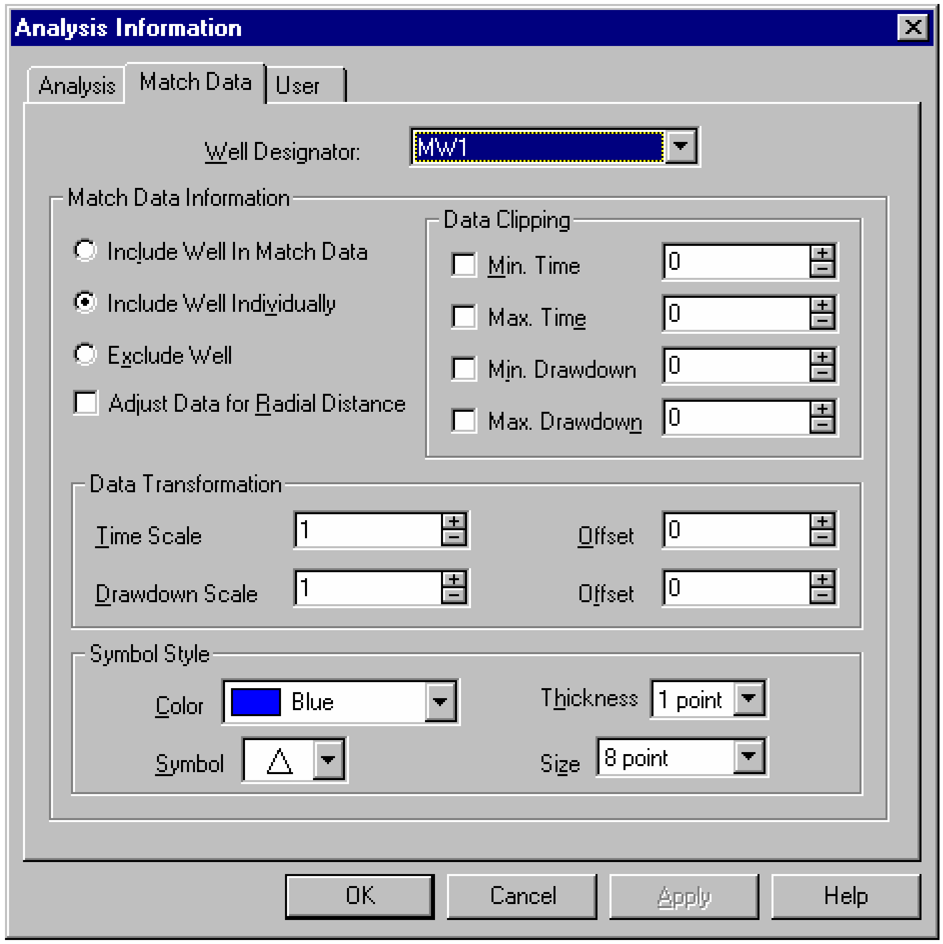


Analysis

Having defined a site and the aquifer test, the next step is to define and perform the analysis. The idea behind the analysis is to decide how you want to reduce the pump test data. The particular data to be analyzed is defined in the *Analysis Information* property sheet which is accessed via the **Edit->Analysis** menu. Like the *Site Information* property sheet and the *Aquifer Test Information* property sheet, the *Analysis Information* property sheet has user fields. In addition to the predefined user fields, the ***Analysis*** tab defines what pumping rate to use. You can use a time average pumping rate, the first pumping rate or a specific pumping rate. Using this mechanism, the entire pumping schedule has been recorded, as well as, the rate you choose to use for this particular analysis. Such information is of particular importance during the peer review process.



The ***Match Data*** tab is now used to set up the time/drawdown data for analysis. As shown below, you have many options on how to assemble the data for analysis. The *Well Designator* combobox contains all wells that were specified as monitored in the Aquifer Test phase. By default, the ***Exclude Well*** radio button is set which means that the well and its data are not to be analyzed. Set either the *Include Well In Match Data* radio button or the *Include Well Individually* radio button if you intend to analyze the data for well.



All wells that have the *Include Well In Match Data* radio button set are combined together into one data set, the Match Data. Additionally, you have the option to check the **Adjust Data for Radial Distance** check-box. If any one of the wells included in the match data set have been adjusted for radial distance, all will be. The adjustment is to divide data points by the square of the radial distance (the distance between the particular monitoring well and the pumping well). As demonstrated in the "Using Multiple Observation Wells" tutorial, a traditional approach is to include all monitored wells into one match data set and adjust for radial distance. You then match the resultant data against a Theis curve to get a "uniform" Transmissivity and Storage coefficient for the ensemble of wells. This traditional procedure was limited the Theis analysis. The tutorial goes on to demonstrate an alternative approach is to include all wells individually, don't adjust for radial distance and optimize them as a group. This yields the same result but is not limited to the Theis analysis. This is a powerful feature of Aquifer^{Win32}.

Again, setting the *Include Well Individually* radio button allows that data for the particular monitoring well to be analyzed independently or as part of a group optimize procedure. Furthermore, you can generate a match data set from any of the monitoring wells and/or include the rest individually. The user interface mechanisms to deal with them will be discussed later.

Regardless of how a well is included, the raw data that was entered as part of the Aquifer Test phase, can now be adjusted. The *Data Clipping* parameters allow you to filter out bad data and the *Data Transformation* parameters allow you to correct for errors in the raw data. It is important to note that the data clipping operation occurs after data has been transformed.

Additionally, you can and should set the *Symbol Style* parameters to differentiate data from different monitoring wells on the graphs.

User Interface Issues for Multiple Well Analyses

During the analysis of multiple wells, the user-interface has enhanced functionality. The enhancements occur in both the spreadsheet and the graph view. Each will be discussed separately along with some additional concepts.

Spreadsheet View

In the simple analysis of data, the spreadsheet view has one tab and data can be entered directly into that tab. There are four columns of data available including Time, Drawdown, Weight and Symbol; however, only two are visible by default. Entering data into these columns is equivalent to entering data into the ***Match Data*** tab of the spreadsheet on the ***Well Data*** tab of the *Aquifer Test Information* property sheet.

In a multiple well analysis, the spreadsheet functionality is somewhat changed. Now the data in ***Match Data*** tab is calculated and the Time and Drawdown columns of data should not be edited; however, the Weight and Symbol columns can be edited. If editing the raw data for the individual wells is desired, click the **View->Well Data** menu. In so doing, tabs are added to the spreadsheet for each well. These tabs can be used to edit the raw data. Once changes have been made to the raw data, click the **Calc->Match Data** menu to recalculate the data in the ***Match Data*** tab. If you edit the data using the ***Well Data*** tab of the *Aquifer Test Information* property sheet, this calculation is done automatically when you accept the property sheet.

In a multiple well analysis in which wells are included individually changes the ***Match Data*** tab of the spreadsheet. As discussed in previous sections, each well has data associated with it including raw time/drawdown data, transformed/clipped/weighted data, graph characteristics and analysis parameters. In order to manipulate the data for a specific well, you must make it active. The active well is set by toggling among the available wells using either **Ctrl-D** or the **Edit->Toggle Data Set** menu. If the active well is truly the Match Data well, generated when at least one well has been set to be included in the match data set using the ***Match Data*** tab of the *Analysis Information* property sheet, the spreadsheet and graph view tab is **Match Data**. On the other hand, if the active well is a well included individually, the tab is **Match Data – "Well Name"**. This gives the user a visual indication as to which well is active.

Another important concept in a multiple well analysis is the data weights and symbols. We have already discussed that the Time and Drawdown columns of data are calculated from the raw data. Initially, the weight is initialized to 1 and the symbol is set based on that defined in the ***Match Data*** tab of the *Analysis Information* property sheet for the specific well. During the course of analyzing the data, weights are typically changed for individual data points that are anomalous. Similarly, symbols are typically changed for individual data points to indicate those that have been weighted or ignored. If you later want to change the raw data, the data weights and symbols will be maintained unless you add or delete data points. Any change in the number of raw data points will result in a re-initialization of both weights and symbols. The weights will be reset to 1 and the symbols will be reset to the symbol indicated in the ***Match Data*** tab of the *Analysis Information* property sheet. If you change the symbol for any well in the ***Match Data*** tab of the *Analysis Information* property, you will be asked if you want to reset all symbols. This action

will reset the symbols for all wells in the analysis; however, the data weights will not be affected.

Graph View

The graph view functionality is changed when multiple wells are analyzed. As in the spreadsheet view, the ***Match Data*** tab's name is defined by the currently active well. The active well is set by toggling among the available wells using either **Ctrl-D** or the **Edit->Toggle Data Set** menu. When performing a manual match, the data points associated with the currently active well are the ones that can be moved and selecting a type curve affects only the active well. The analysis parameters associated with the currently active well are the only ones affected by such a manual match. Finally, the **Calc->Optimize** and **Calc->Reset Data Offset** only affect the active well.

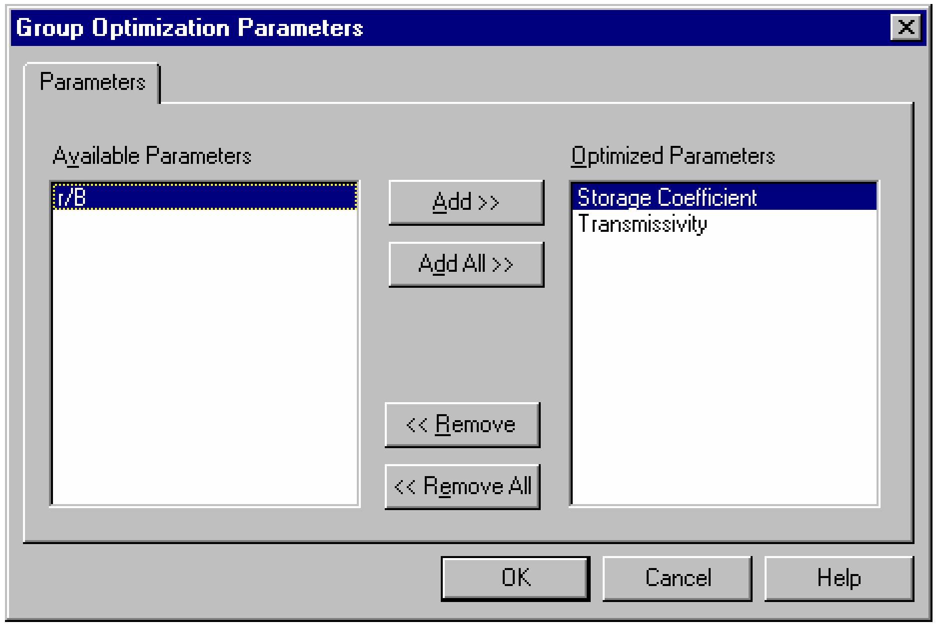
Previously, we mentioned that each well has a unique set of analysis parameters which can be accessed using the **Edit->Solution** menu. Ignoring Group Optimization for the moment, each well is analyzed independently of the rest; however, they are all presented on the same graph. If you manipulate parameters for one well, it doesn't affect the others.

It was previously discussed that the spreadsheet can display a separate tab for the raw data of each well. When selecting the raw data in the spreadsheet view, the graph view is transformed to display a graph of drawdown versus time. Like other graphs, you have full control of the display characteristics of this graph and changes to this graph do not affect other graphs. The initial characteristics of this graph were copied from the default well graph which can be set up using the **Edit->Default Well Graph** menu. The copy occurs when the well was moved from the *Available Wells* list to the *Monitored Wells* list in the ***Wells*** tab of the *Aquifer Test Information* property sheet; therefore, subsequent changes to the default well graph do not affect current wells.

Group Optimization

Group optimization represents the real power of multiple well analysis. Having gone to the trouble of setting up the data for multiple wells, you can now determine the best parameter values for the test as a whole. We have discussed that each well has a unique set of parameters that can be manipulated individually. Group optimization can be performed on all analysis parameters that are free to vary, not fixed, for all wells involved in the analysis. What group optimization does is to perform the Marquardt (modified Gauss-Newton) nonlinear least-squares method using all data from all wells to find the best parameters. If no parameters have been set to be optimized as a group, which is the default, the **Calc->Optimize Group** menu would be equivalent to toggling among each well and optimizing them individually.

Using the **Edit->Group Optimize Parameters** menu, as shown below, you can select which parameters are to be optimized across all wells. Simply move the parameters from the *Available Parameters* list to the *Optimized Parameters* list. In the specific case below for a Hantush and Jacob, 1955 (Leaky Aquifer) analysis, the parameter r/B is expected to vary among wells so it is not optimized as a group. After performing the group optimization process, you will get the best uniform value of Transmissivity and Storage Coefficient for the test. Using traditional methods, you would have to get individual values for each well and average them in some way to get an answer; the exception is a Theis response for which you could divide by the radial distance squared and match to the Theis curve to get the answer.



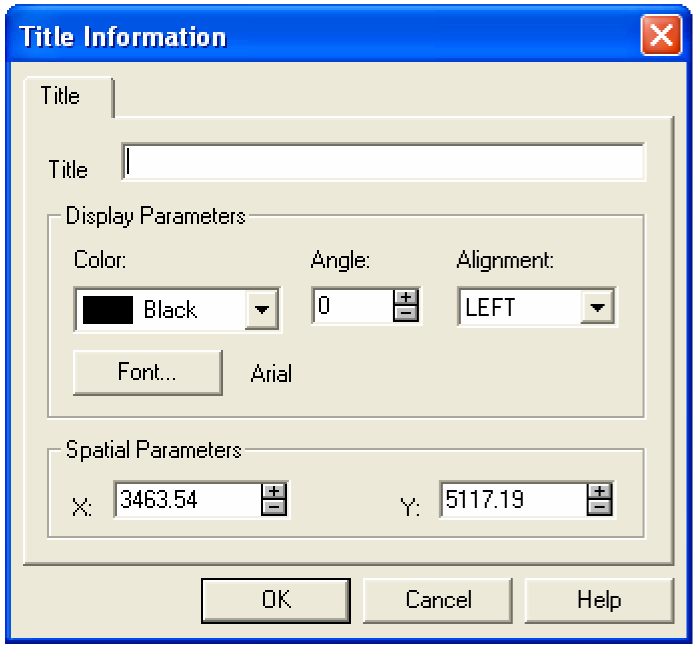
Annotations

Aquifer^{Win32} has numerous ways to enhance graph and map views with additional elements. These elements can be copied and pasted from one graph to another or from one map to another. Although each type of annotation will be discussed in subsequent sections, the characteristics common to all of them will be discussed here. Furthermore, refer to the corresponding **Add** menu description in the Menus and Dialogs chapter for details about the specific fields on the property sheets.

Spatial Parameters are common to all annotations. In graph views, the *Spatial Parameters* represent inches from the lower left corner of the view while in map views, the *Spatial Parameters* represent map units. It is important to remember that graphs are scaled to fit the views they are contained in so one inch will not necessarily represent one inch on the screen. If you print graphs with **Use Specified Dimensions** set on the ***Graphs*** tab of the *Page Setup Options* property sheet, one inch will represent one inch on the printed page.

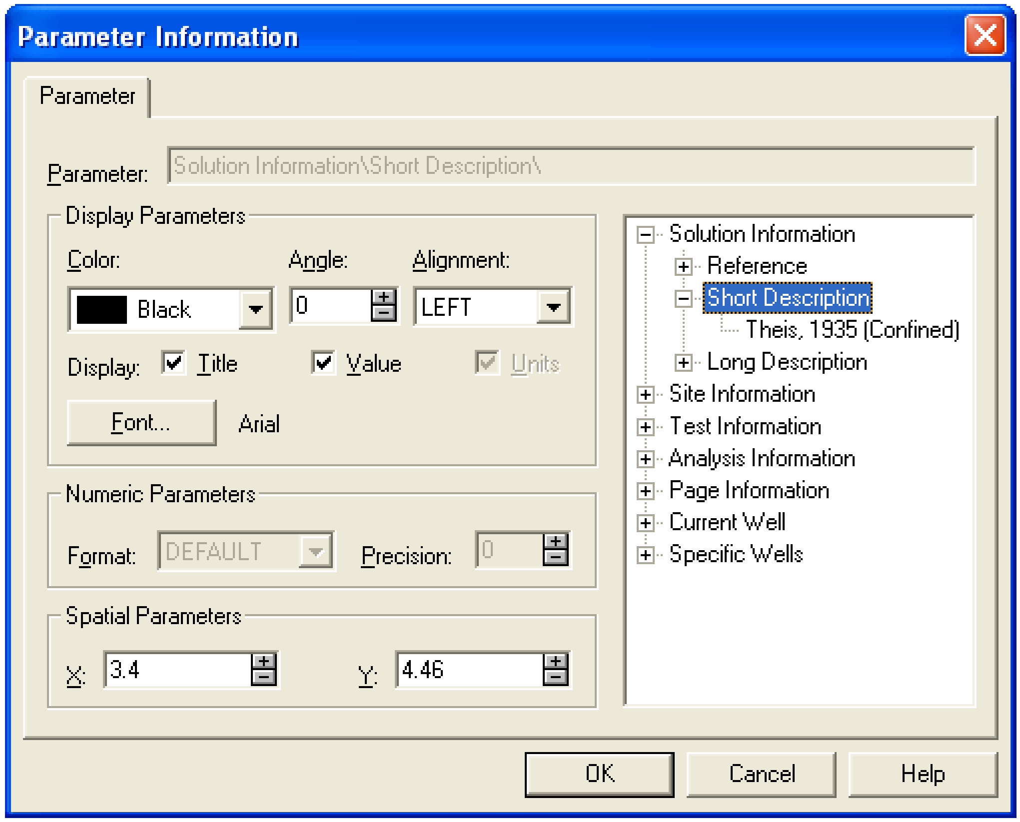
Titles

Using Title elements, you can annotate any graph or map with text items. As shown below, you can control the *Color*, *Angle*, *Alignment* and *Font* of the text.



Parameters

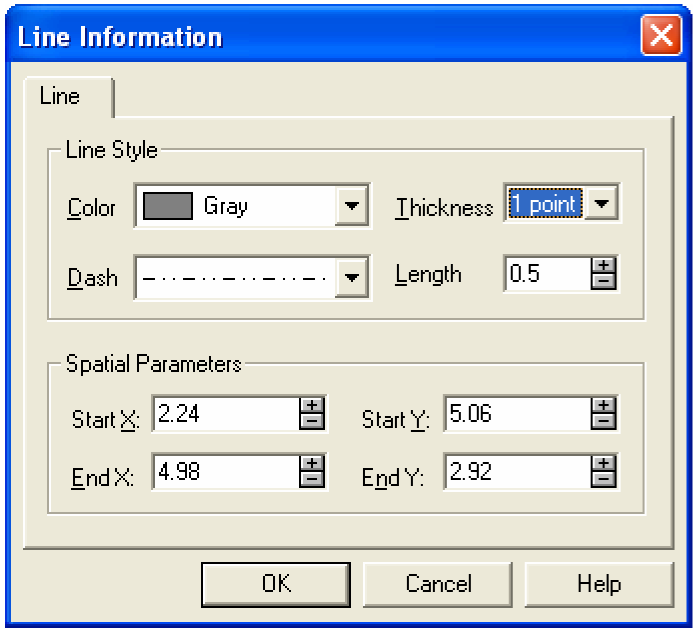
Parameter elements are similar to Title elements except that the text is referenced rather than specified. Available references include all User Fields and Analysis Parameters. The tree control is used to select the reference by drilling down into the available items and selecting one. You can select any item in the tree that has exactly one child item. The child item is the text currently associated with the selected reference. The associated text is not static and references are continuously resolved. You can control whether to display the *Title*, *Value* and/or *Units*. For numbers, you can specify the *Format* and *Precision*.



If the reference becomes invalid the parameter object is not displayed. This can happen if you reference an Analysis Parameter like r/B for Hantush and Jacob, 1955 and subsequently change to Theis, 1935. Invalid parameters are retained and will reappear when they become valid.

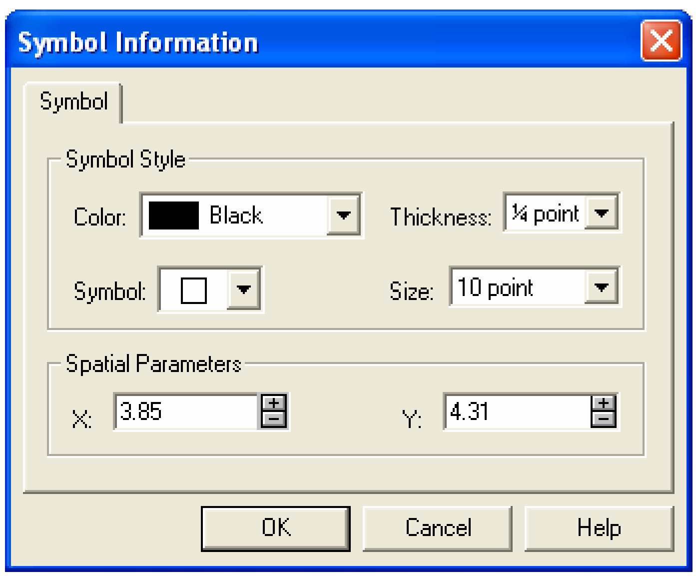
Lines

Using Line elements, you can annotate any graph or map with lines. As shown below, you can control the *Color*, *Thickness*, *Dash* and *Length* of the line. There are nine dash patterns to choose from. The *Length* parameter controls the frequency with which the dash pattern is repeated.



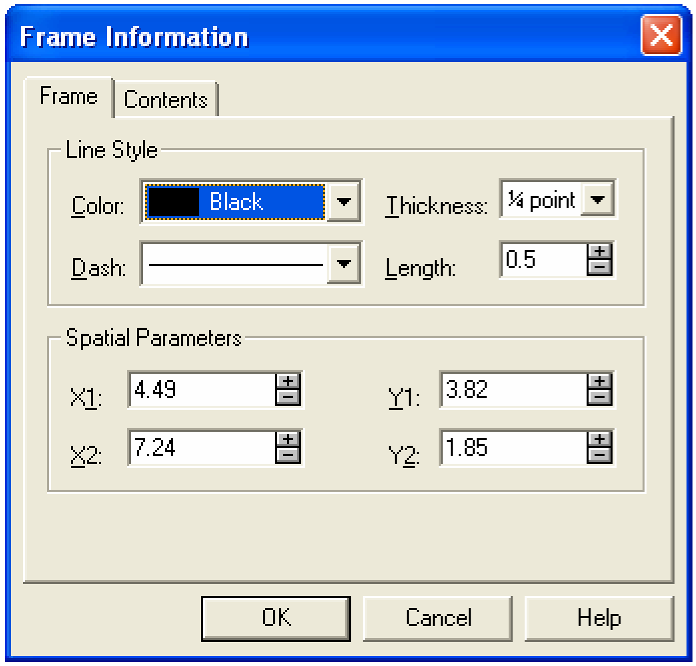
Symbols

Using Symbol elements, you can annotate any graph or map with symbols. As shown below, you can control the *Color*, *Thickness*, *Symbol* and *Size* of the symbol. There are ten symbols to choose from. The *Thickness* parameter controls the thickness of the lined use when drawing the line segments that make up the symbol.

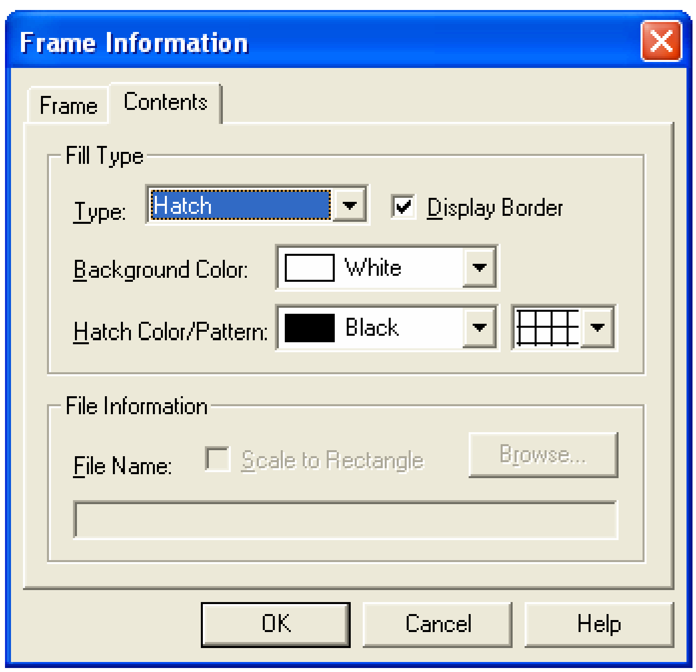


Frames

Using Frame elements, you can annotate any graph or map with rectangular frames that can be simple rectangles, contain solid colors, contain hatch patterns, contain bitmaps or contain metafiles. As shown below, you can control the *Color*, *Thickness*, *Dash* and *Length* of the line used to draw the optional border of the frame; refer to the Lines subsection for more details.



Additionally, you can control the contents of the frame element. The **Display Border** check-box controls whether the border is displayed. The *Type* combobox defines whether the frame is empty or is filled. Depending on the *Type*, you can specify the *Background Color*, *Hatch Color* and *Hatch Pattern*. In the case of Bitmaps and Metafiles, you must specify the file that contains the bitmap or metafile. Once loaded, the contents of the file is stored within the Aquifer^{Win32} document and the *File Name* field is purely for reference. For Bitmaps, the **Scale to Rectangle** check-box controls whether the bitmap is expanded or contracted to fit the rectangle; if you want the screen image and the printed image to be the same, this check-box must be checked.

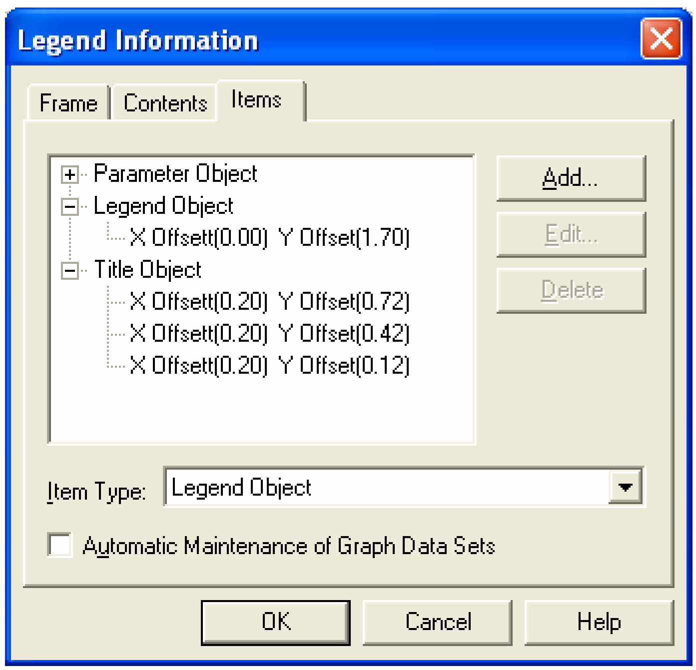


Legends

General

Legend elements are essentially enhanced Frame elements. In addition to the characteristics outlined in the previous subsection on Frames, Legends can contain additional annotations such as Titles, Parameters, Lines, Symbols, Frames and other Legends. You can nest legends as deeply as you like keeping in mind that the origin for each embedded legend, the lower left hand corner, is reset to 0,0. The scale within a legend is inherited from the parent view. Since the position of all embedded items within a legend is relative to the lower left corner of the legend, the legend can be moved and all items contained within it are moved as well.

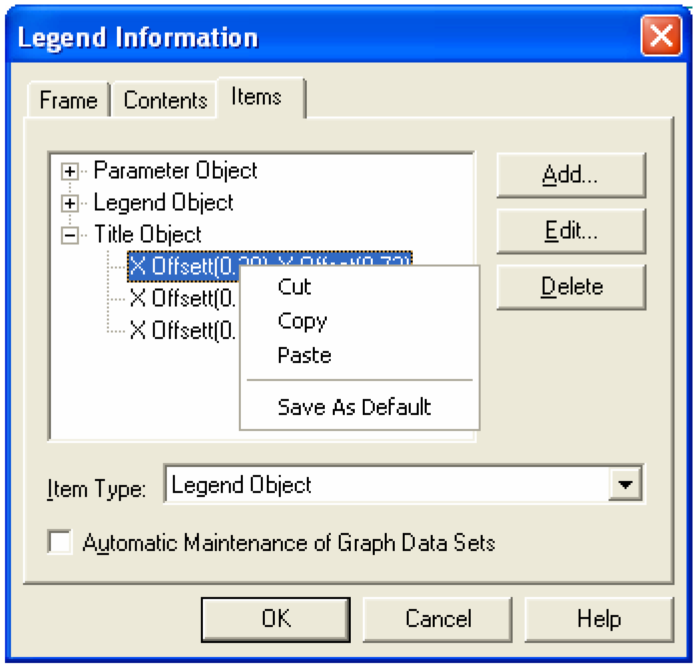
Items within legends cannot be moved and edited using the standard editing features of the view. The ***Items*** tab on the *Legend Information* property sheet is the mechanism by which they are edited. As shown below, a tree control lists all the items contained within the legend. The items are categorized based on their type and listed by X and Y Offset. To add an item, set the *Item Type* to the appropriate value and click the **Add** button. Selecting an item activates the **Edit** and **Delete** buttons. In the case of both **Add** and **Edit**, the property sheet for the item is displayed. Although legend layout can be tedious due to the lack of a graphical interface, legends provide very powerful annotation capabilities.



The **Automatic Maintenance of Graph Data Sets** check-box only appears if the parent view of the legend is a graph. If this check-box is checked, symbols and labels for each data set in the graph will be automatically displayed within the legend. If the legend is higher than it is wide, the symbol/label items will be spaced evenly vertically and centered horizontally within the legend. If the legend is wider than it is high, the symbol/label items will be spaced evenly horizontally and centered vertically within the legend. Typically, a legend set up using **Automatic Maintenance of Graph Data Sets** does not contain any other elements although it is likely embedded in another legend element.

Default Legends

A context menu (accessed via clicking the right mouse button or using the **Shift-F10 keys**) exists on the *Legend Information* property sheet to Cut/Copy/Paste individual legend items; in addition, there is a **Save As Default** menu item. If you desire to define a legend to be used as the default when a particular analysis is created, use this feature save a legend as the default. Control of whether the default legend is displayed is achieved using the **View->Default Legend** menu. Default legends are stored on the computer containing the software. If the document is moved to a different computer different default legends will be displayed. If you want to save the default legend with the document on any system, click on the legend to select it, click the **Edit->Copy** menu, click the **View->Default Legend** menu to turn off the default legend and click the **Edit->Paste** menu and save the document. Now the legend is part of the document.



Default legends are associated with each supported solution/view combination and enable the user to set the legend that is displayed when new documents are created. The default legends are saved as files in the Defaults directory with the file name being inherited from the file name of the active analysis dynamic link library and the following extensions:

.prl	Predicted View
.stl	Step Test Step 2 View
.spl	Step Test Special View
.wel	Well View
.mal	Curve Match View
.mdl	Flow Model Main View
.sml	Simulation Main View
.stl	Analysis Site View
.cnl	Concentration Graph View

.lcl   Line Calculation Graph View

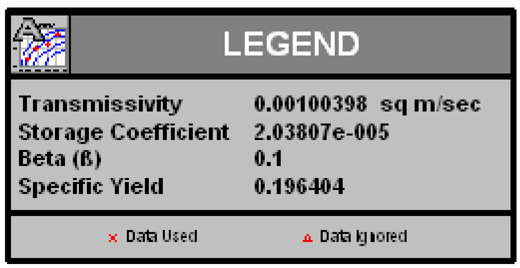
As an example, the default legend for the Curve Match View for Theis, 1935 would be saved as Anal0005.mal.

Wizard

One of the most common support issues has been setting up legends and we have attempted to make this process much easier. In Version 3, adding a legend is as simple as clicking the mouse in the view, clicking the **Add->Legend** menu and dragging a rectangle. The following wizard is activated to help in setting up the wizard. If you simply click the finish button, a legend will be created containing the pertinent parameters and will be located in the lower right corner of the graph or map.

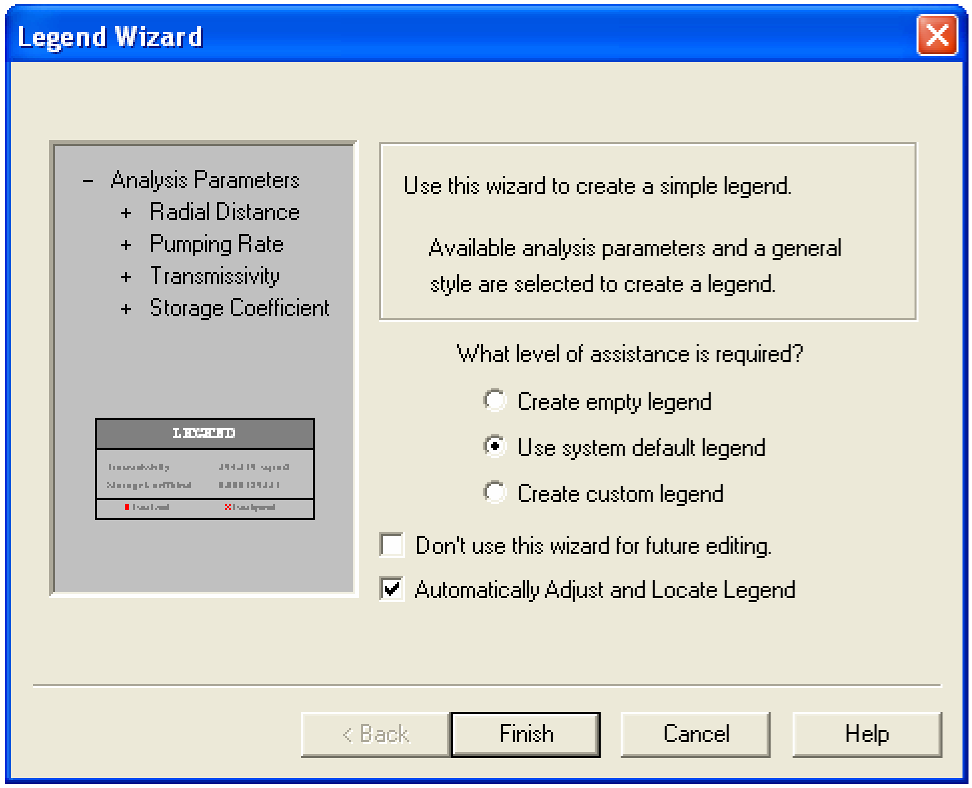
To make a legend as in previous versions, select the **Create empty legend** radio button and click the **Finish** button; double click on the newly created legend box and set it up as you see fit.

The legend generated by the wizard was patterned after the example legends created in the example files shipped with previous versions of the program. The specific legend created using the following example wizard screens is below.

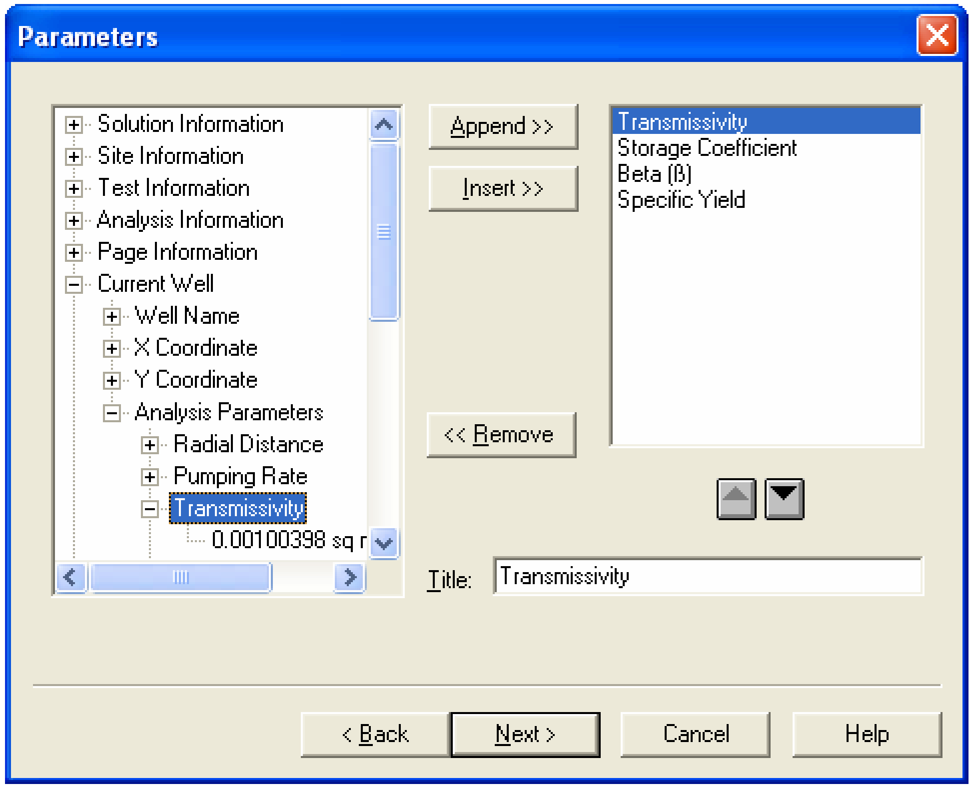


The legend has three sections, the Title sublegend, the Parameter sublegend and the Symbol sublegend. The Title and Symbol sublegends are optional as is the bitmap in the Title sublegend.

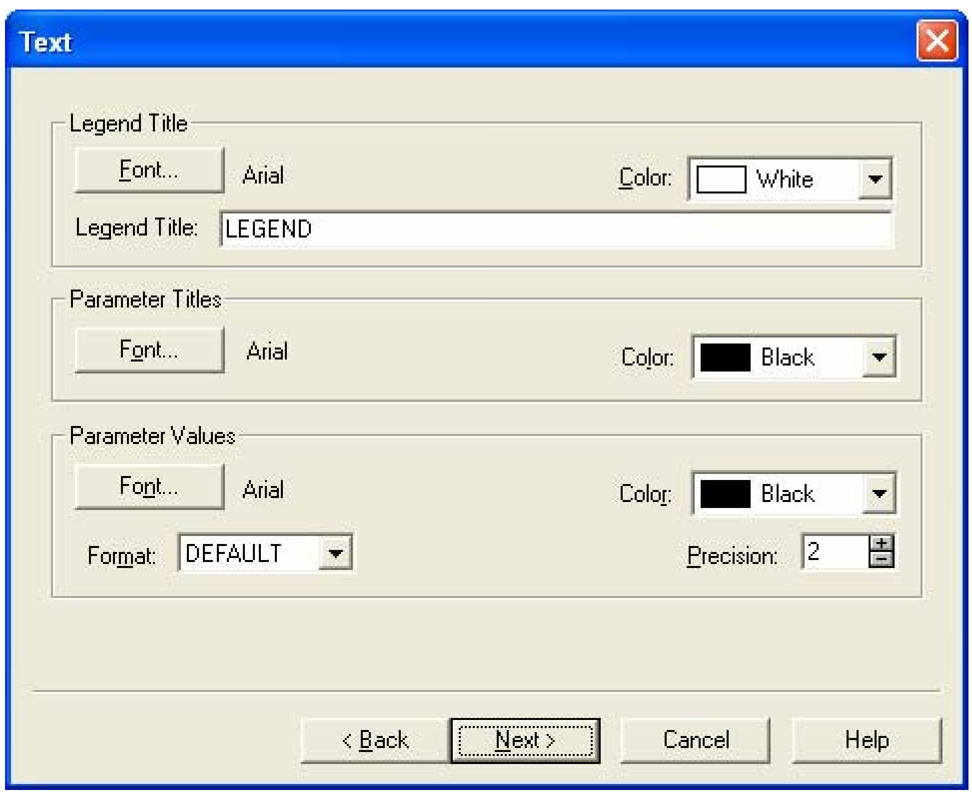
To customize the legend created with the wizard, click the **Create custom legend** radio button and the **Finish** button becomes a **Next** button to continue the customization. Two other options are available via check boxes. The **Don't use this wizard for future editing** option overrides the default behavior that, when you double click the legend to edit it, the wizard will not be used. The **Automatically Adjust and Locate Legend** check box controls whether the system will automatically maintain the location of the legend.



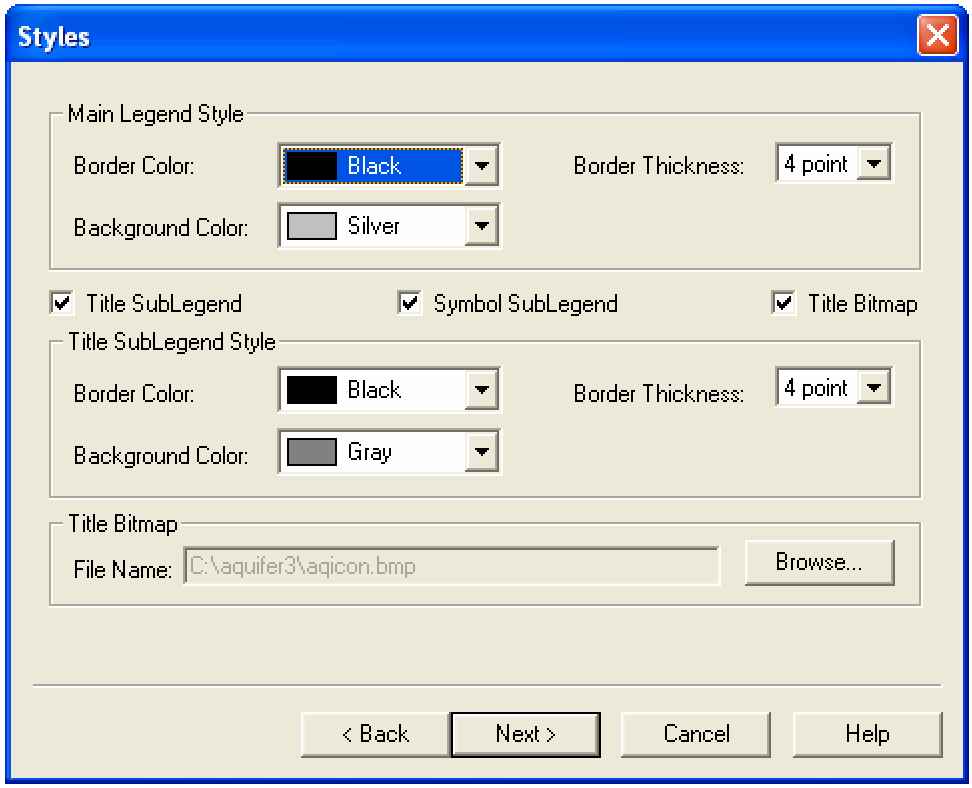
If you chose to create a custom legend, the next step in the wizard enables you to choose the items you want added to the legend. The ***Parameters*** step presents a list of parameter items which is dynamic and defined by the active document type, analysis type and view type. As with Parameters in previous versions, you drill down into the options and click on the item you want, in this case Transmissivity. Using the **Append**, **Insert** and **Remove** buttons, you select what you want. You can also change the default title for an item by selecting it in the right list box and entering the new title in the **Title** edit field.



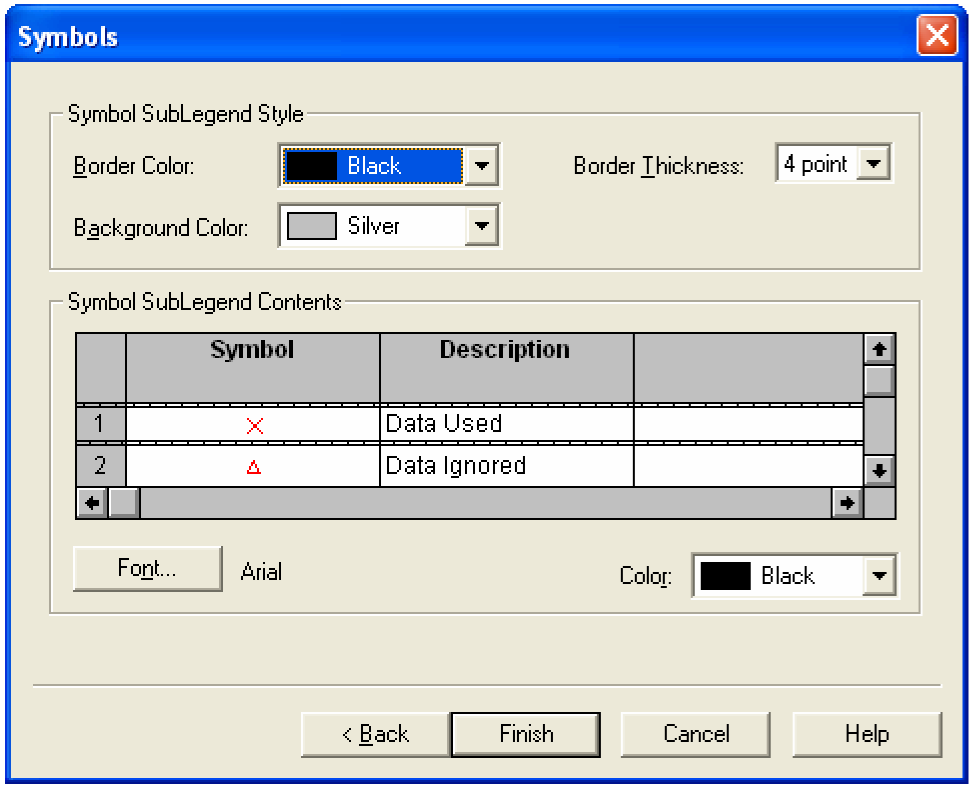
The ***Text*** step of the process controls colors and fonts of the legend title, parameter titles and parameter values. It also controls the overall format and precision for parameter values.



The ***Styles*** step defines the layout of the legend including sublegends, border colors and thicknesses, background colors and title bitmap.



The ***Symbols*** step is only displayed when the **Symbol Sublegend** checkbox has been checked in the previous step. You can control the contents of the symbol sublegend.



On Screen Editing of Legends

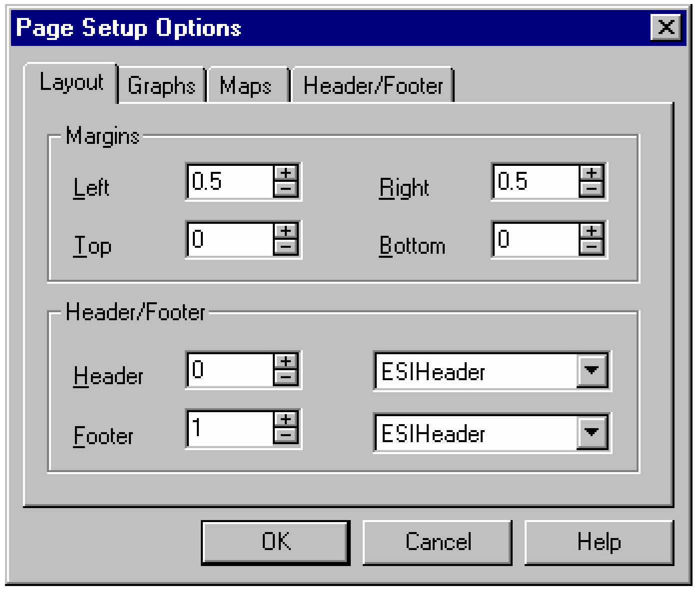
If you have created a legend using the legend wizard and have selected the **Automatically adjust and locate legend** option any changes made on screen will be automatically undone when the screen refreshes. The on screen editing features are for older legends, those created as empty legends using the wizard or those that have selected the **Don't use this wizard for future editing** option.

On screen manipulation now drills into legends no matter how deeply they are nested to allow the typical on screen manipulations to be performed. In addition, a context menu has been created for legends that allows for Cut/Copy/Paste/Select All/Delete/Add/Add to Legend operations within the legend.

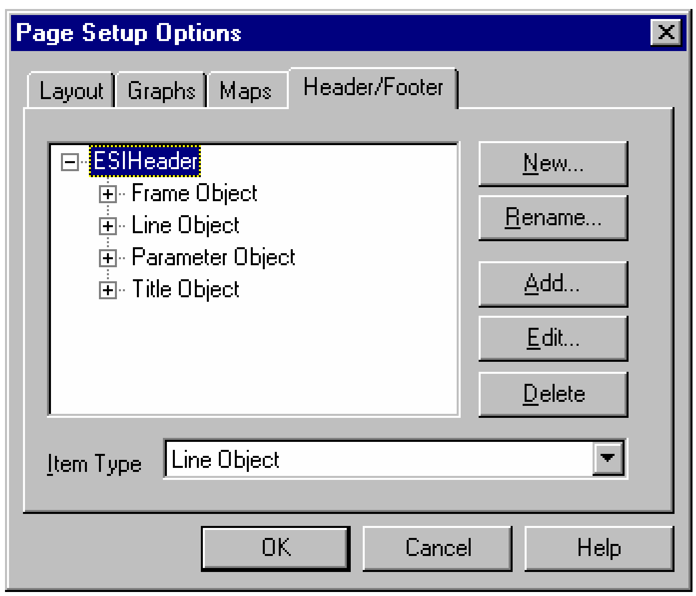
Of particular note is the **Add to Legend** menu. When items have been selected in the main view, they can be added to the legend using this menu. In most cases, you will want to drag the object into the legend and position it before using this option. You can, however, attach items to the legend that are not located inside the legend. When the legend is moved, they maintain their relative position to the legend.

Headers/Footers

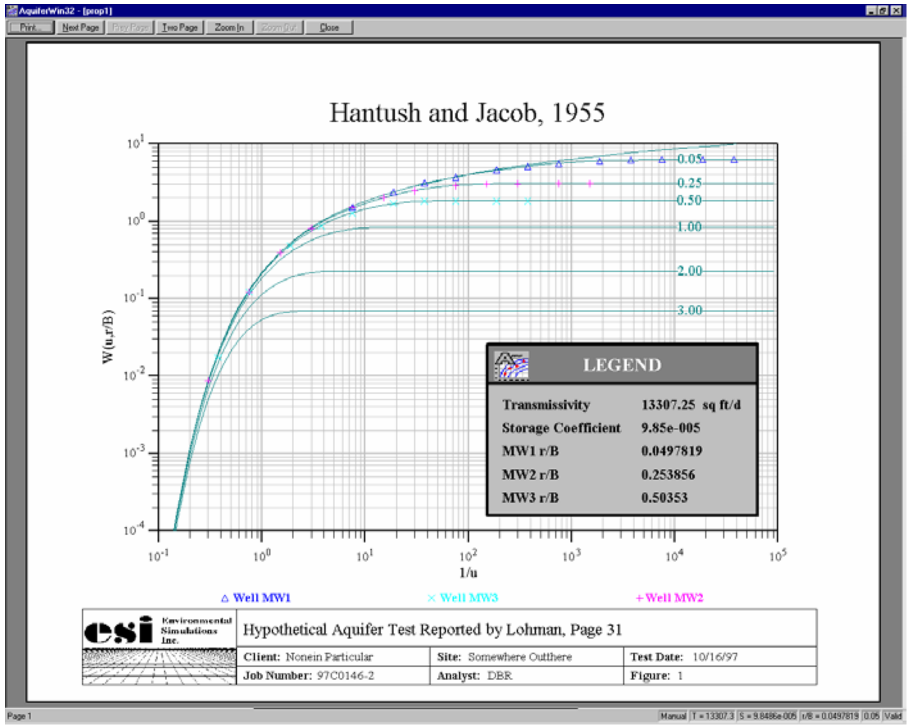
Headers and Footers are special cases of Legends that only appear on printed output. As shown below in the *Page Setup Options* property sheet, the printed output can have margins and space can be reserved within the margins for a Header and/or a Footer. The user specifies the height of the Header and/or Footer in inches and selects one of the available Header/Footer elements in the adjacent combobox.

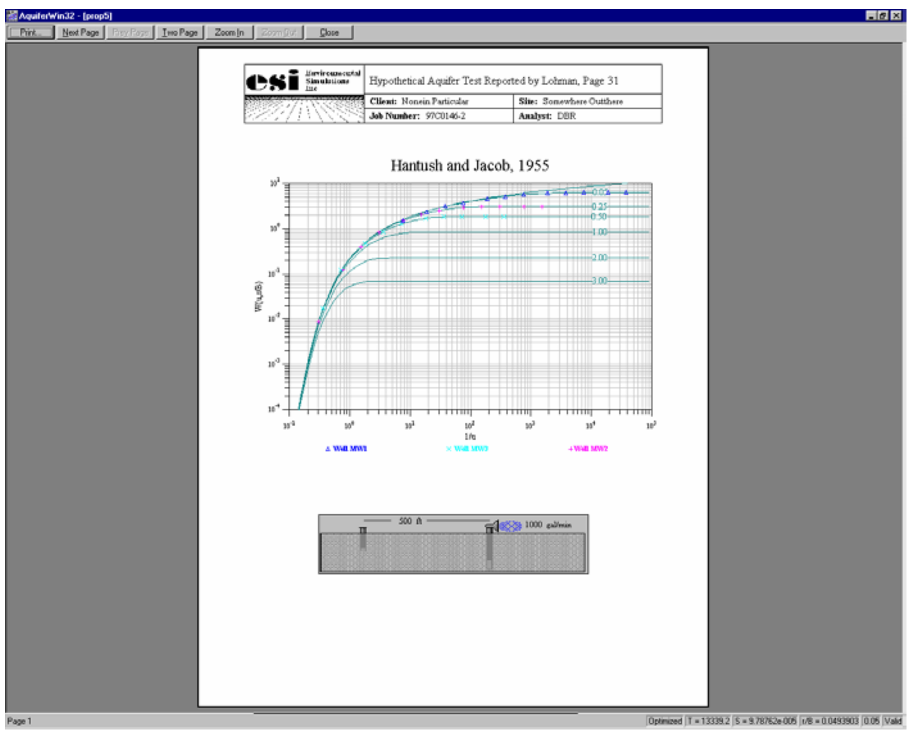


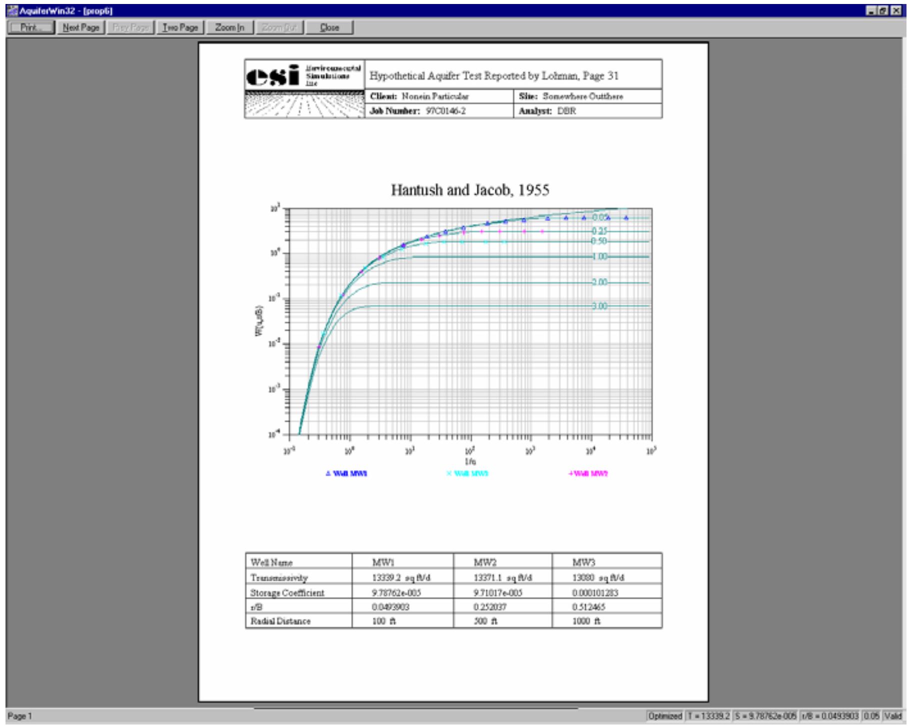
The ***Header/Footer*** tab of the *Page Setup Options* property sheet is very similar to the **Items** tab of the *Legend Information* property sheet and the user should refer to the Legends subsection of the Annotations section for details on how to layout the contents of the Header/Footer. The **New** and **Rename** buttons are used to create and rename the available Header/Footer objects. Additionally, a context menu is available to Cut/Copy/Paste header/footer objects to move them from one document to another.



Although header/footer layout can be tedious due to the lack of a graphical interface, header/footer objects provide very powerful annotation capabilities for the printed output. You can produce very diverse results and three examples are shown below that present the results of the same analysis using the **File->Print Preview** menu.

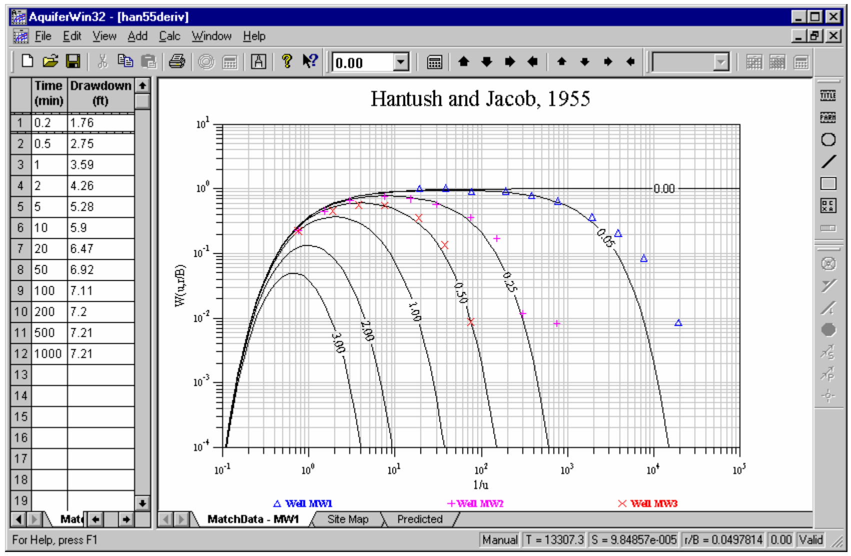

Hantush and Jacob, 1955
W(u,r/B)
1/u
LEGEND
Transmissivity 13307.25 sq ft/d
Storage Coefficient 9.85e-005
MW1 r/B 0.0497819
MW2 r/B 0.253856
MW3 r/B 0.50353
Well MW1
Well MW3
Well MW2
Hypothetical Aquifer Test Reported by Lohman, Page 31
Client: Nonein Particular
Site: Somewhere Outthere
Test Date: 10/16/97
Job Number: 97C0146-2
Analyst: DBR
Figure: 1

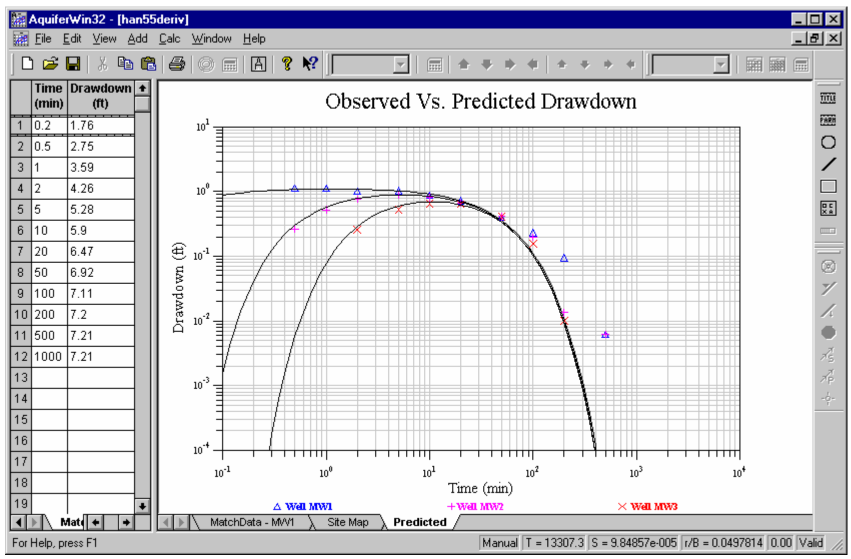

Hypothetical Aquifer Test Reported by Lohman, Page 31
Client: Nonein Particular
Site: Somewhere Outthere
Job Number: 97C0146-2
Analyst: DBR
Hantush and Jacob, 1955



Derivative Analysis

Analysis of pump test data can be greatly improved using a plot know as a derivative plot. Although rather simple in concept, such a plot can be very instrumental in accurately reducing the data from a pump test. It is beyond the scope of this manual to go into the theory of analyzing a derivative plot and the user is referred to texts like Horne, 1995 and journal articles like Spane & Wurstner, 1993 for the details. Suffice it to say that the characteristics of curves representing the first-order pressure derivative versus time can be more distinctive than the traditional type curves. The difference is the result of the sensitivity of the derivative to small variations in the pressure change that occurs during a pump test. This sensitivity can be used to identify wellbore storage effects, boundary effects and the establishment of radial flow conditions. The following two graphs demonstrate the marked difference of the derivative plot from the traditional leaky-confined type curves that flatten out with time.





The implementation of the derivative analysis within Aquifer^{Win32} was adapted from two sources. The first-order pressure derivative of the data is performed as per Spane & Wurstner, 1993 and the algorithm from their DERIV program was adapted. As indicated in their paper, the differential algorithm is based on the preferred algorithm listed by Bourdet et al., 1989; the algorithm calculates the first derivative of the pressure change with respect to the natural logarithm of the change of time. Two options are available for calculating the data slopes before and after a given point, LEAST SQUARES and FIXED ENDPOINT. The fixed endpoint uses the points predeeding and following the point of interest by the specified distance along the x axis. The least squares regression option uses all the points preceeding and following the point of interest within the specified distance in the calculation.

Spane & Wurstner, 1993 recommend the fixed end point options for data from published type curves or data devoid of significant noise. For noisy test data the least squares option is preferred. In Aquifer^{Win32} the least squares option is the default.

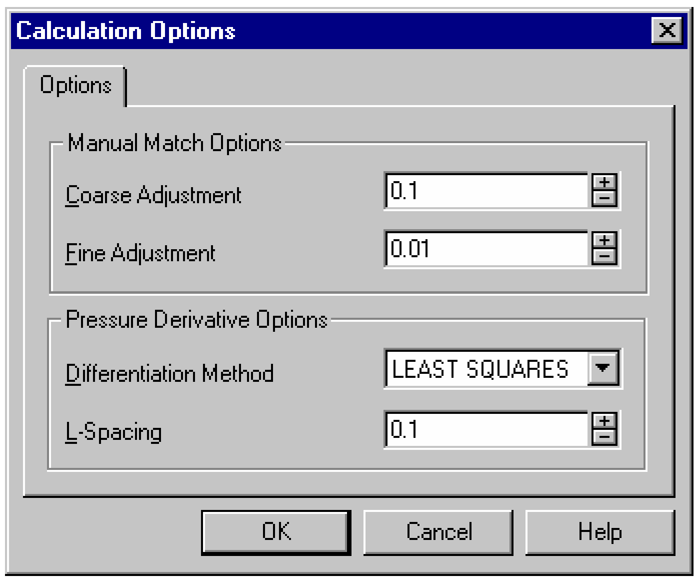
The distance along the x-axis to use in the aforementioned calculations is referred to as the L-Spacing. The L-Spacing ranges from 0 to .5 in which 0 uses the points immediately adjacent to the point of interest. Values greater than .2 smooth out noisy data but can also cause a loss of resolution.

Since Aquifer[Win32] directly calculates the values for type curves, the presure derivatives are directly calculated using the equation presented by Horne, 1995.

$$t(\frac{\partial p}{\partial t})_i = (\frac{\partial p}{\partial \ln t})_i = \frac{\ln(t_i / t_{i-k})\Delta p_{i+j}}{\ln(t_{i+j}/t_i)\ln(t_{i+j}/t_{i-k})} + \frac{\ln(t_{i+j}t_{i-k}/t_i^2)\Delta p_i}{\ln(t_{i+j}/t_i)\ln(t_i/t_{i-k})} - \frac{\ln(t_{i+j}/t_i)\Delta p_{i-k}}{\ln(t_i/t_{i-k})\ln(t_{i+j}/t_{i-k})}$$

In the above equation, the differentiation interval or L-Spacing is used to be consistent with the derivatives of the data. In the special case where the L-Spacing has been set to 0 to cause the adjacent data points to be used, the type curves will be generated using an L-Spacing of .1.

The *Calculation Options* property sheet used to control the differentiation method and l-spacing is accessed using the **Calc->Options** menu and is displayed below.



Manual Match Options

Coarse Adjustment	The distance in inches to move the data relative to the type curves when performing a coarse adjustment
Fine Adjustment	The distance in inches to move the data relative to the type curves when performing a fine adjustment

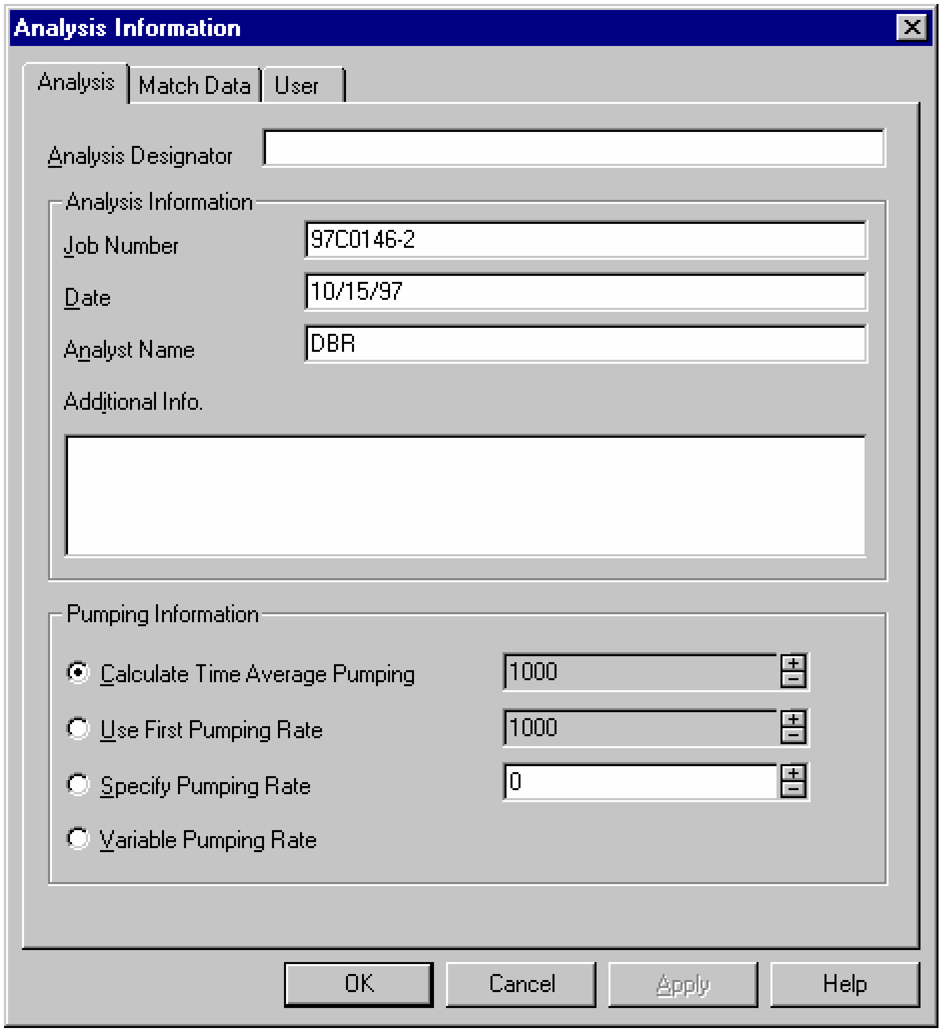
Pressure Derivative Options

Differentiation Method	Specifies the calculation method to use when calculating derivatives
LEAST SQUARES	See discussion above
FIXED ENDPOINT	See discussion above
L-Spacing	Specifies the interval over which data and type curve derivatives are calculated

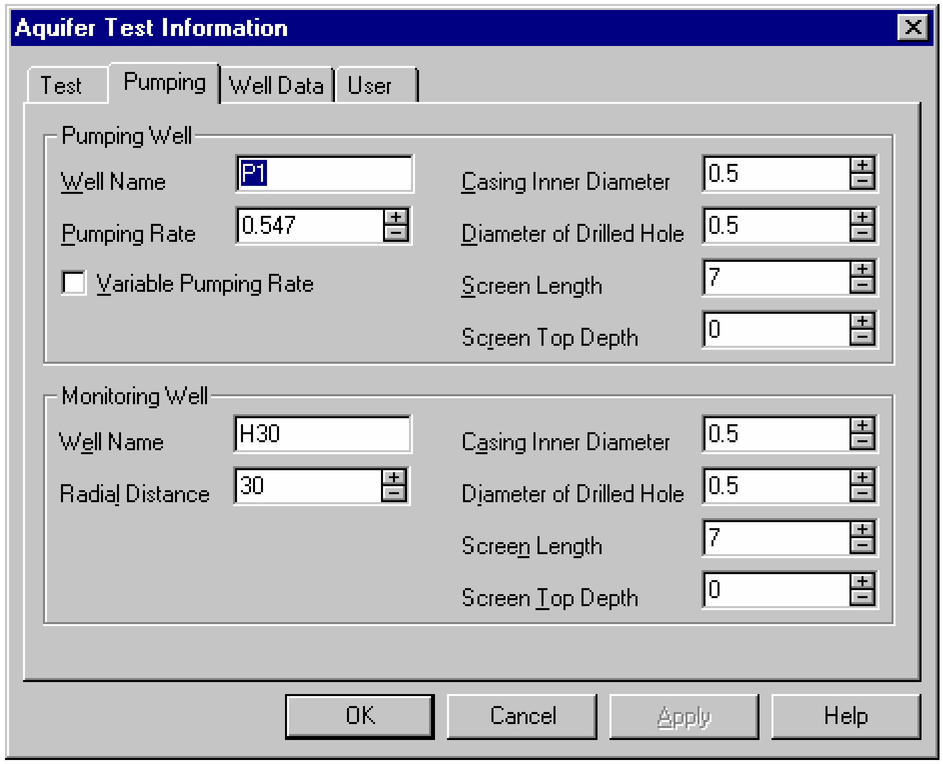
Variable Pumping

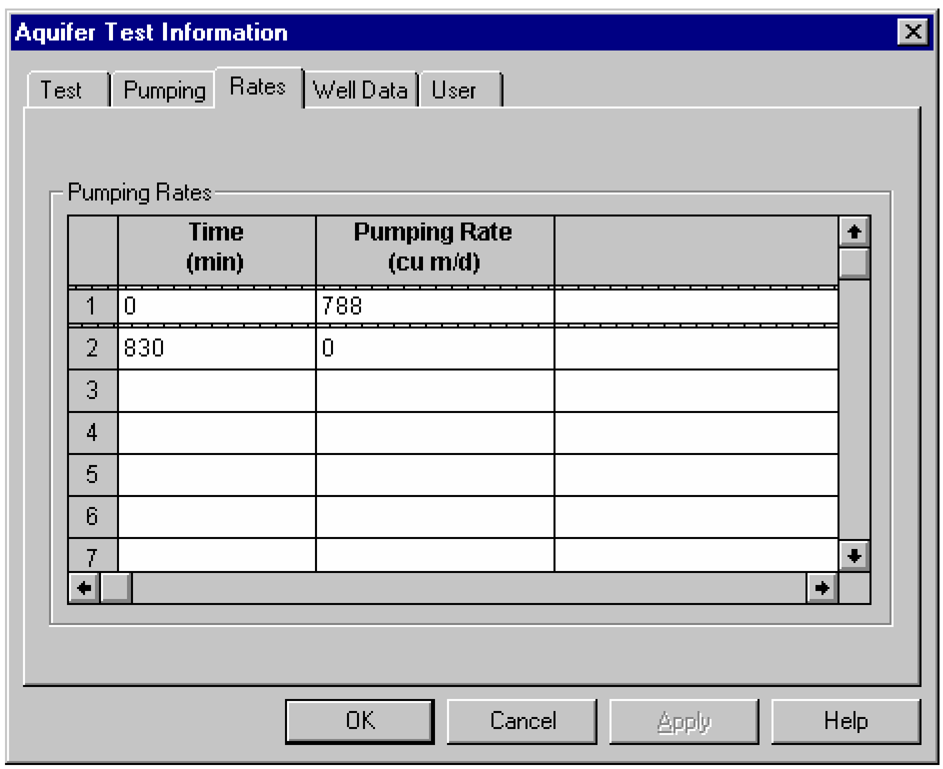
Variable Pumping

Aquifer^{Win32} has always provided a mechanism to store the full pumping schedule for a given pump test when wells have been defined. Now variable pumping schedules can be directly used in the analysis of the pump test. Although this represents a major advancement to the functionality of the program, the only user interface change is the addition of the *Variable Pumping Rate* radio button on the ***Analysis*** tab of the *Analysis Information* property sheet shown below.



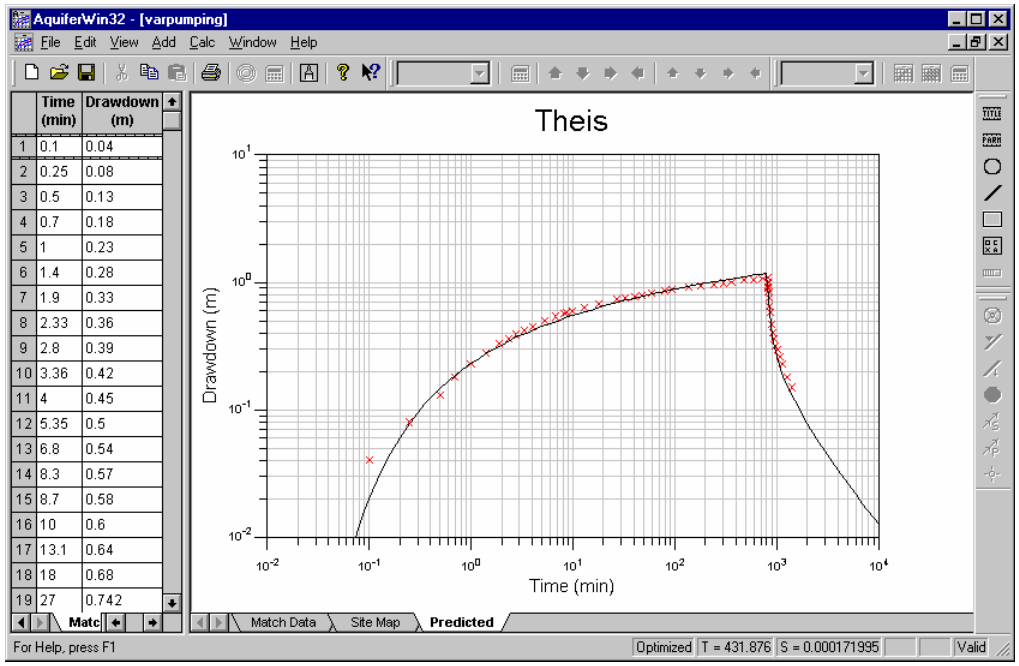
For simple solutions not involving wells defined in the site map, a very simple change has been made to support variable pumping rates. By clicking the "*Variable Pumping Rate*" checkbox, an additional tab, ***Rates***, is added to the *Aquifer Test Information* property sheet. The pumping rates can be added to spreadsheet in the ***Rates*** tab.





Once activated by either method, variable pumping rates will be taken into account in both the Marquardt nonlinear least-squares solution and in the calculation of predicted drawdown. Since traditional curve matching techniques and the type curves they employ do not take into account variable pumping rates, the type curve will not reflect changes in pumping rates; however, it is still valid to match the data from the first stage of pumping to the type curves. The resultant parameters are valid

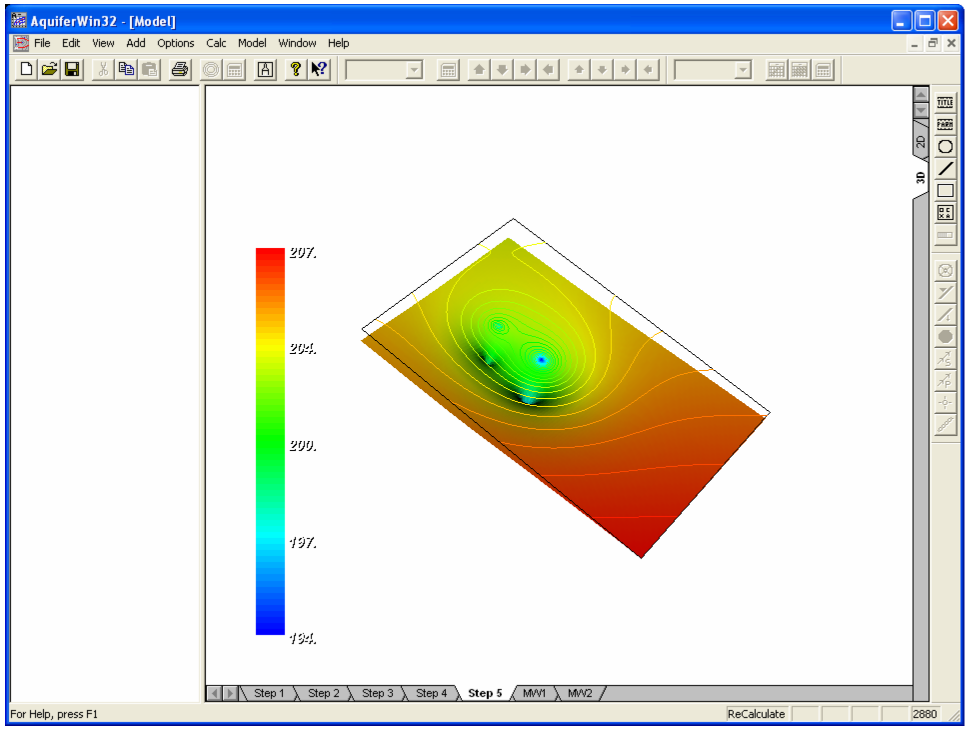
and serve as the best method to generating the initial estimates for parameters to be used during optimization. In the case of a variable pumping rate analysis, the "goodness of fit" is best viewed in ***Predicted*** view tab as shown below.



3D Perspective View

Contents

The 3D vertical tab contains the 3D perspective view of the hydraulic head or drawdown values. Contour lines can be optionally displayed as well. All the normal annotations can be added; however, the annotations are fixed in position in the view and are not moved/rotated when the 3D perspective is manipulated.



.

View Manipulation

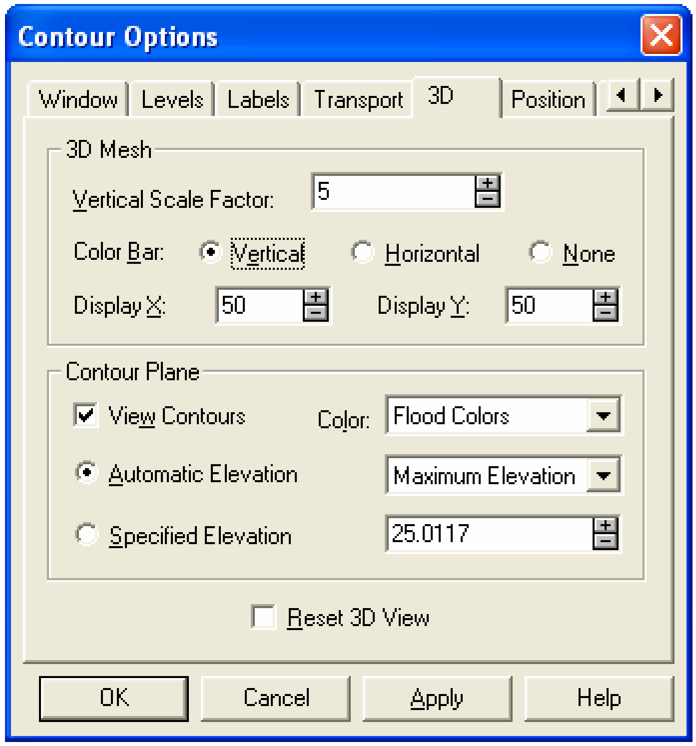
Selecting the **3D Manipulation** menu from the edit field context menu allows the manipulation of the 3D view using the mouse and keyboard. Clicking and/or holding the left mouse button in the view causes the display to rotate in the direction toward the cursor location. Holding the **Shift** button while clicking and/or holding the left mouse button causes the display to move in the direction toward the cursor location.

Clicking and/or holding the right mouse button causes the display to zoom in and zoom out. If the cursor is in the upper half of the view, it will zoom out. If the cursor is in the lower half of the view, it will zoom in. The amount of the zoom is controlled by how far the cursor is from the vertical center of the view.

The **Reset 3D** menu is used recenter the 3D perspective view. This is sometimes required when changes have been made via the Contour Options property sheet or on screen editing.

Contour Options

Four tabs on the Contour Options property sheet apply to the 3D view and are described below. The color flood parameters apply to both the contour view and the 3D view.

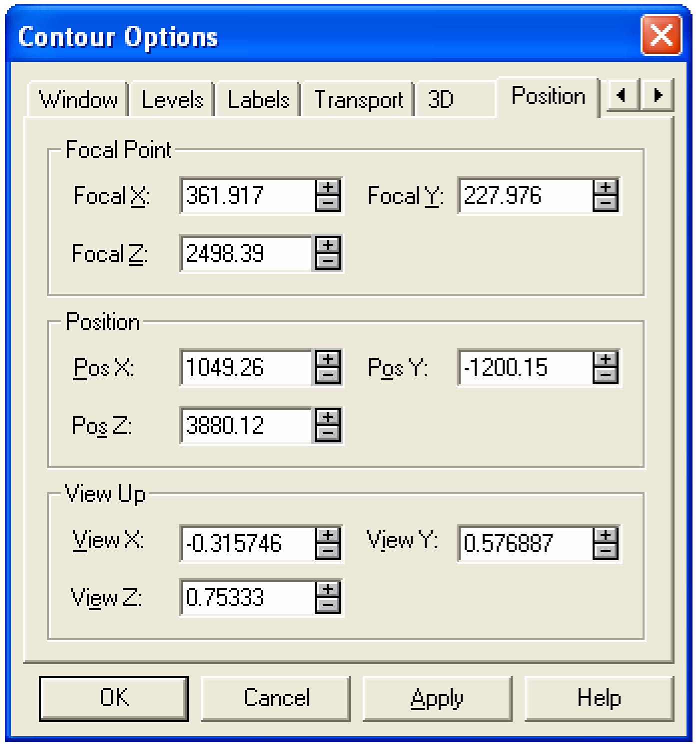


3D Mesh

Vertical Scale Factor:	A multiplier applied to head/drawdown to increase the vertical scale relative to the horizontal scale
Color Bar:	Controls the presence and location of the default scale bar
Display X:	The x-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar
Display Y:	The y-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar

Contour Plane

View Contours:	When checked, a contour map is displayed in the 3D perspective view
Color:	Controls the color of the contours which can be either Black or Flood Colors
Automatic Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked the contour map will automatically be relocated as the contour data changes
Specified Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked, the value entered in the adjacent edit field will be used to locate the contour plane
Reset 3D View	At times, the perspective drawing can leave the field of view. Checking this option will relocate it into view



Focal Point

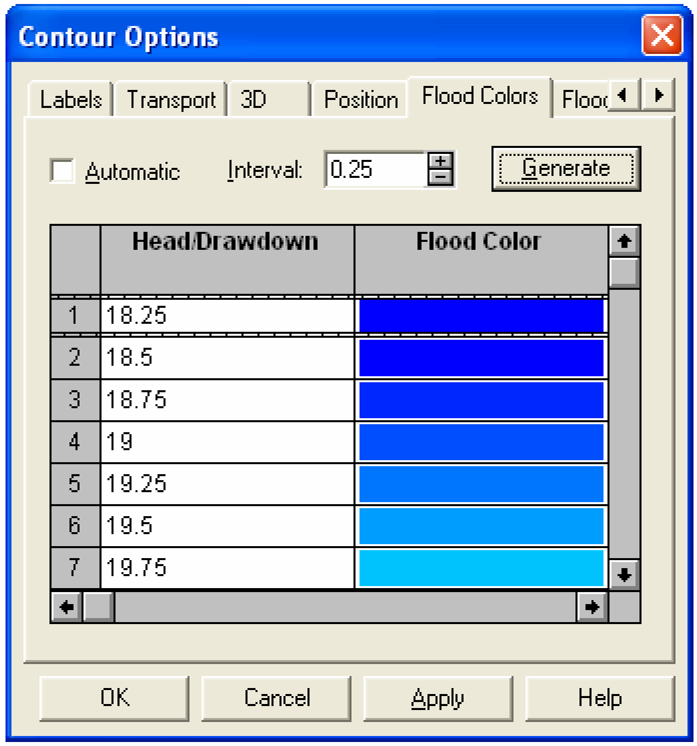
Focal X:	The x-coordinate of the focal point used to display the 3D perspective drawing
Focal Y:	The y-coordinate of the focal point used to display the 3D perspective drawing
Focal Z:	The z-coordinate of the focal point used to display the 3D perspective drawing

Position

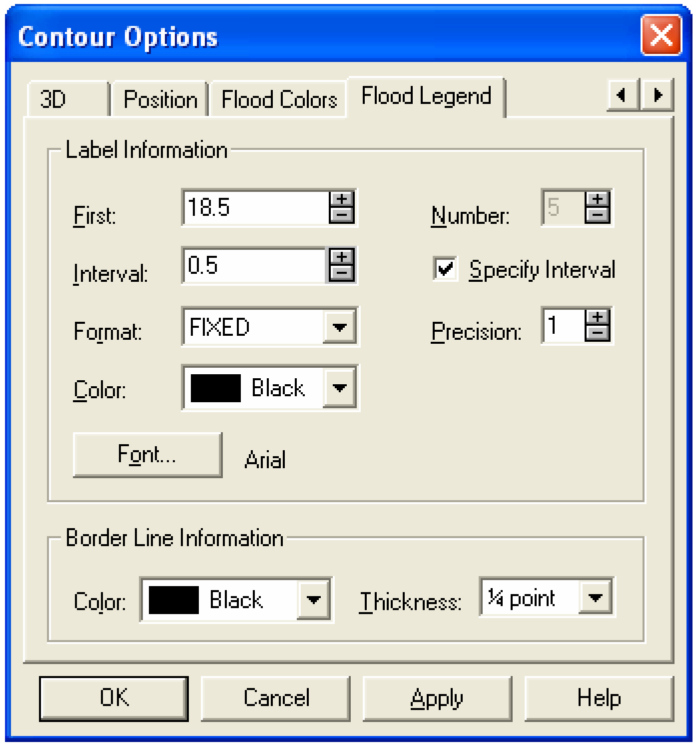
Position X:	The x-coordinate of the camera position used to display the 3D perspective drawing
Position Y:	The y-coordinate of the camera position used to display the 3D perspective drawing
Position Z:	The z-coordinate of the camera position used to display the 3D perspective drawing

View Up

View X:	The x-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Y:	The y-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Z:	The z-coordinate defining the view up direction for the camera used to display the 3D perspective drawing



Automatic	When checked, the distribution of colors is automatically determined by the application
Interval:	The interval to use when generating flood colors for values of head/drawdown to be used when the **Generate** button is clicked.
Generate	When clicked, a new color distribution will be generated using the same algorithm as the **Automatic** setting; however, these can be edited by the user



Label Information

First	Sets the head/drawdown value for the first labeled value in color flood legends
Number:	The number of equally spaced labels to use when labeling the color flood legends
Interval:	The interval between adjacent labels; used when the Specify Interval option is activated
Specify Interval	When checked, the value for *Interval* is used when labeling color flood legends
Format:	The numeric format to use when displaying the labels on color flood legends
Precision:	The number of digits to the right of the decimal point to use when displaying the label on color flood legends
Color	The color to use when displaying labels on color flood legends
Font	Defines the font, font style and size for the label

Border Line Information

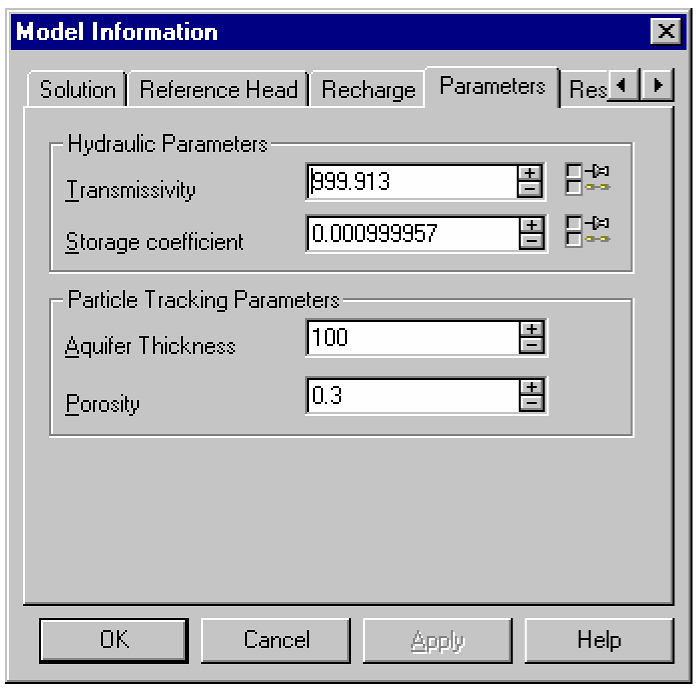
Color:	The color to use when displaying the border around the color flood legend
Thickness:	The thickness, in points, of the line used when displaying the border around the color flood legend

Model Parameters

A model parameter is a data value that is either required for or calculated by the currently selected model. Parameters can be numeric values such as *Transmissivity*, *Storage coefficient* and *Aquifer Thickness*. They can also be enumerated (specific values from a predefined list) such as *Block Type*. Since model parameters are the central theme to any modeling application, it is essential to understand how to set and manipulate them.

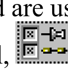
A list containing all parameters for those models supported by WinFlow could be large. Obviously, the user desiring to perform a simple Theis-based model should be sheltered from the other extraneous parameters in order to avoid confusion. Further, this application will routinely undergo enhancement to include additional models; in a typical application, this would exacerbate the problem by adding additional parameters over time. Our solution to the problem is to encapsulate each model into a separate Dynamic Link Library and use the *Model Information* property sheet to dynamically alter the user-interface based on the selected model. In this manner, the user need only deal with those parameters dictated by any given model and the addition of a new model need not change the main application.

The *Model Information* property sheet is where most of the parameters are edited. As previously stated, this property sheet is dynamic in that it presents a list of available models and adjusts the tabs of information in the property sheet based on the selected model type as shown below.



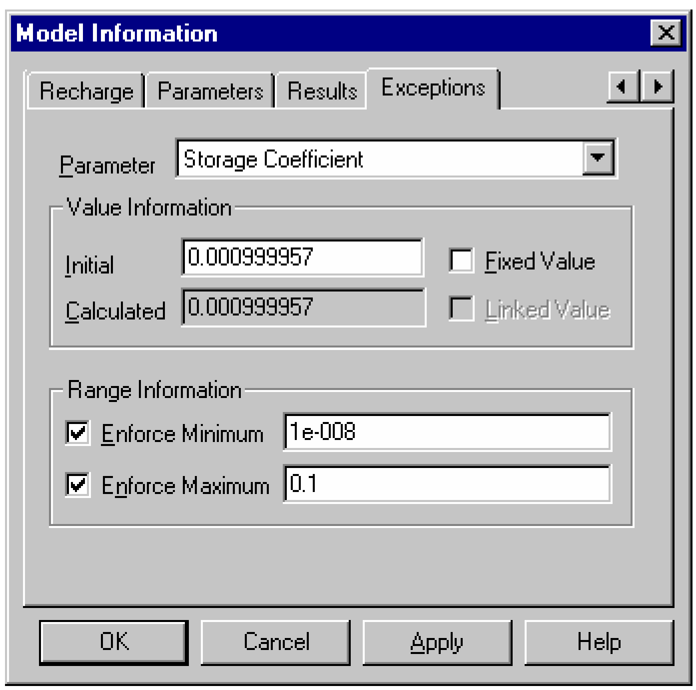
In addition to other tabs, the *Model Information* property sheet presents two groups of tabs relating to the specific parameters for the selected model solver. The first group contains the initial estimate for each parameter while the second group contains the calculated result for each after an auto-calibration. If the model does not support auto-calibration, the second group will be absent. There can be any number of tabs for each group depending on how many parameters are involved in the selected model. In the case of a “Theis, 1935 (Confined)” solution, there is one ***Parameters*** tab and one ***Results*** tab. In this simple case, the ***Parameters*** tab and ***Results*** tab are visually identical; however, they have different functionality.

The parameter fields in the ***Results*** tab are all "read-only" such that the values can be selected and copied to the clipboard but not modified. In some cases, parameters calculated by the model only appear on the ***Results*** tab. Such calculated parameters are provided in case the user would like to verify calculations based on the appropriate equations.

Fields on the ***Parameters*** tab are, in most cases, editable. The parameter values represent the initial value to be used during model calculations and are used as the initial estimate during optimization calculations. A special control, , appears to the right of most parameters in models that support auto-calibration. This control contains two separate check-boxes each of which can constrain adjacent parameter value. Although not applicable to all parameters, the lower check-box controls whether a parameter is "linked". A linked parameter gets its value and units from a value either specified or calculated in some other property sheet. Currently, no linked parameters exist but could in the future. You can click on the check-box to unlink a parameter. After it has been unlinked, the value and/or units can be changed. If it is subsequently linked, the value and units will revert back to those of the value to which it is linked. This concept allows you to manipulate the parameters during a model more easily.

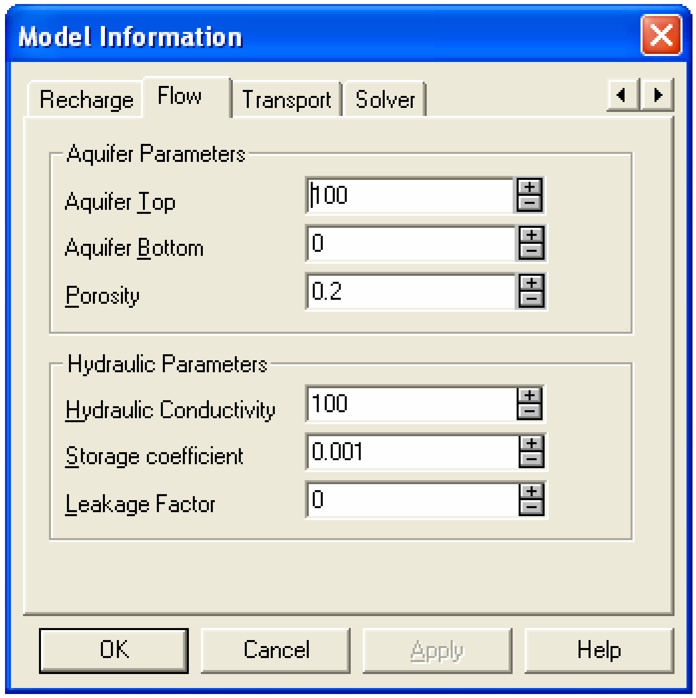
The upper check-box controls whether a parameter is "fixed" during optimizations. Any parameter that is fixed will not be allowed to change during the optimization calculations.

Should you desire to constrain a given parameter with respect to its minimum and/or maximum acceptable values during an optimization, the ***Exceptions*** tab is where to do it. In reality, this step may be required to get the solution to converge. The ***Exceptions*** tab contains a combobox listing all parameters involved in the solution. Once a specific parameter is selected, you can dictate how the optimization calculations treat it. As shown below, you can enforce a minimum value and/or a maximum value. You can also fix, link and specify the initial value as an alternative to the ***Parameters*** tab.



WinFlow/WinTran Parameters

Flow Parameters



The flow parameters are accessed using the **Edit->Flow Model** menu and are located on the ***Reference Head*** tab, the ***Recharge*** tab, on the ***Parameters*** tab (WinFlow) or ***Flow*** tab (WinTran). These parameters define the physical characteristics of the aquifer system. These characteristics include:

Hydraulic conductivity,

Aquifer Bottom Elevation,

Aquifer Top Elevation,

Reference Head Value,

Regional hydraulic gradient,

Recharge rate, and

Porosity (used for particle-tracking and transport).

Hydraulic conductivity is assumed to be homogeneous throughout the infinite aquifer and has units of [L/T], e.g. ft/d. The aquifer **top** and **bottom elevations** have units of length [L], e.g. ft, and are used to compute transmissivity. In addition, the flow model allows for conversion to unconfined flow. Therefore, if the head falls below the top of the aquifer the model becomes unconfined.

The **reference head** defines a point where the head is known. In the steady-state model, the reference head is always constant and never changes during simulations. All computations are based upon the reference head, which should be located as far from wells, ponds, etc. as possible. The reference head is analogous to a constant head in a numerical model.

The **regional gradient** is used to superimpose the regional flow system on your model. You enter the gradient here or by double-clicking the green arrow on the screen representing the reference head. The gradient is dimensionless (L/L) and is computed by dividing the change in water level over a specified distance by that same distance.

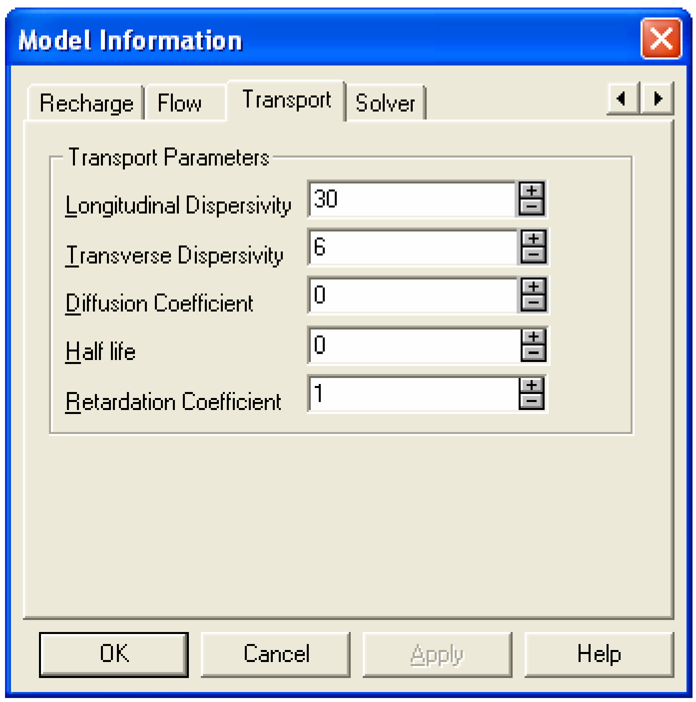
Recharge rate is the infiltration rate over the entire model. Recharge is primarily for regional simulations in which model boundaries are simulated with wells and linesinks. In this case, the recharge then creates water table divides and mounds between these regional features. Recharge should not be used for small-scale models such as those used to simulate remediation systems around gasoline stations and other similar sites.

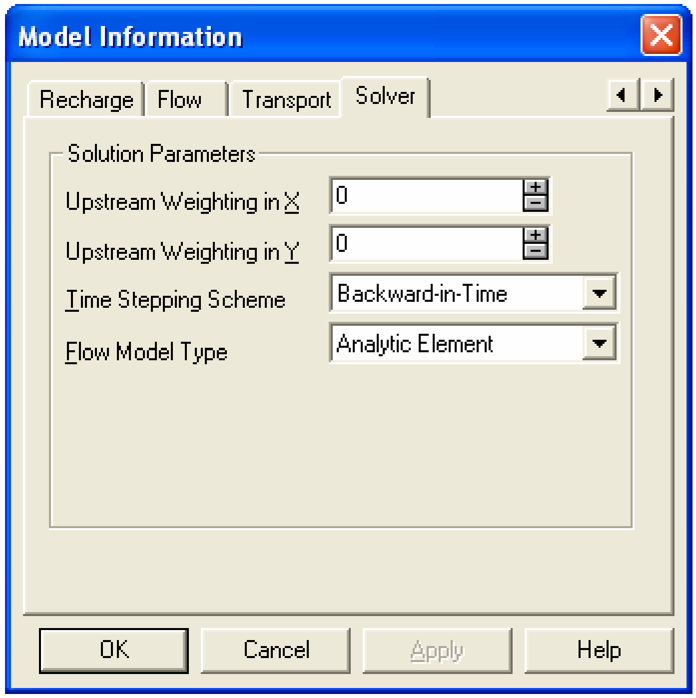
Porosity is used to compute the average linear groundwater velocity in WinTran. The porosity value entered in WinTran is actually the effective porosity and defines the ratio of connected void space to the volume of aquifer material. The porosity is used in computing velocity according to the following equation:

$$V = \frac{K}{n}\frac{dh}{dl}$$

where V is the average linear groundwater velocity (L/T), K is the hydraulic conductivity (L/T), n is the effective porosity, and dh/dl is the groundwater gradient.

Transport Parameters





The transport parameters are accessed using the **Edit->Flow Model** menu and are located on the ***Transport*** tab and the ***Solver*** tab. Transport parameters include dispersivity (longitudinal and transverse), coefficient of molecular diffusion, recharge concentration, retardation coefficient, and contaminant half-life. In addition the upstream weighting factors for the X- and Y-direction are specified.

Dispersivity is a scale-dependent parameter which is generally larger as the scale of the contaminant plume increases. A typical rule of thumb is that the dispersivity is 10 percent of the length of the contaminant plume (National Research Council, 1990). However, values of dispersivity reported in the literature range generally range from 1 to 100 percent of the problem scale (Gelhar, 1986). There are two values of dispersivity used in the WinTran transport model, longitudinal and transverse. Longitudinal dispersivity represents the spreading of the contaminant plume in the direction of groundwater flow. The transverse component represents spreading perpendicular to the flow direction. Usually, the longitudinal dispersivity is 5 to 10 times higher than transverse.

Molecular diffusion occurs when a contaminant spreads in water due to concentration gradients. That is, dissolved contaminants will spread in water from areas of high concentration to areas of lower concentration. This process is caused by random movement of molecules in a fluid. The coefficient of molecular diffusion (or simply the diffusion coefficient) is expressed in units of L^2/T (e.g., cm^2/s) and is often assumed to equal zero in advective-dominated transport. Only in very slow-moving groundwater is diffusion important. Bear and Verruijt (1987) estimate the diffusion coefficient to be approximately 1×10^{-5} cm^2/s in dilute systems.

Parameters are also assigned in the transport model based upon the type of constituents (chemicals) being simulated. These compound-specific parameters include the **half-life** for decaying species and chemical reaction types. Most transport models, including WinTran, lump chemical reactions into sorption processes in which a distribution coefficient (k_d) controls the relative velocity of the compound compared to the ground-water velocity. In this case, a **retardation coefficient** is computed which retards the velocity of the contaminant relative to the groundwater velocity.

Most contaminant transport models require you to enter a decay coefficient, which is different from the contaminant half-life. The decay coefficient is defined in the following equation:

$$\lambda = \ln 2 / t_{1/2}$$

where λ is the decay coefficient, ln 2 is the natural log of 2, and $t_{1/2}$ is the half-life of the contaminant. The half-life is the time required for half of the original mass of contaminant to decay. WinTran requires you to enter the half-life of the contaminant and computes the decay coefficient internally.

While the half-life is most often used for radioactive elements, such as uranium, it can also be used to express the decay of organic compounds through biodecay. The *Handbook of Environmental Degradation Rates* (Howard et al., 1991) is a good reference for contaminant half-life data.

WinTran requires you to enter the retardation coefficient directly rather than the distribution coefficient (k_d). Calculating the retardation coefficient is done by using the following equation:

$$R = 1 + k_d (\rho_b/n)$$

where R [dimensionless] is the retardation coefficient, k_d [L^3/M] the distribution coefficient, ρ_b [M/L^3] is the bulk density of the aquifer, and n [dimensionless] is the porosity. Other more complex reactions have been used in numerical models, however, these have not been commonly applied and are not supported in WinTran.

Recharge concentration is the contaminant concentration in the infiltrating recharge waters. Note, however, that this creates a source of contamination over the entire model domain.

The advective-dispersive transport equation is more difficult to solve numerically than the groundwater flow equation (Javandel et al. 1984). The problems are particularly severe when advection dominates over dispersion. In this situation, the finite-element solution, as employed in WinTran, can exhibit oscillations producing negative concentration values. One method for overcoming these oscillations is to use an upstream-weighted formulation of the transport equations. The upstream weighting factors on this dialog are used to implement this upstream weighting technique. An upstream weighting factor of implements full upstream weighting, while an upstream weighting factor of zero eliminates upstream weighting.

Time Stepping

Time-stepping parameters control the transient transport simulation. These parameters include the number of time steps, the initial time value, the length of the first time step, the maximum time step size, and the time step multiplier. WinTran starts the simulation at the initial time value using the initial time step size. Subsequent time steps are multiplied by the time-step multiplier to obtain the new time step size. This multiplication continues until the maximum time step size is reached or the end of the simulation occurs. Normally, you need to start with small time steps and gradually move to larger ones in order to have a good mass balance (i.e., mass balance error less than 10 percent). The mass balance error, current time step number, and time value are displayed in the status bar as the model runs. If the mass balance error rises above 10 percent, you should stop the simulation and adjust the time stepping parameters, dispersivity, or porosity until a better mass balance is achieved. The last chapter in the documentation provides some guidance on selection of these parameters.

The final parameters on the Time Stepping dialog define the time approximation in the transport solution. You may choose from Backward-in-time or Centered-in-time. The Centered-in-time approximation is more accurate but may exhibit instability. The

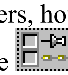
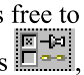
Backward-in-time introduces more numerical dispersion but is more stable. It is usually recommended that you start with backward-in-time and move to centered-in-time after you have a stable model.

Automatic Calibration

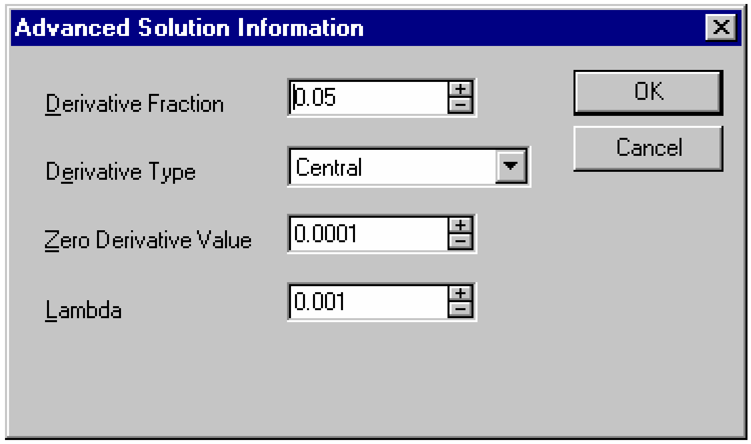
In some solutions supported by WinFlow, the Marquardt (modified Gauss-Newton) nonlinear least-squares technique can be used to find the best statistical match between the observed response data and the model you have chosen. Click the  button on the Match toolbar or choose **Calc->Optimize Model** from the menu to implement the automatic match.

You may view detailed results of the nonlinear least-squares match by selecting **Edit->Model** from the menu. The first tab, ***Solution***, on the property sheet displays the type of model you have performed. Other tabs include:

Parameters	The initial parameter values
Results	Optimized parameters computed by nonlinear least-squares
Exceptions	Unlink or unfix parameters and set enforced minimum and maximum parameter values
Reference Head	Values used to define the plane of initial heads
Recharge	Values used to define the recharge elipse for steady-state simulations

The Marquardt nonlinear least-squares procedure will estimate any parameter you choose. You may fix (or hold constant) any of the parameters, however, by selecting **Edit->Solution**, choosing the ***Parameters*** tab, and using the  icon. If the little pushpin is not checked, then the parameter is free to vary during the optimization. If you click the pushpin so that it looks like this , then that parameter is fixed and will not vary.

The automatic procedure will not always converge to a valid solution the first attempt. You may get an error message stating that the numerical solution has become unstable. This means that the Marquardt procedure could not come up with an improved estimate for the pertinent parameters. One problem with the Marquardt method is that your initial guesses for parameters must be fairly close to the "right" answer before it will work properly. This dialog is accessed from the **Advanced** button on the ***Solution*** tab of the *Model Information* property sheet.



If you have convergence problems, this dialog is the first place to go. You should increase the value of *Lambda* by several orders of magnitude. With higher values of Lambda the solution is more likely to converge but the precision of convergence is diminished. If you set *Lambda* to a value of 1 or 10 and you are still can't achieve convergence, go to the *Model Information* property sheet and look at the ***Results*** tabs. One or more of the parameters are likely diverging and have become invalid. Go to the ***Exceptions*** and start enforcing minimum and maximum values. The more you constrain the solution, the more likely it will converge.

Manipulating the information on the *Advanced Solution Information* dialog, fixing and/or constraining parameters and changing initial guesses are the three areas where you can influence the convergence. Sometimes it takes a good bit of trial and error to achieve convergence, especially if the data deviates substantially from theoretical.

Menus and Dialogs

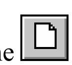
Introduction

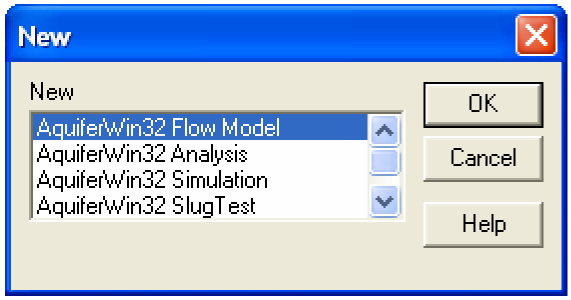
This chapter discusses the detailed data entered on all dialogs and property sheets in Aquifer^{Win32} and thus serves as a reference section of the manual. The dialogs and property sheets are shown for each menu selection and are presented in order from left to right and top to bottom in each menu. Aquifer^{Win32} actually has many different menus depending upon the active document type and active view type. You may make a view active by simply clicking the left mouse button anywhere within the view and the menus will change accordingly. Alternatively, the **F6** key toggles between views in the split window.

File Menu

The **File** menu contains several choices in a dropdown menu. Some of these items are standard to most Windows applications, including **New**, **Open**, **Close**, **Save**, **SaveAs**, **Print**, **Print Preview**, **Print Setup**, **Send**, and **Exit**. Other items are unique to Aquifer^{Win32}, including **Page Setup** (to set margins on the output page), **Import** (to import DOS text files containing the time-drawdown data), and **Export** (to create a text file report or to create graphics files in either DXF or Windows Metafile, wmf, formats). Each of these selections from the **File** menu is described below.

New

To start a new aquifer test, simply click the  button on the Standard toolbar or select the **File->New** menu. Selecting **New** from the **File** menu displays five types of documents or analysis types, including (1) “AquiferWin32 Analysis”, (2) “AquiferWin32 Simulation”, (3) “AquiferWin32 Slug Test”, (4) “AquiferWin32 Step Test” and (5) “AquiferWin32 Flow Model”, as shown below.

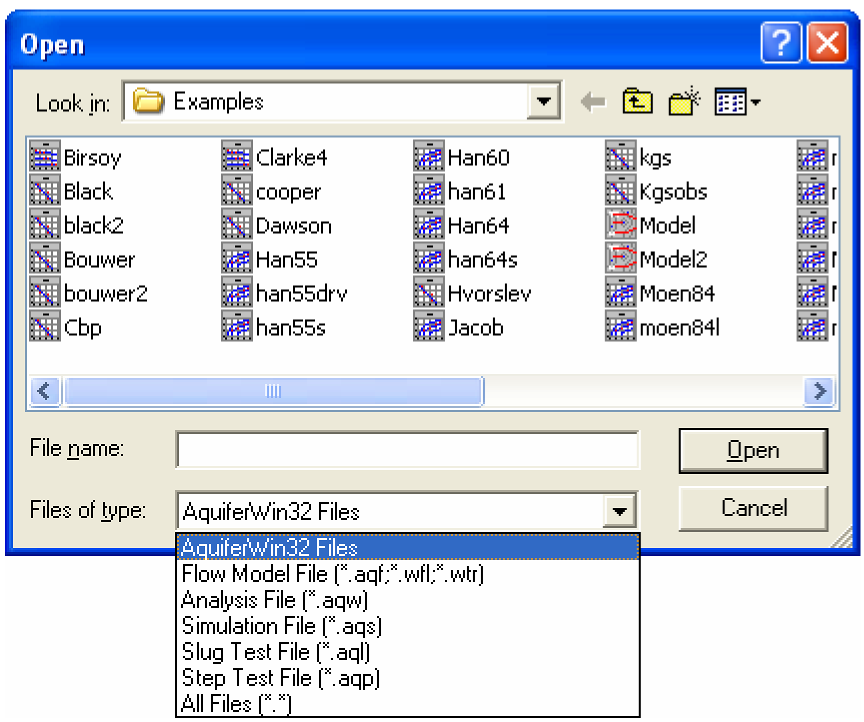


Clicking the button on the Standard toolbar creates an **Analysis** document in the Standard and Professional versions and a **Slug Test** document if you are only licensed for the Slug Test version of Aquifer^{Win32}. To create a **Simulation, Slug Test, Step Test** or **Flow Model** document, you must select **File->New** menu.

Open

Use the **Open** menu to read an existing Aquifer^{Win32} file. You may also use the button on the Main or press **Ctrl-O**.

The Open dialog is a standard Windows dialog that allows you to choose any directory or drive on your computer. By default, all supported file types are displayed; however, you can change the "*Files of type:*" combobox to any one of the five supported file types, a **Flow Model** document (*.aqf), an **Analysis** document (*.aqw), a **Slug Test** document (*.aql), a **Simulation** document (*.aqs) and a **Step Test** document (*.aqp).



Close

Use the **Close** menu to close the active Aquifer^{Win32} document. You will be prompted to save the document if you have made any changes or it is a newly created document.

Save

The **Save** menu saves the current document to a disk file. The name of the file is shown at the top of the Aquifer^{Win32} window. For a new document that has not been saved yet, you may want to use the **Save As** menu in order to provide a name of your choice. The **Save** menu may also be executed from the  button on the Standard toolbar or by pressing **Ctrl-S.**

Save As

The **Save As** menu allows you to save the current document with a new name. A standard Windows file dialog prompts for the file name and path.

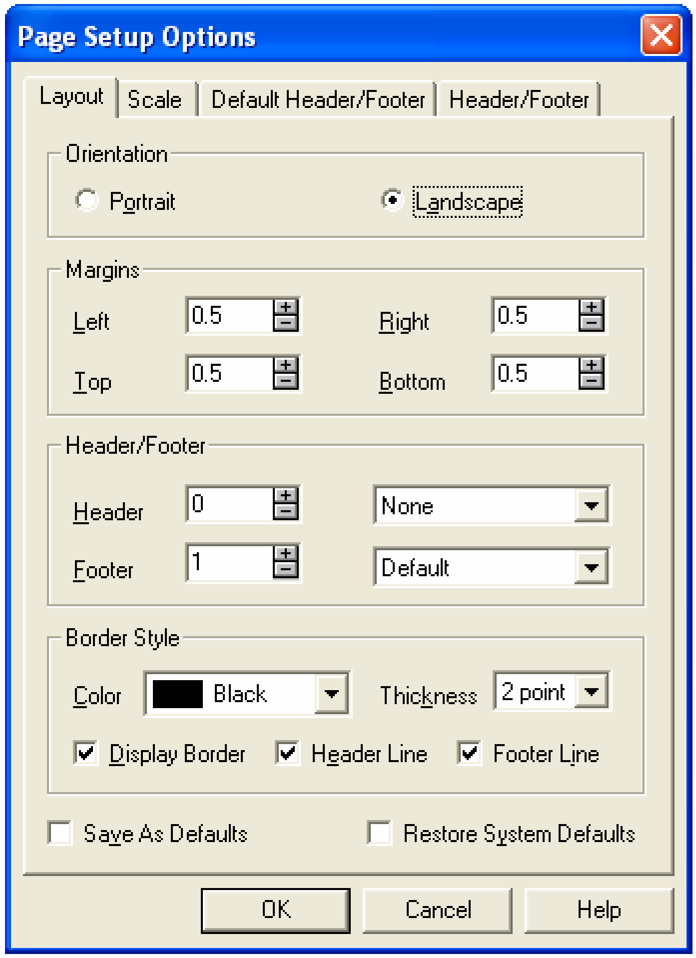
Print, Print Preview, Print Setup

There are four menu selections related to printing the currently active view. These include **Print**, **Print Preview**, **Print Setup**, and **Page Setup**. The first three options are standard Windows menus and will work the same as your other Windows applications. The current view can be printed to any printer, plotter, fax modem, or other device supported by Windows. The only potential problem with printing is related to the rotated fonts used in type curve and contour labels. Some Windows device drivers do not properly support rotated fonts. <u>It is very important to select **Print Setup** and choose the option to "Print TrueType as Graphics".</u> Most Windows device drivers will support this option, which allows for proper font rotation. If after selecting this option the type curve and contour labels are not properly rotated, you should call the manufacturer of your printer or other output device to get an updated Windows driver.

Page Setup

NOTE: This menu is not available in the spreadsheet view.

Choose **Page Setup** to control the margins and scale on the printed output. You can also use this menu to define headers and footers and select which ones will occur on the printed output via the *Page Setup Options* property sheet.



Orientation

Portrait	When selected, the printed output uses the short dimension of the paper as the x-axis and the long dimension of the paper as the y-axis
Landscape	When selected, the printed output uses the long dimension of the paper as the x-axis and the short dimension of the paper as the y-axis

Margins

Left	Left margin in inches
Right	Right margin in inches
Top	Top margin in inches
Bottom	Bottom margin in inches

Header/Footer

Header	Space, in inches, between the top margin and printing area reserved for printing the header
Header combobox	Specifies what to print in the header area if Header is greater than 0 inches

None – No header will be printed

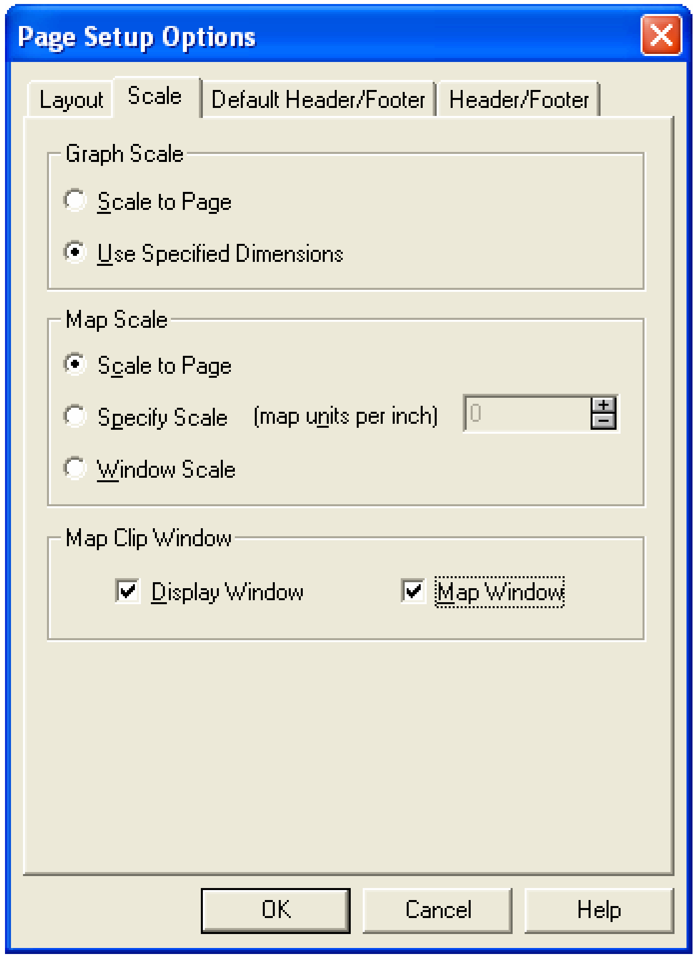
Default –	Uses the information on the ***Default Header/Footer*** page to construct the header
Custom -	Stands for any number of user-designed header/footer elements

Footer	Space, in inches, between the bottom margin and printing area reserved for printing the footer
Footer combobox	Specifies what to print in the footer area if Footer is greater than 0 inches

None –	No footer will be printed
Default –	Uses the information on the ***Default Header/Footer*** page to construct the footer
Custom -	Stands for any number of user-designed header/footer elements

Border Style

Color	The color to use when displaying the border lines
Thickness	The line thickness to use when displaying the border lines
Display Border	When checked, a border line is displayed around the printable area defined by the margins
Header Line	When checked, a border line is displayed separating the header area from the main printable area
Footer Line	When checked, a border line is displayed separating the footer area from the main printable area
Save As Defaults	When checked, the values contained in this property sheet are stored in the system registry when the property sheet is accepted and are used as the defaults when an new document is created
Restore System Defaults	When checked, the default system values are restored when the property sheet is accepted; this operation occurs before the **Save As Defaults** operation so having both checked resets both the document and the system registry



Graph Scale

Scale to Page	While maintaining the ratio of vertical to horizontal graph dimensions, the graph is scaled to fill the printed page.
Use Specified Dimensions	The graph dimensions specified in the *Graph Information* property sheet are used to print the graph.

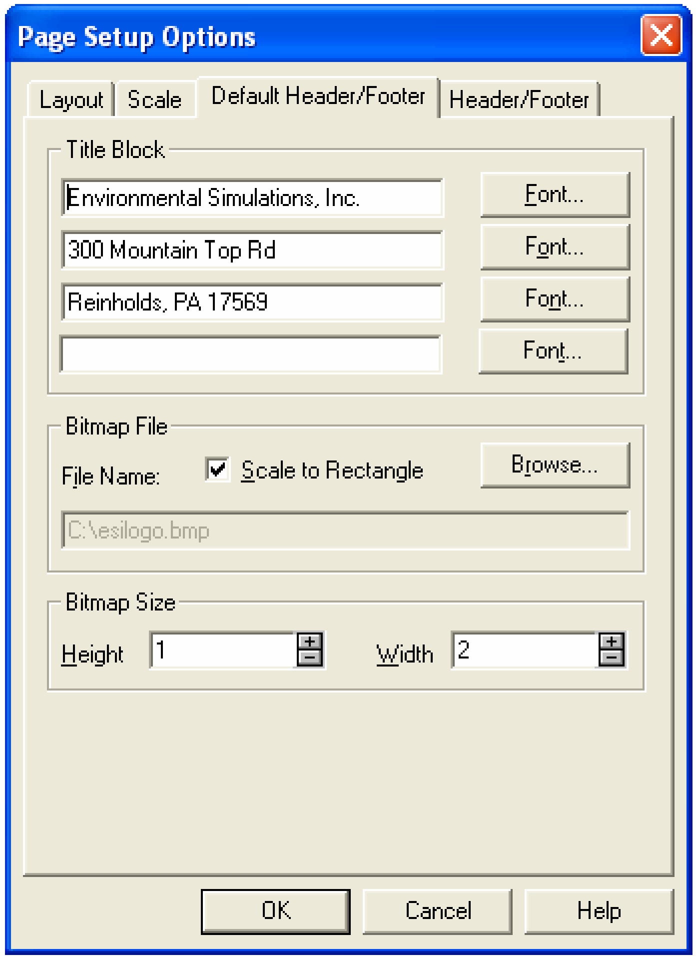
Map Scale

Scale to Page	The map scale is adjusted to fill the printed page.
Specify Scale	The map scale of the printed page is set to the value in the *(map units per inch)* field. The output will be paginated accordingly.
Window Scale	The prevailing map scale in the display window is used in printing. . The output will be paginated accordingly.

Map Clip Window

Display Window	The printed map is clipped to that which is displayed in the display window and scaled as defined above.
Map Window	The printed map is clipped to the defined map window and scaled as defined above.

NOTE: If both check-boxes are checked, the intersection between the display window and map window is used as the clipping window.



Title Block

Each of the four data fields and **Font** buttons define a line of text in the title block section

Bitmap File

File Name: The name of the file that was loaded for Bitmap and Metafile type frames

NOTE: The contents of the file are loaded and stored within the document so the file is no longer required and the name is for historical reference only.

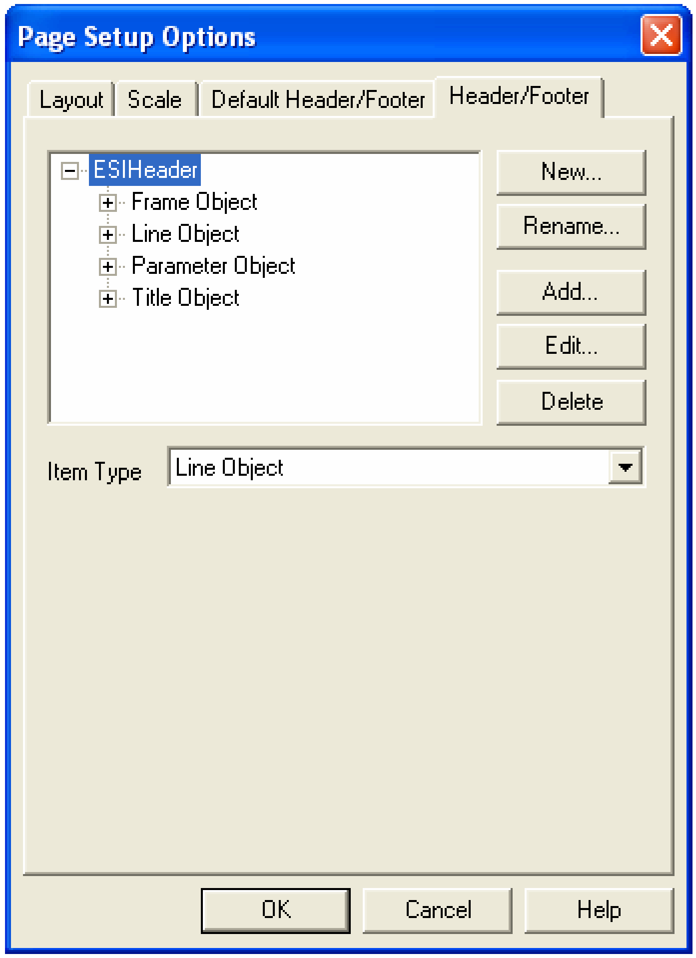
Scale to Rectangle If checked, the bitmap is expanded/compressed to fill the frame

NOTE: This check-box is usually checked, otherwise, the screen display and printed output will not be the same.

Browse Click this button to display a standard File Open dialog used to locate the Bitmap or Metafile file stored on the computer.

Bitmap Size

Height	Specifies the height, in inches, of the frame used to contain the bitmap or metafile
Width	Specifies the width, in inches, of the frame used to contain the bitmap or metafile



New:	Creates a new header/footer and prompts for a name.
Rename:	Prompts the user for a new name for the selected header/footer.
Add:	Adds an item to the selected header/footer. The item selected in *Item Type* is added.
Edit:	Edits the currently selected item.
Delete:	Deletes the currently selected item.
Item Type:	Specifies the currently active item to be added when the **Add** button is clicked.

NOTE: A context menu is provided to allow Cut/Copy/Paste of headers and footers to move them between documents.

Import

In the spreadsheet view of an **Analysis**, **Slug Test** or **Step Test** document, the **Import** menu allows you to import data from a text file. The spreadsheet view must be the active view before you can use the **Import** option. Simply click within the spreadsheet to make it active.

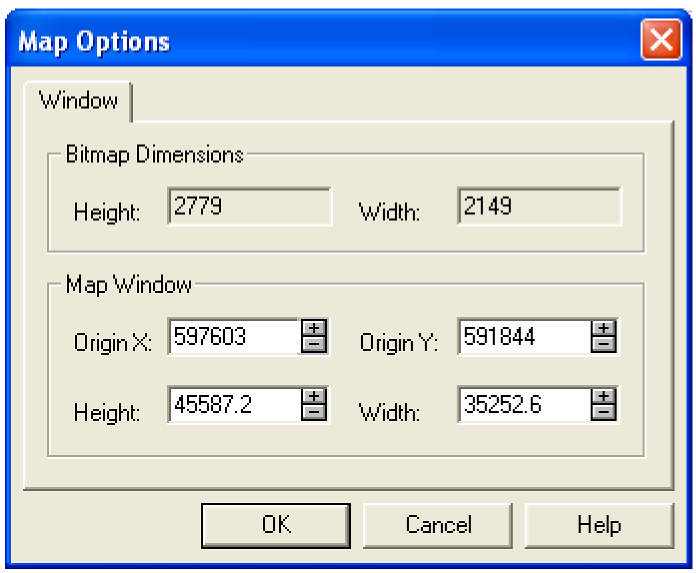
The data values in the file must be separated by a comma character and each row of data must be terminated with a newline character. Additionally, the order of the values in the file corresponds to the order of the columns displayed on the spreadsheet. Since the number of and order of columns on the spreadsheet can be adjusted make sure the data matches the spreadsheet into which it will be imported.

Map

The **Map** menu allows you to import a basemap in one of two different formats. The map view must be the active view before you can use the **Map** menu. Simply click within the map view to make it active.

The File Open dialog activated by this menu has a *File of Type* combobox that supports ESI's map file format, DXF file format and Bitmap file format. Set the combobox to the appropriate type and accept the dialog. If the file type is DXF, you will be prompted to enter a multiplier to scale the DXF coordinates.

If you have selected Bitmap a property sheet will be displayed as below. If the bitmap you are using was generated from a scanned quad map, it may have come with a .tfw (world) file. You can rename this file to filename.bmw where the filename is the same as for the bitmap file and the Map Window parameters will be automatically calculated from the world file.



Bitmap Dimensions

Height:	The number of vertical pixels in the bitmap being imported.
Width:	The number of horizontal pixels in the bitmap being imported.

Map Window

Origin X:	The x-coordinate in map units corresponding to the lower left corner of the bitmap being imported.
Origin Y:	The y-coordinate in map units corresponding to the lower left corner of the bitmap being imported.
Height:	The height of the bitmap in map units.
Width:	The width of the bitmap in map units.

Import Wells

In the ***Test Simulator*** view of a **Simulation** document or the ***Site Map*** view of an **Analysis** or **Slug Test** document, the **Import Wells** menu allows you to import data for wells. The file must be a text file (ASCII file) containing three columns of data separated by commas. The order of the columns is as follows:

(1) Well name,

(2) X coordinate of well, and

(3) Y coordinate of well

The well import file name must use a *.dat* extension. For example, the name of a valid file would be *wells.dat*.

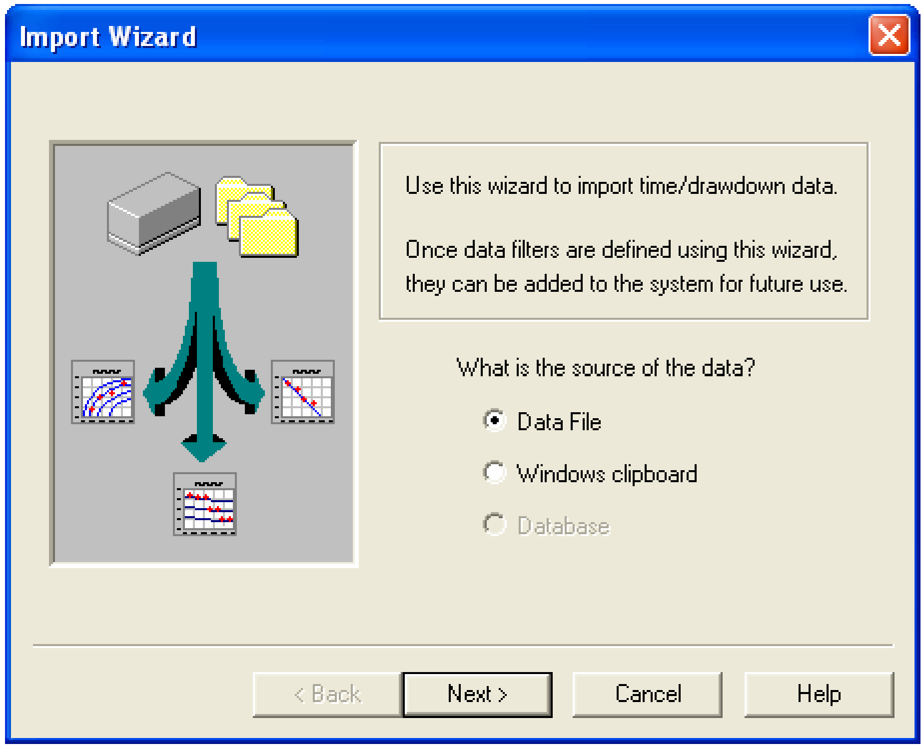
Selecting **Import** from the **File** menu displays a standard Windows file dialog. Choose the path and file name to import. Aquifer^{Win32} reads data from the file and adds the wells to the current model. If you are replacing existing wells, you must first select and delete the existing wells. Importing new wells at the same location as existing wells will give you 2 wells at each location.

Import Wizard – Time/Drawdown

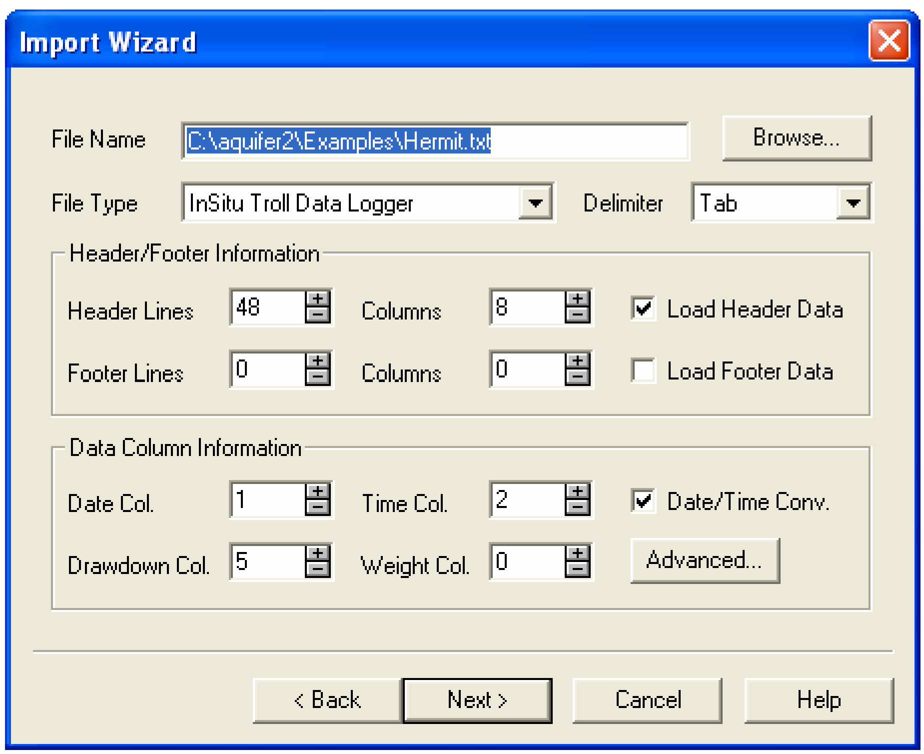
In the spreadsheet view, the import wizard enables easier data importing from data logger files and other applications. The **Import Wizard->Time/Drawdown** menu activates the import wizard which can be used to define filters for data formats of disk files and data copied to the Windows' clipboard. The wizard separates the information into header information, data, and footer information. Any number of columns are supported in each of the three categories. Several data delimiters are supported and the data is loaded into a tab spreadsheet for review before the data is actually imported into the program. The time data can be either elapsed time or any of a number of supported data/time formats. Well response can be head or drawdown; in the case of head data, the user specifies the static head from which to calculate drawdown. Similarly, elapsed time is calculated from date/time data given a start date/time value. Additionally, both time and drawdown data can be transformed and clipped.

Once defined, the filter can be added to the program and will appear in the list of supported file types. Such filters can also be used for data in the Windows' clipboard. A case in point is In-Situ's Data Manager program; once loaded into the Data Manager program, the information can be copied to the Windows' clipboard and imported directly into Aquifer^{Win32} without creating an intermediate data file. Likewise, filters can be defined for custom spreadsheets defined in a spreadsheet program like Microsoft Excel and copied to the clipboard and imported directly into Aquifer^{Win32}.

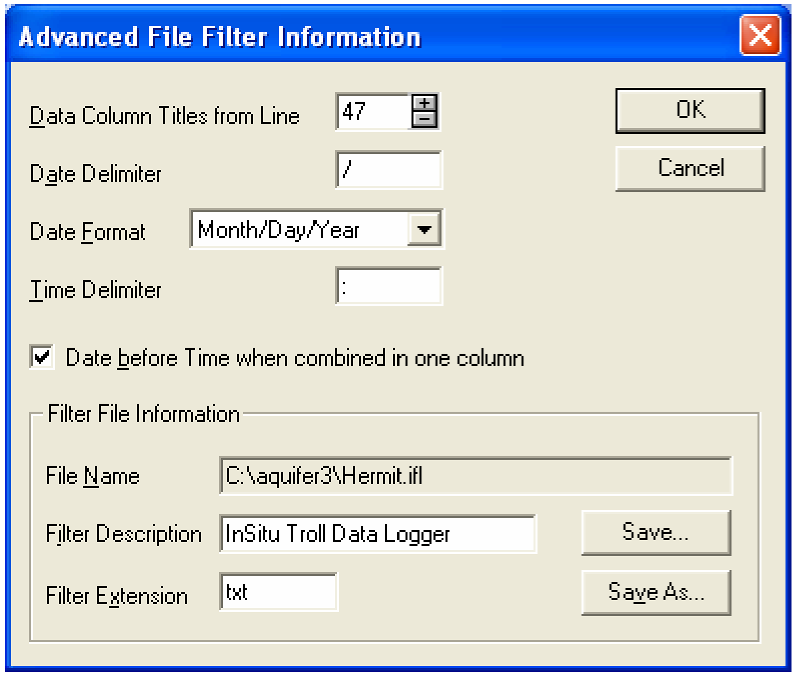
The following are descriptions of the pages involved in the wizard.



Data File	Specifies that the data will be read from a data file
Windows clipboard	Specifies that the data will be read from the Windows' clipboard
Database	Specifies that the data will be imported directly from a database (not currently supported)

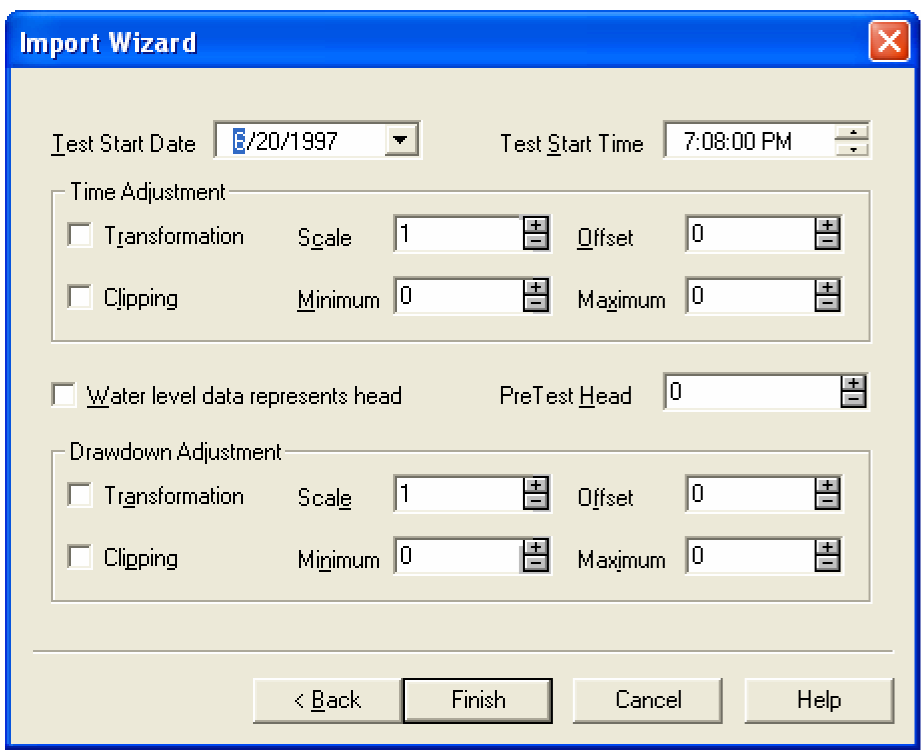


File Name	Identifies the data file to import
Browse	When clicked, presents a File Open dialog to locate the appropriate data file
File Type	
Generic Data File -	Loads system default values
InSitu Data Logger -	Loads values specific to the InSitu Data Logger format
Isodaq Data Logger -	Loads values specific to the Isodaq Data Logger format
Custom -	The list can be enhanced to include as many custom formats as needed
Delimiter	
Comma -	Specifies a data delimiter of the comma character
Space -	Specifies a data delimiter of the space character
Tab -	Specifies a data delimiter of the tab character
Header/Footer Information	
Header Lines	Specifies the number of lines of header information before the data columns begin
Columns	Specifies the number of columns of header information present on each line
Load Header Data	When checked, the header information will be loaded into the subsequent tab spreadsheet
Footer Lines	Specifies the number of lines of footer information after the data ends
Columns	Specifies the number of columns of footer information present on each line
Load Footer Data	When checked, the footer information will be loaded into the subsequent tab spreadsheet
Data Column Information	
Date Col.	Indicates which column contains the value for date
Time Col.	Indicates which column contains the value for time
Date/Time Conv.	When checked, the date/time data is converted to elapsed time
Drawdown Col.	Indicates which column contains the drawdown data
Weight Col.	Indicates which column contains the data weights
Advanced	When clicked, presents the following dialog with advanced settings



Data Column Titles from Line	Indicates which line, 0 if none, to take data column titles from
Date Delimiter	Specifies the delimiter used to separate components of the date format
Date Format	
Month/Day/Year -	Month first, Day second, Year third
Day/MonthYear -	Day first, Month second, Year third
Time Delimiter	Specifies the delimiter used to separate components of the time
Date before Time…	When checked, the date value precedes the time value when both are combined into one column
File Filter Information	
File Name	Specifies the file name to store the filter information in; must be located in the program directory to be accessible to the application
File Description	Specifies the descriptive text displayed in the File Type combobox
File Extension	Specifies the default file extension for files of this type, eg. dat
Save	When clicked, updates the current filter
Save As	When clicked, presents a File Save dialog to specify the file into which to save the filter

This tab spreadsheet displays the columns of data loaded from the data source with optional tabs for the header and footer information. The data is read-only and used to verify that the filter information has been set up properly.



Test Start Date	Specifies the date for the start of the pump test; used to calculate elapsed time from date/time data

Test Start Time	Specifies the time for the start of the pump test; used to calculate elapsed time from date/time data
Time Adjustment	
Transformation	When checked, the time data are transformed by multiplying it by the Scale and adding the Offset
Scale	Specifies the value by which to multiply the time value if transformation is active
Offset	Specifies the value to be added to the scaled time value if transformation is active
Clipping	When checked, the time data are clipped based on the specified Minimum and Maximum values
Minimum	Specifies the minimum acceptable value for time data; data points with a time value less than this value are not imported
Maximum	Specifies the maximum acceptable value for time data; data points with a time value greater than this value are not imported
Water level data…	When checked, the water level data represents head and must be subtracted from the PreTest Head to calculate drawdown
PreTest Head	Specifies the head value at the start of the test
Drawdown Adjustment	
Transformation	When checked, the drawdown data are transformed by multiplying it by the Scale and adding the Offset
Scale	Specifies the value by which to multiply the drawdown value if transformation is active
Offset	Specifies the value to be added to the scaled drawdown value if transformation is active
Clipping	When checked, the drawdown data are clipped based on the specified Minimum and Maximum values
Minimum	Specifies the minimum acceptable value for drawdown data; data points with a drawdown value less than this value are not imported

Maximum Specifies the maximum acceptable value for drawdown data; data points with a drawdown value greater than this value are not imported

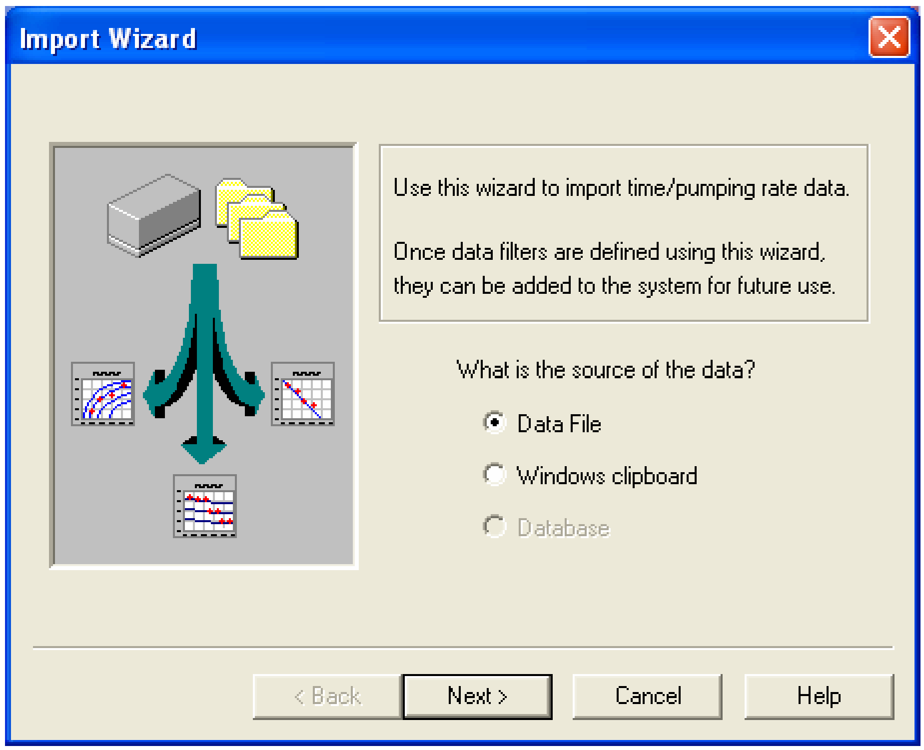
Import Wizard – Pumping Rate

In the spreadsheet view, the import wizard enables easier data importing from data logger files and other applications. The **Import Wizard->Pumping Rate** menu activates the import wizard which can be used to define filters for data formats of disk files and data copied to the Windows' clipboard. The wizard separates the information into header information, data, and footer information. Any number of columns are supported in each of the three categories. Several data delimiters are supported and the data is loaded into a tab spreadsheet for review before the data is actually imported into the program. The time data can be either elapsed time or any of a number of supported data/time formats. Elapsed time is calculated from

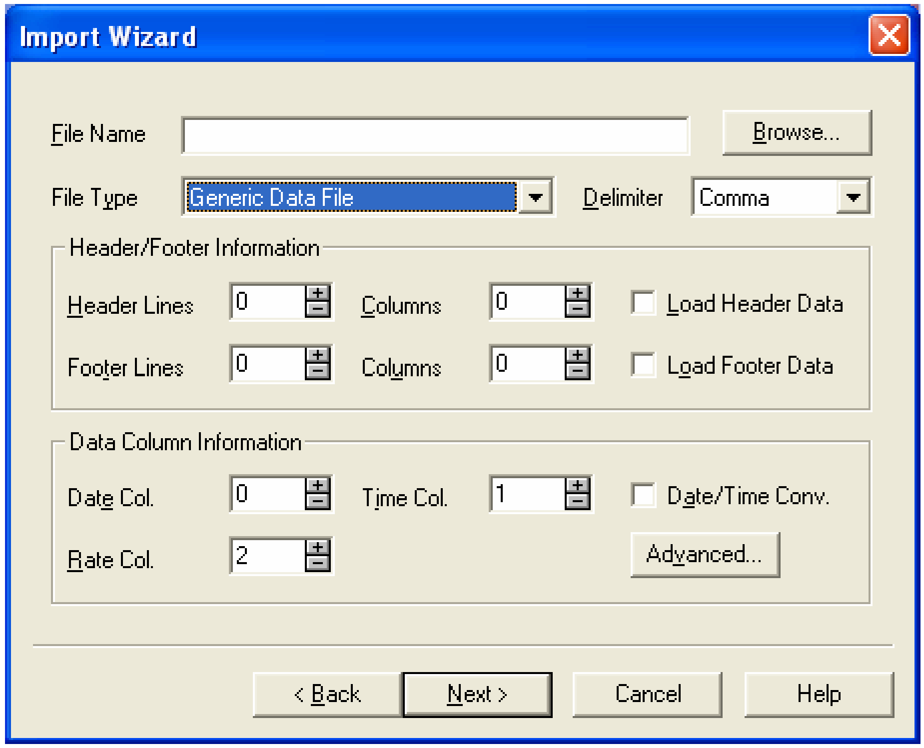
date/time data given a start date/time value. Additionally, both time and pumping rate data can be transformed and clipped.

Once defined, the filter can be added to the program and will appear in the list of supported file types. Such filters can also be used for data in the Windows' clipboard. A case in point is In-Situ's Data Manager program; once loaded into the Data Manager program, the information can be copied to the Windows' clipboard and imported directly into Aquifer^{Win32} without creating an intermediate data file. Likewise, filters can be defined for custom spreadsheets defined in a spreadsheet program like Microsoft Excel and copied to the clipboard and imported directly into Aquifer^{Win32}.

The following are descriptions of the pages involved in the wizard.



Data File	Specifies that the data will be read from a data file
Windows clipboard	Specifies that the data will be read from the Windows' clipboard
Database	Specifies that the data will be imported directly from a database (not currently supported)



File Name	Identifies the data file to import
Browse	When clicked, presents a File Open dialog to locate the appropriate data file

File Type

Generic Data File -	Loads system default values
InSitu Data Logger -	Loads values specific to the InSitu Data Logger format
Isodaq Data Logger -	Loads values specific to the Isodaq Data Logger format
Custom -	The list can be enhanced to include as many custom formats as needed

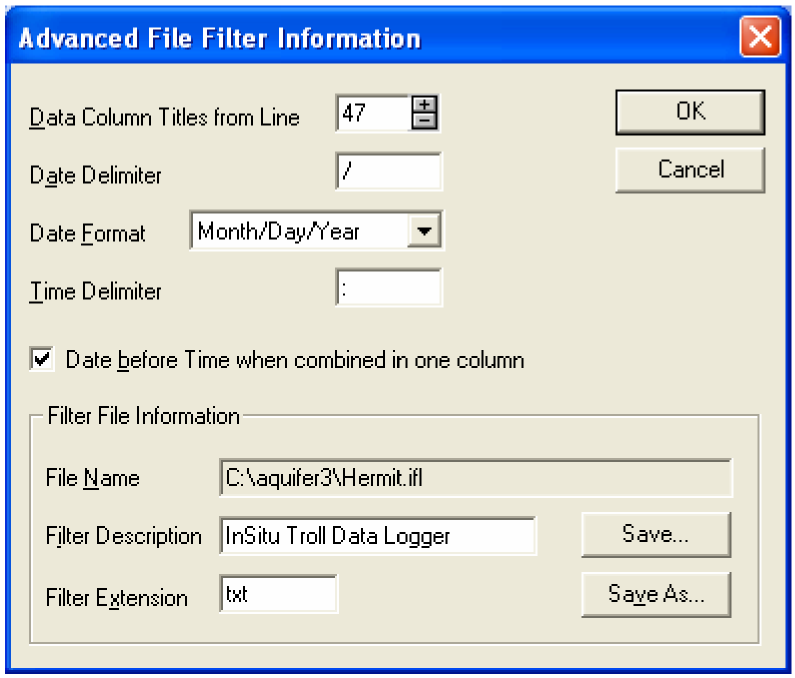
Delimiter

Comma -	Specifies a data delimiter of the comma character
Space -	Specifies a data delimiter of the space character
Tab -	Specifies a data delimiter of the tab character

Header/Footer Information

Header Lines	Specifies the number of lines of header information before the data columns begin
Columns	Specifies the number of columns of header information present on each line
Load Header Data	When checked, the header information will be loaded into the subsequent tab spreadsheet
Footer Lines	Specifies the number of lines of footer information after the data ends

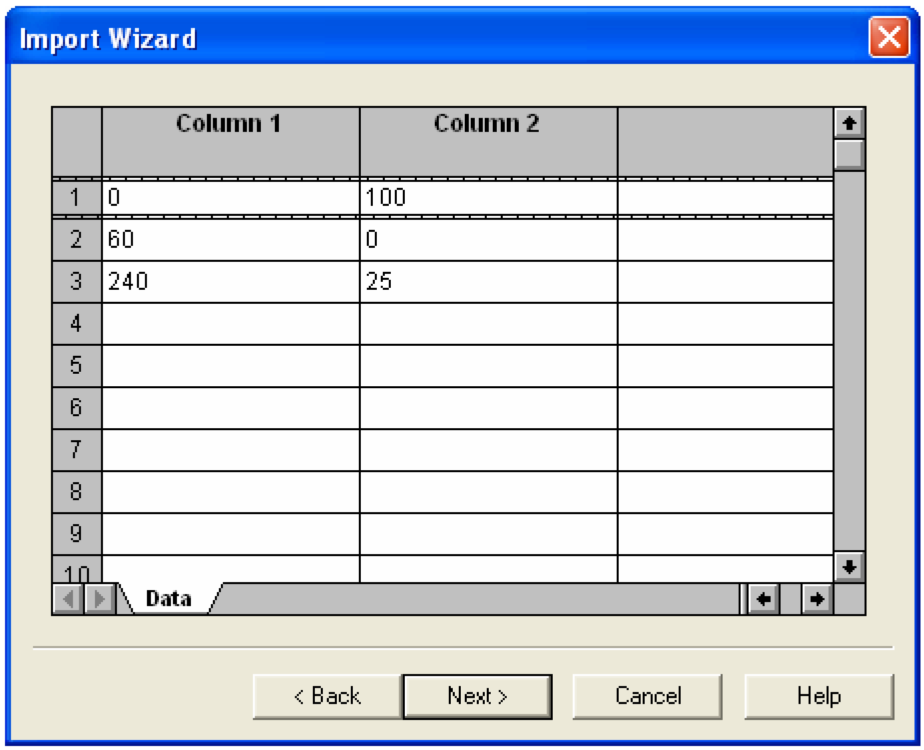
Columns	Specifies the number of columns of footer information present on each line
Load Footer Data	When checked, the footer information will be loaded into the subsequent tab spreadsheet
Data Column Information	
Date Col.	Indicates which column contains the value for date
Time Col.	Indicates which column contains the value for time
Date/Time Conv.	When checked, the date/time data is converted to elapsed time
Rate Col.	Indicates which column contains the pumping rate data
Advanced	When clicked, presents the following dialog with advanced settings



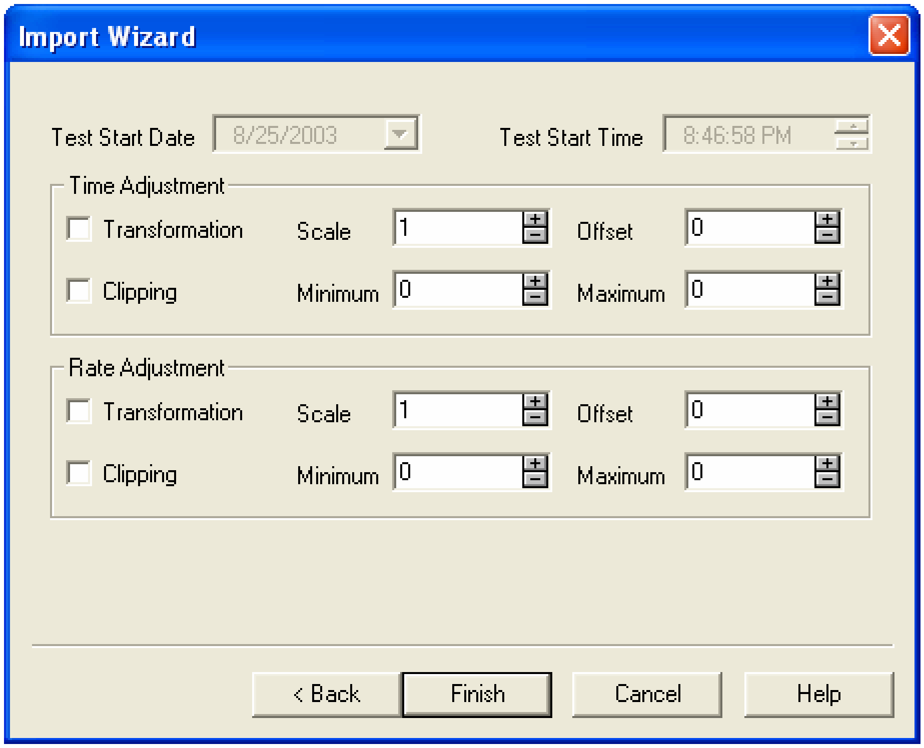
Data Column Titles from Line	Indicates which line, 0 if none, to take data column titles from
Date Delimiter	Specifies the delimiter used to separate components of the date format
Date Format	
Month/Day/Year -	Month first, Day second, Year third
Day/MonthYear -	Day first, Month second, Year third
Time Delimiter	Specifies the delimiter used to separate components of the time
Date before Time…	When checked, the date value precedes the time value when both are combined into one column

File Filter Information

File Name	Specifies the file name to store the filter information in; must be located in the program directory to be accessible to the application
File Description	Specifies the descriptive text displayed in the File Type combobox
File Extension	Specifies the default file extension for files of this type, eg. dat
Save	When clicked, updates the current filter
Save As	When clicked, presents a File Save dialog to specify the file into which to save the filter



This tab spreadsheet displays the columns of data loaded from the data source with optional tabs for the header and footer information. The data is read-only and used to verify that the filter information has been set up properly.



Test Start Date	Specifies the date for the start of the pump test; used to calculate elapsed time from date/time data
Test Start Time	Specifies the time for the start of the pump test; used to calculate elapsed time from date/time data
Time Adjustment	
Transformation	When checked, the time data are transformed by multiplying it by the Scale and adding the Offset
Scale	Specifies the value by which to multiply the time value if transformation is active
Offset	Specifies the value to be added to the scaled time value if transformation is active
Clipping	When checked, the time data are clipped based on the specified Minimum and Maximum values
Minimum	Specifies the minimum acceptable value for time data; data points with a time value less than this value are not imported
Maximum	Specifies the maximum acceptable value for time data; data points with a time value greater than this value are not imported
Rate Adjustment	
Transformation	When checked, the pumping rate data are transformed by multiplying it by the Scale and adding the Offset
Scale	Specifies the value by which to multiply the pumping rate value if transformation is active

Offset	Specifies the value to be added to the scaled pumping rate value if transformation is active
Clipping	When checked, the pumping rate data are clipped based on the specified Minimum and Maximum values
Minimum	Specifies the minimum acceptable value for pumping rate data; data points with a pumping rate value less than this value are not imported
Maximum	Specifies the maximum acceptable value for pumping rate data; data points with a pumping rate value greater than this value are not imported

Export

In a graph view, the **Export** menu is used to save four different file types which provide you with a means of transferring graphics and data to other software packages. Supported file formats include the following:

1. Summary (*.out) – ASCII file presenting analysis results
2. AutoCAD DXF (*.dxf) - file for importing into CAD software
3. Windows Metafile (*.wmf) – graphics import file for most Windows applications such as Microsoft Word or Excel. You should use the Windows metafile if you want to preserve fonts and colors when importing the type curves into other applications. DXF files do not preserve fonts or colors properly.
4. Placeable Metafile (*.wmf) – alternative metafile format required by some Windows applications

In a spreadsheet view, the **Export** menu exports an ASCII file with the time and drawdown data. The file is a comma-delimited text file that can be imported into most applications, such as spreadsheets. Additionally, the order of the values in the file will correspond to the order of the columns displayed on the spreadsheet. Since the number of and order of columns on the spreadsheet can be adjusted make sure the spreadsheet matches what you want to export.

In a map view, the **Export** menu is used to save 10 different file types which provide you with a means of transferring graphics and data to other software packages. Supported file formats include the following:

1. Surfer ASCII (*.grd) - ASCII file containing the matrix of head values computed by Aquifer^{Win32} for recontouring in SURFER
2. Surfer BINARY (*.grd)) - BINARY file containing the matrix of head values computed by Aquifer^{Win32} for recontouring in SURFER
3. Surfer XYLINE (*.bln) - ASCII file containing selected graphic lines for use in SURFER
4. Surfer POSTING (*.pst)) - ASCII file containing selected graphic text for use in SURFER
5. AutoCAD DXF (*.dxf) - file for importing into CAD software
6. SpyGlass (*.spy) - Spyglass is a commercial graphics program that provides more sophisticated visualization capabilities than Aquifer^{Win32}
7. ASCII XYZ (*.dat)) - ASCII file containing the matrix of head values computed by Aquifer^{Win32} in the format x, y, z

8. Geosoft (*.geo) - Geosoft is a commercial graphics program that provides more sophisticated visualization capabilities than Aquifer$^{\text{Win32}}$
9. Windows Metafile (*.wmf) – graphics import file for most Windows applications such as Microsoft Word or Excel. You should use the Windows metafile if you want to preserve fonts and colors when importing the type curves into other applications. DXF files do not preserve fonts or colors properly.
10. Placeable Metafile (*.wmf) – alternative metafile format required by some Windows applications
11. ArcView Shapefile (*.shp) – Exports using ArcView™ Shapefile format.

Send

If electronic mail has been installed on the computer, this menu will appear. The **Send** menu is used to send the active document through electronic mail. The currently supported electronic mail editor is displayed with a copy of the active document included as an attachment.

Most Recently Used Files

The four most recently opened document files are listed at the bottom of the **File** menu for easy access. Simply click on the appropriate file name in the menu and that file will be automatically opened.

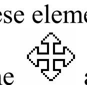
Exit

The **Exit** menu closes all open documents and exits the application. You will be prompted to optionally save each of the currently opened documents that has unsaved changes.

Edit Menu

The **Edit** menu, like the **File** menu, contains both standard Windows menus and those customized for Aquifer$^{\text{Win32}}$. The standard **Edit** menu including **Cut**, **Copy**, **Paste**, and **Delete** has been enhanced to include additional menus items specific to Aquifer$^{\text{Win32}}$. As with other menus, the accessibility of particular menu items is determined by the active view and the active document type.

Cut, Copy, Paste, Delete

The **Cut, Copy, Paste,** and **Delete** menus are standard Windows functions. Each menu operates on the currently selected elements or spreadsheet lines. An element can be a well, title, line, frame, legend, symbol, parameter, or scale. To select one of these elements in a map view, simply move the cursor over the element until you see the  arrow appear, then click the left mouse button. The element will turn a solid color or a bounding rectangle will appear to identify that it has been selected. Multiple selection is achieved by repeating the procedure while holding down the **Shift** key. Now, all of the edit operations will work on this element. If no element has been selected, these menus will appear in gray and cannot be used. One exception is the **Copy** menu which allows copying the entire view to the clipboard as an OLE object and a metafile.

To select lines in a spreadsheet view, click the left mouse button in the leftmost column of the spreadsheet and drag a selection. Alternatively, hold down the shift key and use the arrow keys or page keys to expand the selection.

Use **Cut** to remove the selected elements or spreadsheet lines from the view and put them on the Windows clipboard. You may also use the button on the Standard toolbar to cut. Use the **Delete** menu to simply remove the elements or spreadsheet lines from the view without placing them on the clipboard. Use **Copy** to keep the elements or spreadsheet lines in the view but make a copy on the Windows clipboard. You may also use the button on the Standard toolbar to copy. You may put clipboard elements into the current view using the **Paste** menu selection or the button on the Standard toolbar. The items or lines are pasted back at the same coordinates as the original elements or before the currently selected line of the spreadsheet. These edit features are most useful to transfer elements from one aquifer test analysis document to another. You may have several analyses open at one time and cut, copy, and paste elements from one document to another.

During spreadsheet paste operations, the data is assumed to be in Windows text clipboard format with **Tab** characters separating data values and a newline character at the end of each data row. During spreadsheet cut and copy operations this same format is produced. Additionally, the order of the values in the clipboard corresponds to the order of the columns displayed on the spreadsheet. Since the number of and order of columns on the spreadsheet can be adjusted make sure the data matches the spreadsheet into which it will be pasted.

Undo

The **Undo** menu is only available on the spreadsheet to undo the changes made to the active cell. Once the active cell is changed in the spreadsheet, the previous changes are committed.

Select All

In the spreadsheet view, the **Select All** menu selects all lines in the spreadsheet.

In the map view, the **Select All** menu is used to highlight all elements in a particular category. A third menu appears with the following categories: well (in map views), head linesink (in model views), flux linesink (in model views), pond (in model views), streamline (in model views), particle (in model views), target (in model views), title, line, frame, legend, parameter, or symbol. You may want to select all instances of a particular element in order to cut/copy and then paste into another document.

Select Data Bounds

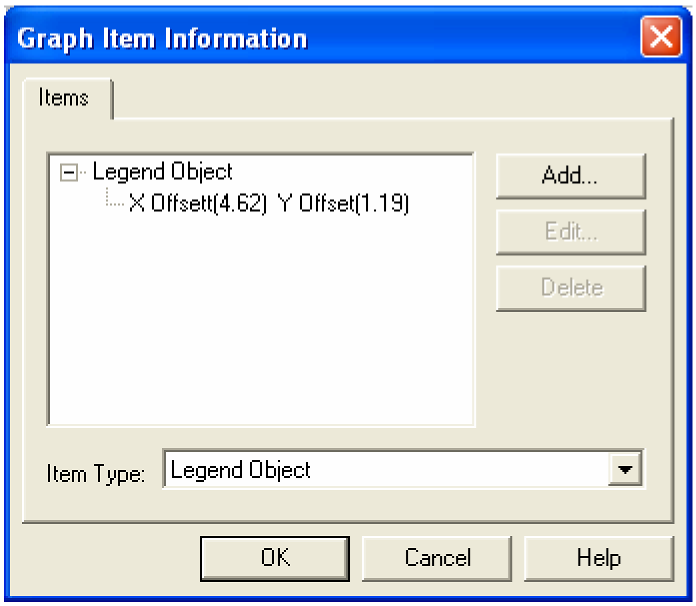
The **Select Data Bounds** menu is only available for the match view and is used to drag a rectangle around data points on the graph to select them in the spreadsheet. In a multiple well analysis, the spreadsheet contains the data for the current well and selecting data bounds operates on the data for the current well. The most common use for this operations is to visually select data points and change the weight and/or symbol; this requires that these columns have been added to the spreadsheet so they can be edited.

Graph Items/Map Items/Analytic Elements

The **Graph Items** menu is displayed in graph views while the **Map Items** menu is displayed in map views. In the case of a **Flow Model** document, the map view displays the **Analytic Elements** menu. They are virtually identical except that the map view versions allows you to edit well data and/or analytic elements in addition to the other types of items.

The *Graph Items Information* property sheet is shown below although the *Map Item Information* property sheet and *Analytic Element Information* property sheet are identical except for the title. This menu selection is used to manually add, delete, or edit objects in the currently active view. These objects include legends, lines, parameters, frames, symbols, and titles. In the case of a map view, wells, head linesinks, flux linesinks, ponds, streamlines, particles, line clusters, circle clusters, and targets are available as well.

The items are categorized by item type and individual items are identified based on their primary X and Y coordinates. As discussed in the concepts section, the coordinate system is map units or scaled inches for map views and graph views respectively. All editing is done using the property sheet specific to the item.

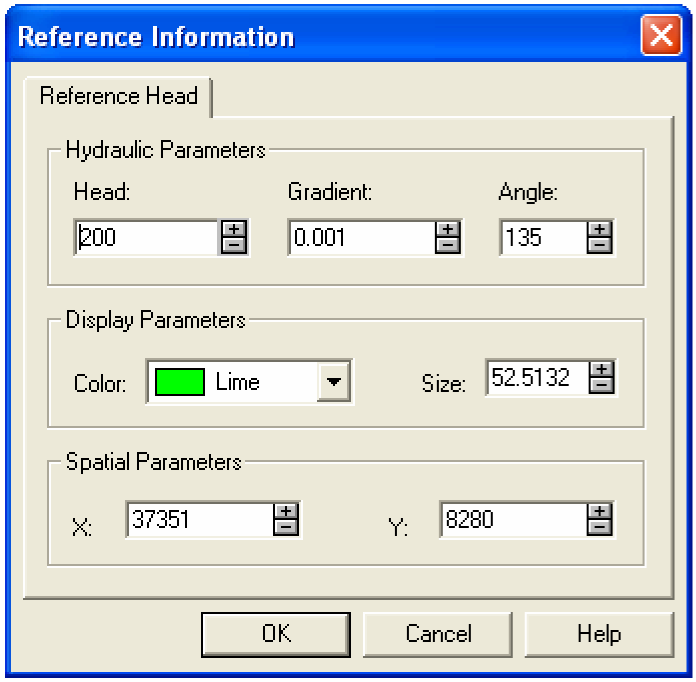


Add	Adds an item to the currently active graph. The item selected in *Item Type* is added.
Edit	Edits the currently selected item.
Delete	Deletes the currently selected item.
Item Type	Specifies the currently active item to be added when the **Add** button is clicked.

Reference Head

The **Reference Head** menu is used to edit the characteristics of the reference head for simulation documents. The reference head represents one location within the aquifer where the head is known. Reference head characteristics that may be edited in the *Reference Information* property sheet include the **X** and **Y** coordinates of the reference location, the head value, and the magnitude and direction of the regional hydraulic gradient. The angle of the regional gradient is entered in degrees with zero

degrees representing East, 90 degrees is North, etc. The regional gradient is dimensionless (ft/ft).



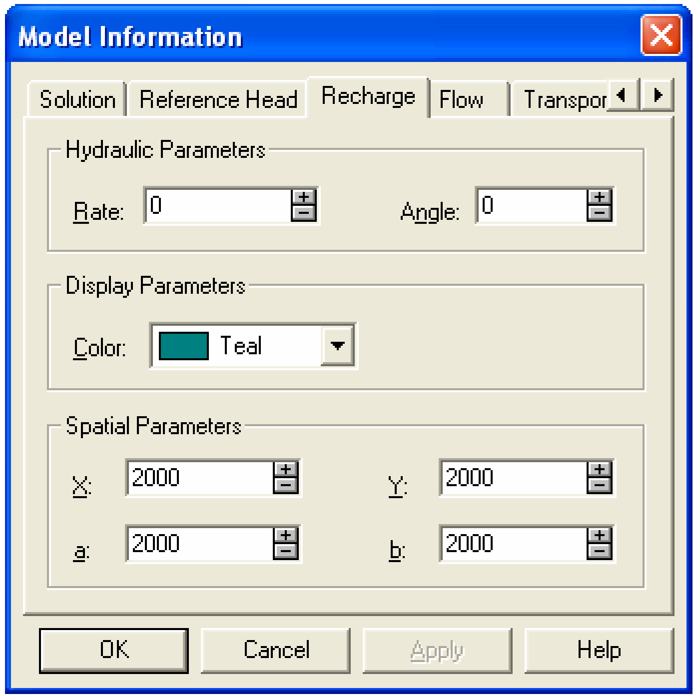
Head	The value for hydraulic head at a point from which to calculate initial head matrix
Gradient	The value for representing the change in hydraulic head with distance used to calculate the initial head matrix
Angle	The angle of rotation used to calculate the initial head matrix.
Display Parameters	
Color:	The color used to display the reference head symbol on the map
Size	The size in map units of the reference head symbol on the map
Spatial Parameters	
X:	The x-coordinate of the point at which the value of hydraulic head is being set
Y	The y-coordinate of the point at which the value of hydraulic head is being set

Recharge Ellipse

The **Recharge Ellipse** menu displays the *Recharge Information* property sheet used to change the characteristics of the recharge rate and location of the recharge ellipse for selected flow models. The property sheet allows you to specify the recharge rate (e.g., ft/d), the angle of the recharge ellipse axis (zero degrees is parallel to the X axis), the X and Y coordinates of the center of the ellipse, and the axes lengths (a and b) of the ellipse. Note that the size of the ellipse does not effect the ultimate shape of

the water table mound that develops in response to recharge. Only the aspect ratio (a/b) effects the shape of the recharge mound.

Recharge is currently only used in the steady-state WinFlow model and the WinTran model. It is primarily for regional simulations in which model boundaries are simulated with wells and linesinks. In this case, the recharge then creates water table divides and mounds between these regional features. Recharge should not be used for small-scale models such as those used to simulate remediation systems around gasoline stations and other similar sites.



Hydraulic Parameters

Rate	Sets the recharge rate in units of length/time
Angle	Set the angle of the recharge ellipse axis (zero degrees is parallel to the X axis)

Display Parameters

Color:	The color used to display the recharge ellipse on the map

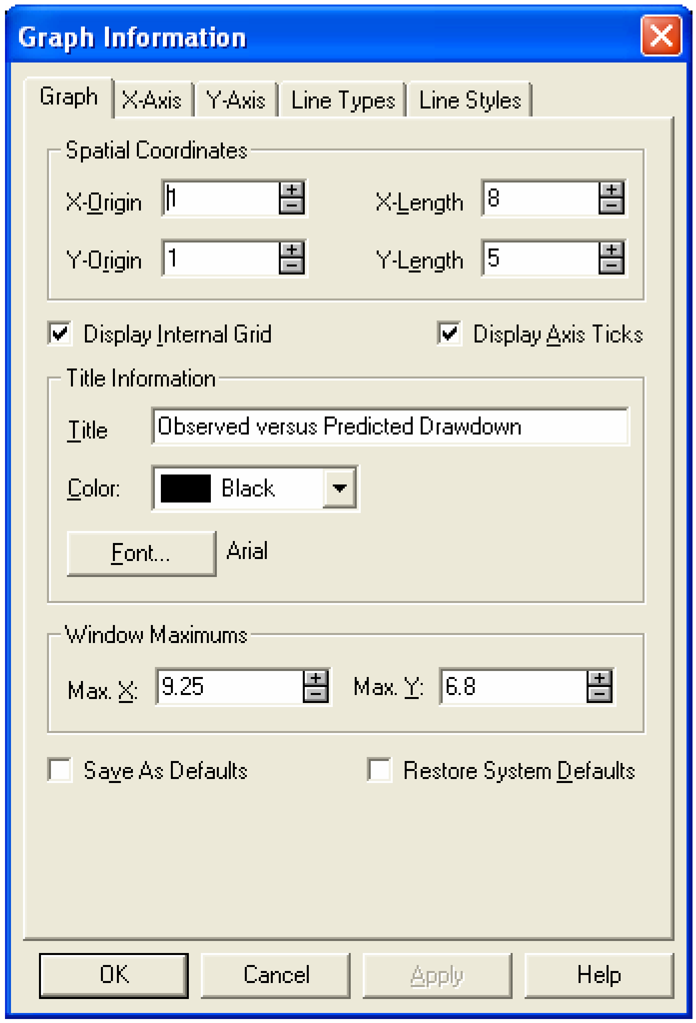
Spatial Parameters

X:	The x-coordinate of the center point of the ellipse
Y:	The y-coordinate of the center point of the ellipse
a:	The x-axis length of the ellipse
b:	The y-axis length of the ellipse

NOTE: Only the aspect ratio (a/b) effects the shape of the recharge mound.

Graph/Default Well Graph

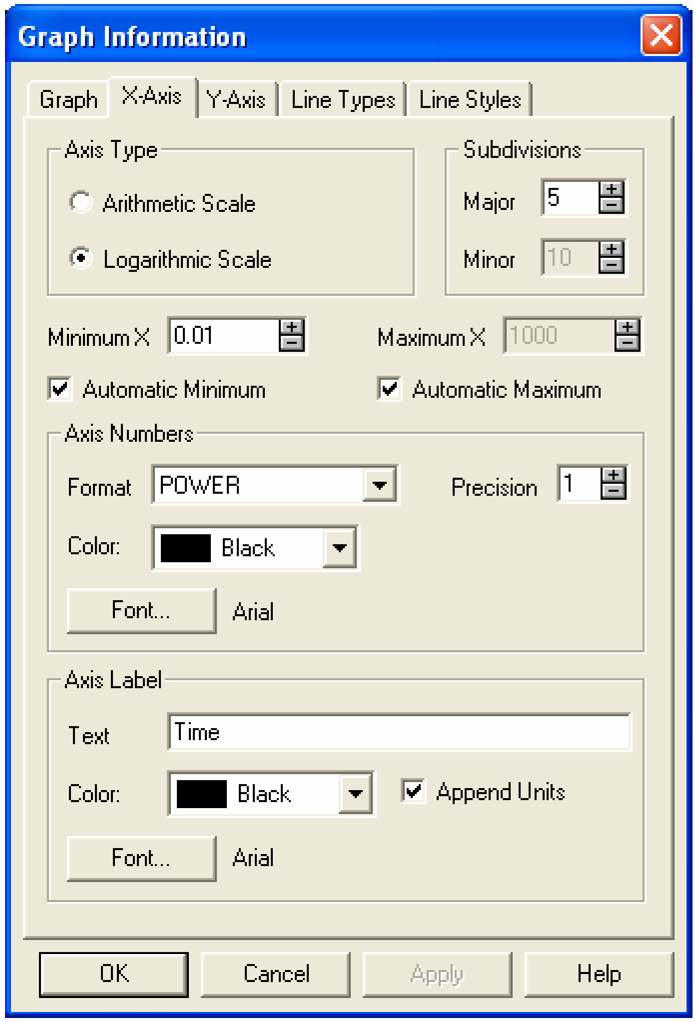
The **Graph, Default Well Graph, Default Line Graph and Default Concentration Graph** menus all control the characteristics of a graph. The *Graph Information* property sheet is displayed with several tabs to allow you to modify the way graphs are drawn on the screen. The **Graph** menu affects the active graph view while the **Default Well Graph** menu only affects the default parameters used when new wells are added to the aquifer test or model. Similarly, the **Default Line Graph** affects the default parameters used when adding new line calculation elements. Finally, the **Default Concentration Graph** affects the default parameters used when creating concentration versus time graphs for monitored wells. You may change the axis annotation, line types, line labels, graph paper, and fonts used on the graph. The following describes each of the items you may edit.



Spatial Coordinates

X-Origin	Horizontal offset of lower left corner of the graph (inches)
X-Length	Length of graph x-axis (inches)
Y-Origin	Vertical offset of lower left corner of the graph (inches)
Y-Length	Height of graph y-axis (inches)

Display Internal Grid	Toggle the display of the internal grid on the graph
Display Axis Ticks	Toggle the display of axis ticks adjacent to axis numbers
Title Information	
Title	The text to be displayed as the title above the graph
Color	The color used for the title
Font	Defines the font, font style, size, effects and color for the title
Window Maximums	
Max. X	Defines the maximum x value, in inches, to reserve for the graph window
Max. Y	Defines the maximum y value, in inches, to reserve for the graph window
Save As Defaults	When checked, the values contained in this property sheet are stored in the system registry when the property sheet is accepted and are used as the defaults when an new document is created
Restore System Defaults	When checked, the default system values are restored when the property sheet is accepted; this operation occurs before the **Save As Defaults** operation so having both checked resets both the document and the system registry



Axis Type

Arithmetic Scale:	Defines an arithmetic x-axis
Logarithmic Scale:	Defines a logarithmic x-axis

Subdivisions

Major:	Sets the number of major subdivisions for the graph
Minor:	Sets the number of minor subdivisions for an arithmetic axis (always 10 for a logarithmic axis)
Minimum X:	Sets the minimum value represented by the x-axis
Maximum X:	Sets the maximum value represented by the x-axis

NOTE: To reverse the axis numbers on an arithmetic scale, set the Minimum X greater than the Maximum X.

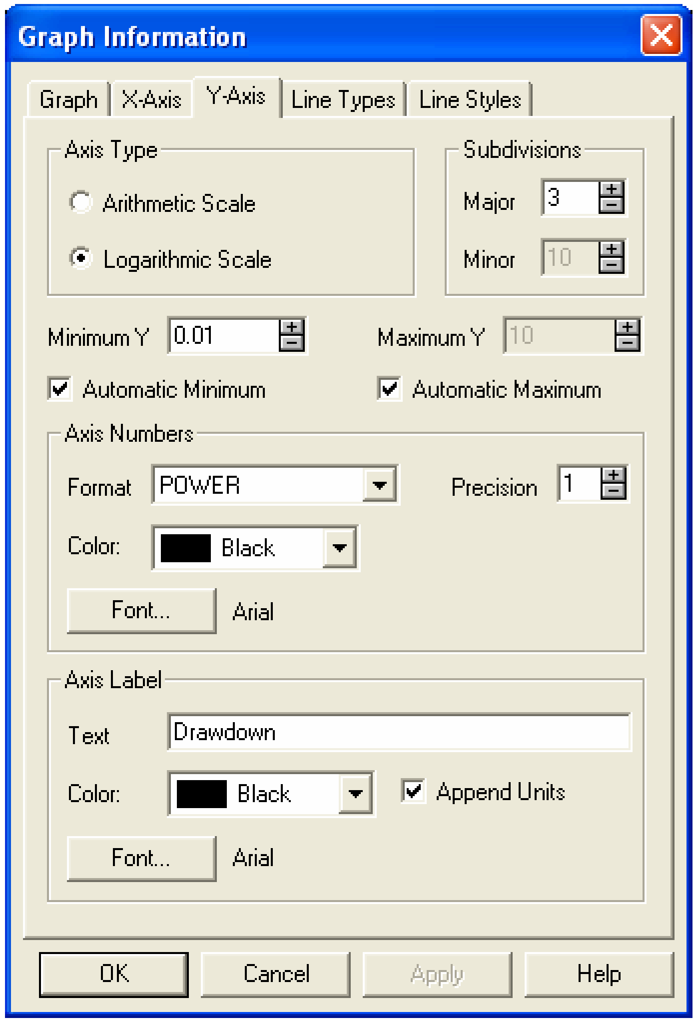
Automatic Minimum	When checked, the minimum axis value is automatically set based on the data being graphed
Automatic Maximum	When checked, the minimum axis value is automatically set based on the data being graphed

Axis Numbers

Format:	Defines the format to use for axis numbers
EXPONENTIAL	generates numbers as 1.0e-001
FIXED	generates numbers as 0.1
POWER	generates numbers as 10^{-1}
Precision:	Defines the number of decimal places to display for FIXED and EXPONENTIAL formats
Color	Defines the color to use when displaying the axis numbers
Font:	Defines the font, font style, size, effects and color for the axis numbers

Axis Label

Text:	The text to be displayed as the x-axis label
Color	Defines the color to use when displaying the axis numbers
Append Units	When checked, the units of the data will be appended to the axis label
Font:	Defines the font, font style, size, effects and color for the label



Axis Type

Arithmetic Scale:	Defines an arithmetic x-axis
Logarithmic Scale:	Defines a logarithmic x-axis

Subdivisions

Major:	Sets the number of major subdivisions for the graph
Minor:	Sets the number of minor subdivisions for an arithmetic axis (always 10 for a logarithmic axis)
Minimum X:	Sets the minimum value represented by the x-axis
Maximum X:	Sets the maximum value represented by the x-axis

NOTE: To reverse the axis numbers on an arithmetic scale, set the Minimum X greater than the Maximum X.

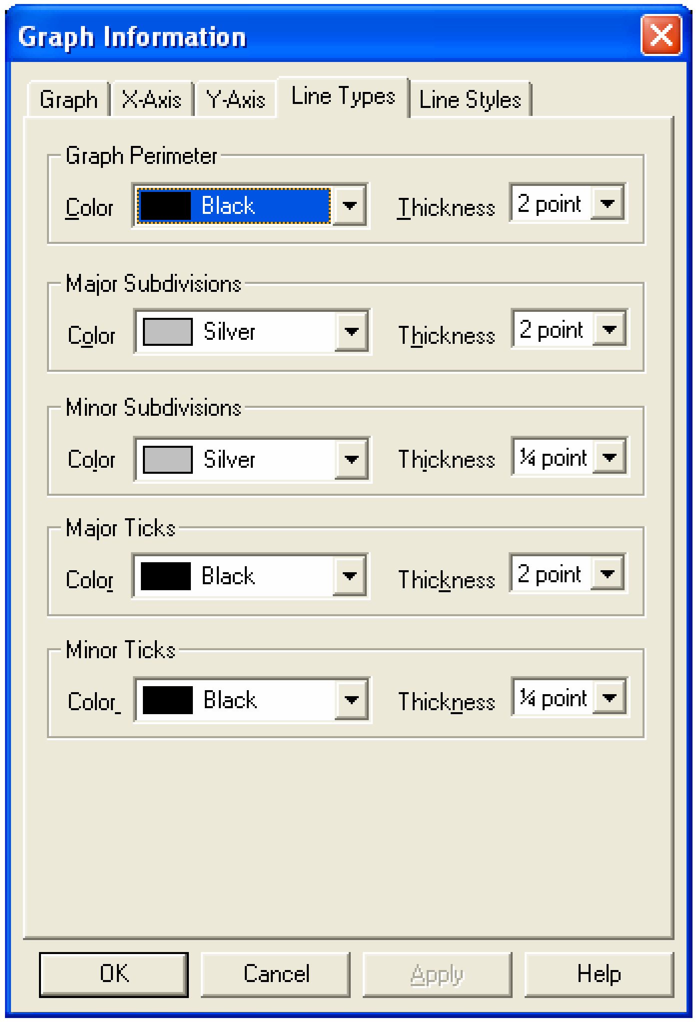
Automatic Minimum	When checked, the minimum axis value is automatically set based on the data being graphed
Automatic Maximum	When checked, the minimum axis value is automatically set based on the data being graphed

Axis Numbers

Format:	Defines the format to use for axis numbers
EXPONENTIAL	generates numbers as 1.0e-001
FIXED	generates numbers as 0.1
POWER	generates numbers as 10^{-1}
Precision:	Defines the number of decimal places to display for FIXED and EXPONENTIAL formats
Color	Defines the color to use when displaying the axis numbers
Font:	Defines the font, font style, size, effects and color for the axis numbers

Axis Label

Text:	The text to be displayed as the x-axis label
Color	Defines the color to use when displaying the axis numbers
Append Units	When checked, the units of the data will be appended to the axis label
Font:	Defines the font, font style, size, effects and color for the label



Graph Perimeter

Color:	Sets the color of the graph perimeter line
Thickness:	Sets the thickness of the graph perimeter line in points

Major Subdivisions

Color:	Sets the color of the graph major subdivisions
Thickness:	Sets the thickness of the graph major subdivisions in points

Minor Subdivisions

Color:	Sets the color of the graph minor subdivisions
Thickness:	Sets the thickness of the graph minor subdivisions

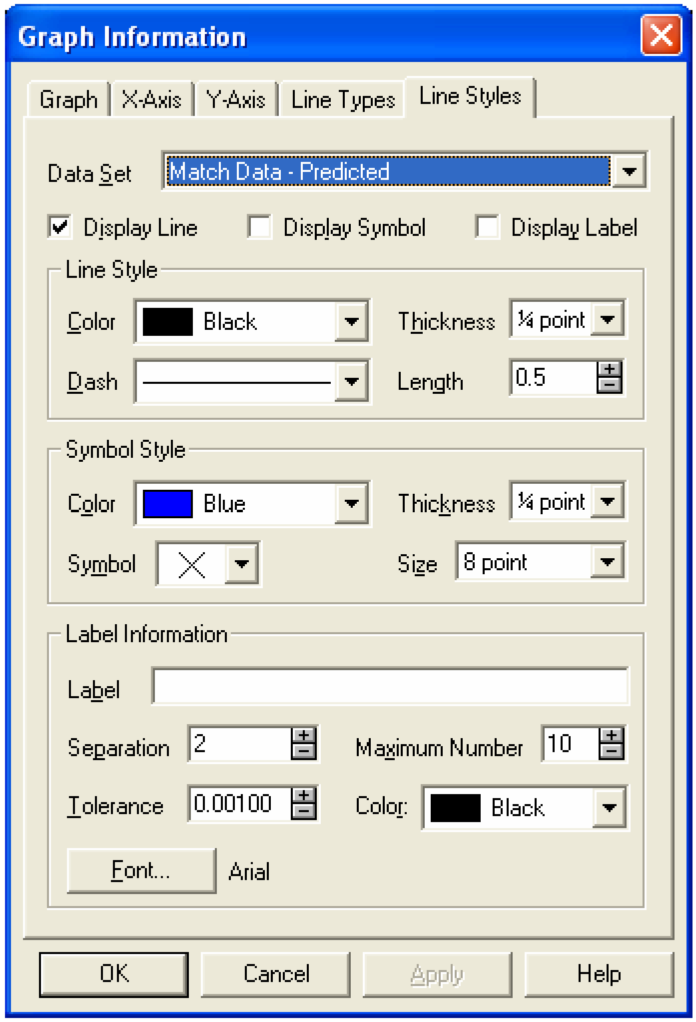
Major Ticks

Color:	Sets the color of the graph major ticks
Thickness:	Sets the thickness of the graph major ticks in points

Minor Subdivisions

Color:	Sets the color of the graph minor ticks

Thickness: Sets the thickness of the graph minor ticks



Data Set: Specifies which graph data set is being edited

Default Type Curves - Defines the values used when a new type curve is added to the graph

Selected Type Curve - Defines the Line Style parameters used when displaying the selected type curve

Display Line: Activates the display of lines connecting data points

Display Symbol: Activates the display of symbols at data points

Display Label: Activates the display of a label on the data set

Line Style

Color: Sets the color of the line connecting the data points

Thickness: Sets the thickness in points of the line connecting the data points

Dash: Sets the dash pattern to use for the line connecting the data points

Length:	Sets the length in inches of the dash pattern for the line connecting the data points

Symbol Style

Color:	Sets the color of the symbol displayed at each data point
Thickness:	Sets the thickness in points of the line used to draw the symbol
Symbol:	Sets the symbol to display at each data point
Size:	Sets the size in points of the symbol to display at each data point

Label Information

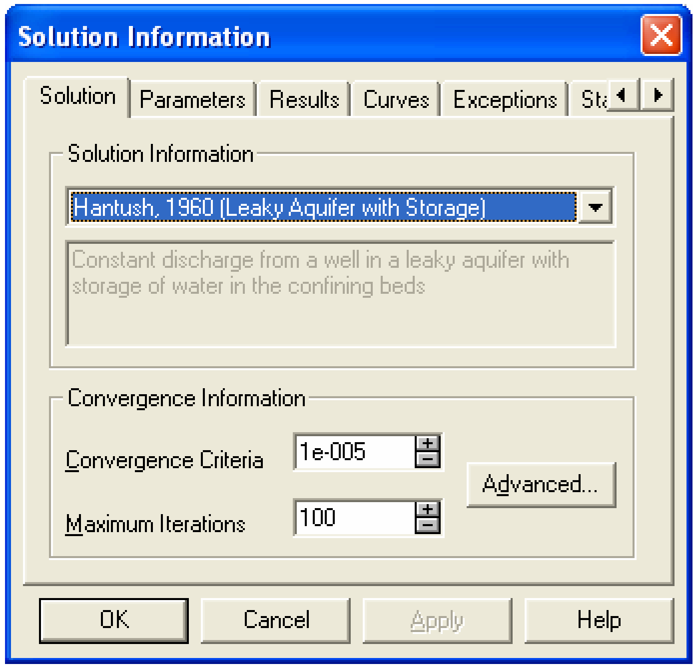
Label:	The text to be displayed as the label on the data set
Separation:	The minimum distance in inches between labels
Maximum Number:	The maximum number of labels to display
Tolerance:	The maximum difference in inches between the total distance along the data curve where the label is to be displayed and the length of the label
Color	Sets the color of the label displayed on the data set

NOTE: The value for tolerance determines how straight the section of the curve has to be for the label to be displayed. If the tolerance is exceeded the label will be advanced forward until an acceptable section of the curve is found.

Font:	Defines the font, font style, size, effects and color for the label

Solution

The **Solution** menu activates the *Solution Information* property sheet which allows you to change the analysis type (i.e., confined versus leaky aquifers etc.), the way in which the automatic curve match is performed, initial guesses for parameter values, the way data are displayed, and the type curves to be displayed. You can also use this property sheet to view the results of the current match. The *Solution Information* property sheet is dynamic and changes depending on the specific analysis selected. You are referred to the Analysis Parameters section of the Concepts chapter for more details. A general discussion of each tab of the property sheet is provided below.

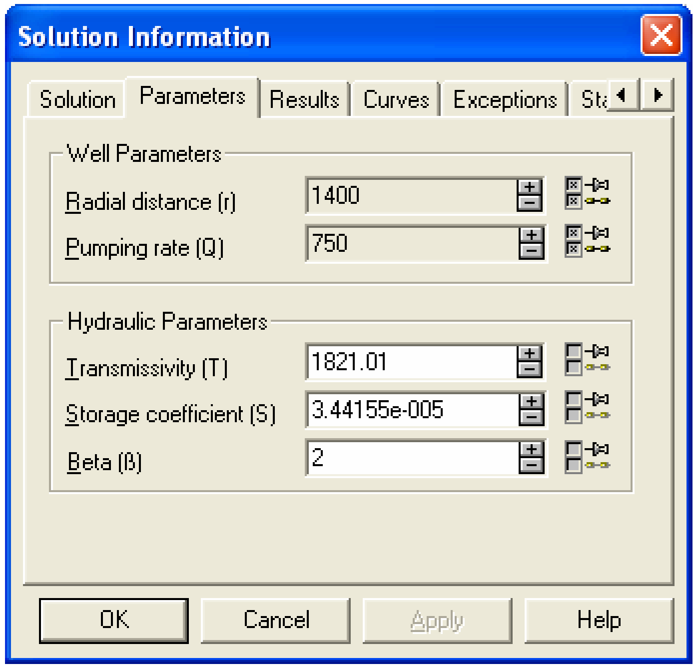


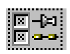
Solution Information

Analysis:	Defines the type of analytic solution to use when evaluating the current aquifer test

Convergence Information

Convergence Criteria:	The change in the sum of squared residuals that determines when the automated curve match algorithm is finished
Maximum Iterations:	The maximum number of nonlinear least-squares iterations allowed for each automated curve match.
Advanced:	This button activates the *Advanced Solution Information* dialog that provides more control of the nonlinear least-squares solution



The  control adjacent to each edit field acts like two check-boxes. The upper check-box controls whether the adjacent parameter is fixed, held constant, during optimization. The lower check-box controls whether the particular parameter is linked to another parameter within the document. Linked parameters are not editable and share both value and units with the parameter to which they are linked. Furthermore, linked parameters are forced to be fixed.

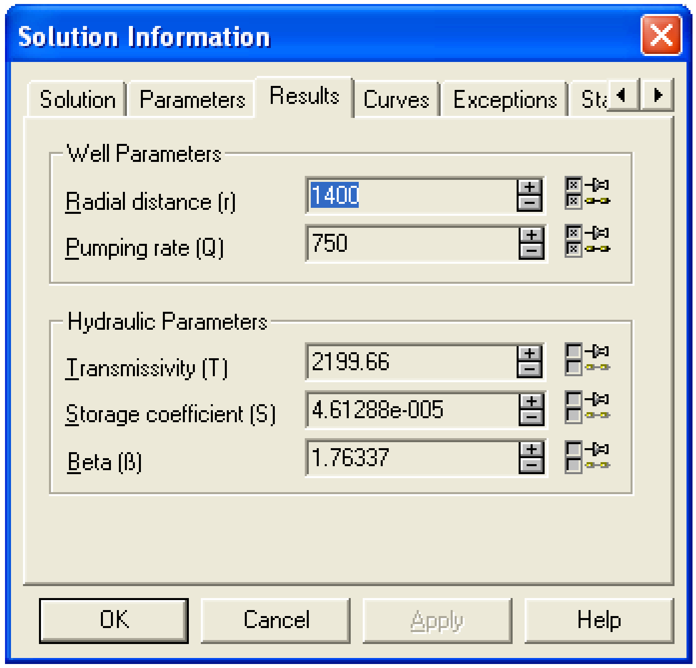
Well Parameters

Radial distance (r)	Initial guess for distance from the pumping well to the observation well
Pumping Rate (Q)	Initial guess for pumping rate at the production well

Hydraulic Parameters

Transmissivity	Initial guess for transmissivity
Storage coefficient	Initial guess for storage coefficient
Beta (ß)	Initial guess for Beta

NOTE: These example parameters are specific to the Hantush, 1960 analysis. Additional parameters and/or parameter pages may appear in this area depending upon the type of aquifer test analysis specified.



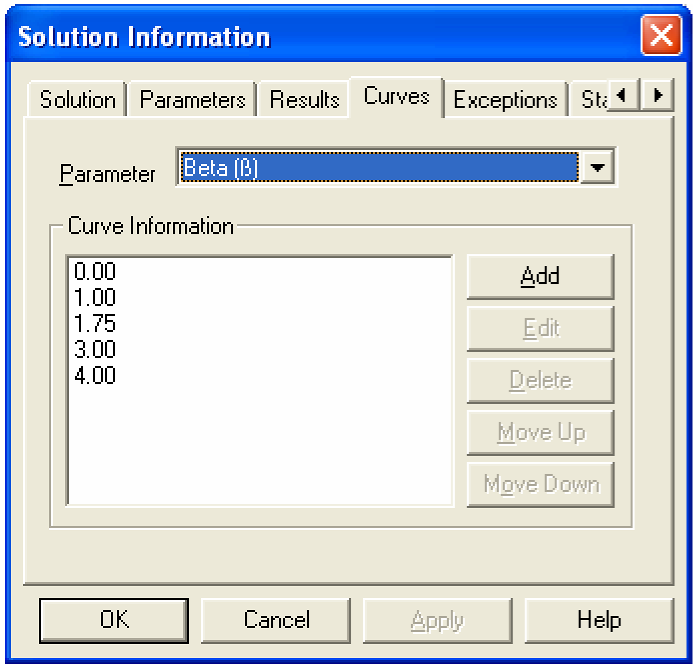
Well Parameters

Radial distance (r)	Calculated value for distance from the pumping well to the observation well
Pumping Rate (Q)	Calculated value for pumping rate at the production well

Hydraulic Parameters

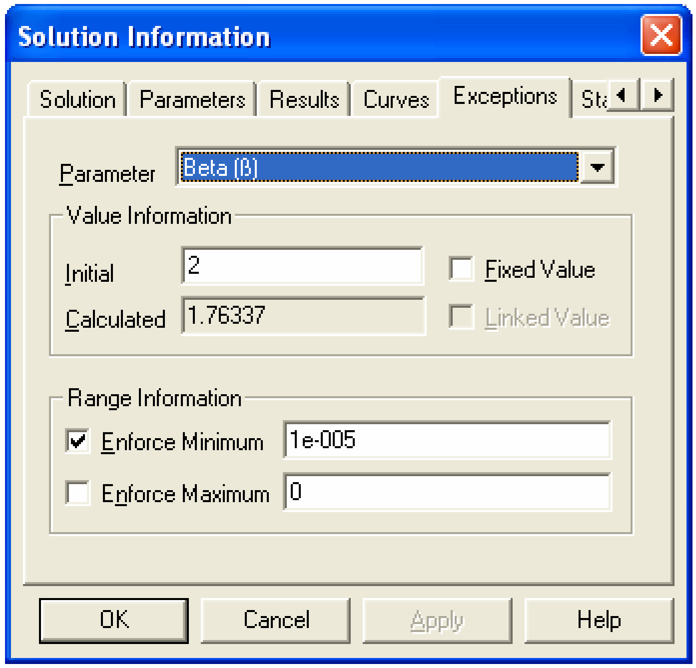
Transmissivity	Calculated value for transmissivity
Storage coefficient	Calculated value for storage coefficient
Beta (ß)	Calculated value for Beta

NOTE: These example results are specific to the Hantush, 1960 analysis. Additional parameters and/or results pages may appear in this area depending upon the type of aquifer test analysis specified.



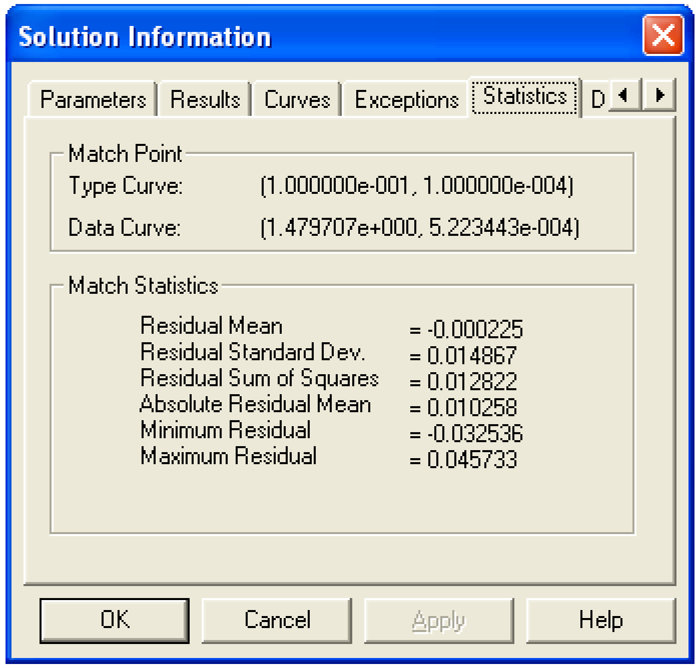
Parameter	Defines the parameter used when generating type curves
Curve Information	
Add	Add a new value for which to generate a type curve
Edit	Change the currently selected type curve value
Delete	Remove the currently selected type curve value
Move Up	Move the currently selected type curve value up in the list
Move Down	Move the currently selected type curve value down in the list

NOTE: This page may be absent if the active solution does not support multiple type curves. Also, the order in the list controls the order the curves are drawn. In some cases, the selected type curve color is overwritten when type curves are drawn. In this case, move the selected type curve to the bottom of the list.

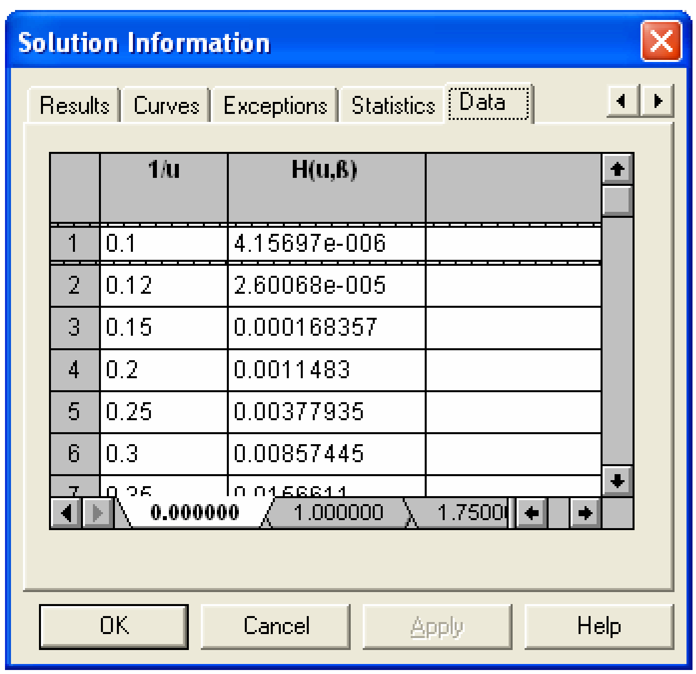


Parameter	Selects the parameter to edit
Value Information	
Initial	Initial guess to use for the selected parameter
Fixed Value	If checked, the value is fixed for the selected parameter during optimization
Calculated	Calculated value for the selected parameter
Linked Value	If checked, the value is linked to a parameter elsewhere in the document. Linked values are linked in both value and units and force the **Fixed Value** check-box to be checked.
Range Information	
Enforce Minimum	If checked, the adjacent value is the minimum value allowed during optimization
Enforce Maximum	If checked, the adjacent value is the maximum value allowed during optimization

NOTE: It is often necessary to enforce minimum and maximum values for parameters to achieve convergence on a solution.



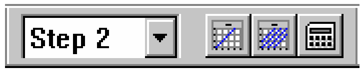
The ***Statistics*** tab displays the results from the current match. A residual is the difference between the field drawdown and the value computed according to the current match. The automatic method tries to minimize the residual sum of squares. The Match Point data are useful if you are trying to compare your results with those obtained from more traditional graph paper methods.

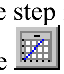


The ***Data*** tab contains a tab spreadsheet containing the data points used to represent each type curve. The spreadsheet is read-only; however, a context menu is available to export the data points for individual type curves.

Toggle Step

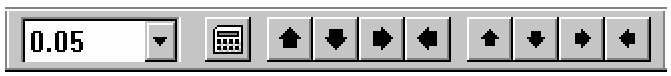
The **Toggle Step** option is active only in **Step Test** documents. Selecting this item changes the currently selected step and updates the combobox in the Step toolbar shown below.



The currently active step controls which subset of data points and their respective linear regression line are active. The currently active step is used when either the **Calc->Linear Regression->Single Step** menu or the  button on the Step toolbar is clicked.

Toggle Type Curve

Selecting the **Toggle Type Curve** menu changes the currently selected type curve when there are multiple type curves being displayed. It also updates the combobox in the Match toolbar as shown below.



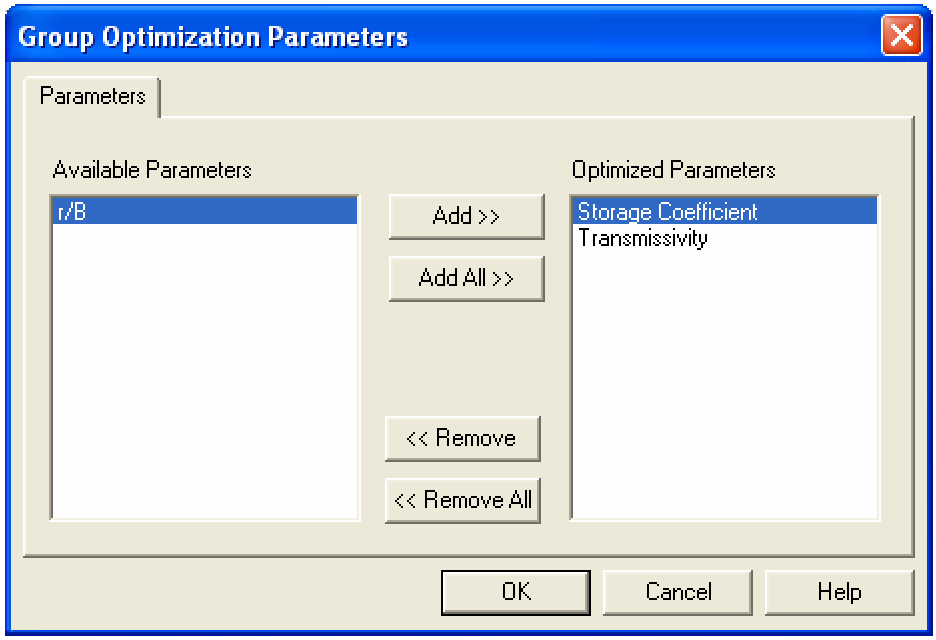
For single type curve analyses (e.g., the Theis method for confined aquifers), this menu is not active. Further, this menu is only available when the Match Data view is active.

Toggle Data Set

Selecting the **Toggle Data Set** menu changes the currently selected well in a multiple well analysis. The data points for the currently selected well are displayed in the spreadsheet and are active in the Match Data view. Both automatic and visual matching use the active well. The name of the currently selected well is displayed on the Match Data view tab at the bottom left of the graph view. This menu is only available in a multiple well analysis.

Group Optimize Parameters

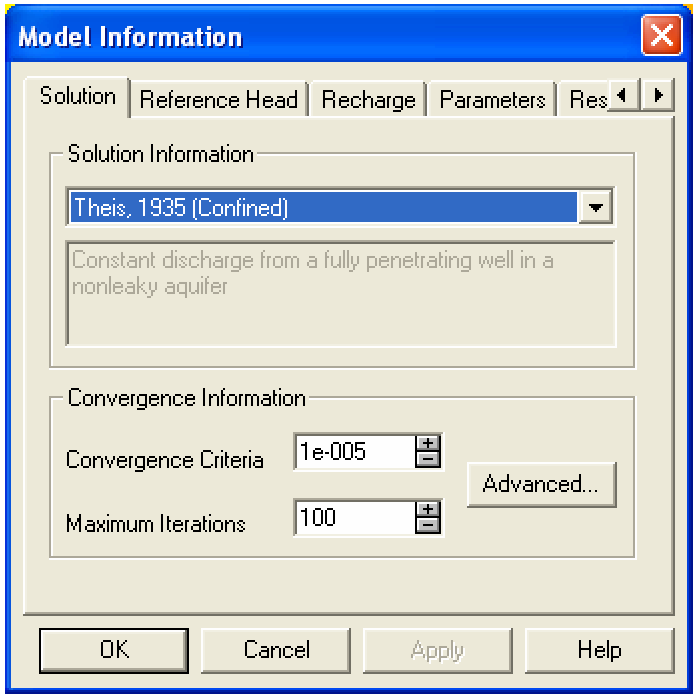
Selecting the **Group Optimize Parameters** menu displays the *Group Optimization Parameters* property sheet. This property sheet has a ***Parameters*** tab containing an Add/Remove listbox pair. This menu is only available in multiple well analyses. Optimizing the wells as a group uses data from all wells in estimating the parameter values for the automatic match. The *Group Optimization Parameters* property sheet is displayed below.



Available Parameters:	Contains the parameters that can optimized for all wells. In order for a parameter to appear in either list, it cannot be fixed in any well in the group.
Optimized Parameters:	Contains the parameters that will be optimized across the group of wells.
Add>>:	Moves the currently selected parameter from the *Available Parameters* list to the *Optimized Parameters* list.
Add All>>:	Moves all parameters from the *Available Parameters* list to the *Optimized Parameters* list.
Remove>>:	Moves the currently selected parameter from the *Optimized Parameters* list to the *Available Parameters* list.
Remove All>>:	Moves all parameters from the *Optimized Parameters* list to the *Available Parameters* list.

Flow Model

The **Flow Model** menu accesses the *Model Information* property sheet which allows you to change the solution type (i.e., confined versus leaky aquifers etc.), the way in which the model parameters are optimized, model parameter values, the reference head and the recharge ellipse. The *Model Information* property sheet is dynamic and changes depending on the specific analysis selected. If a solution does not support optimization, the Results and Exceptions pages will not be present. Additionally, the Parameter and Results pages have help links to the specific solution help files accessed by pressing the **F1** key. A general discussion of each tab of the property sheet is provided below.

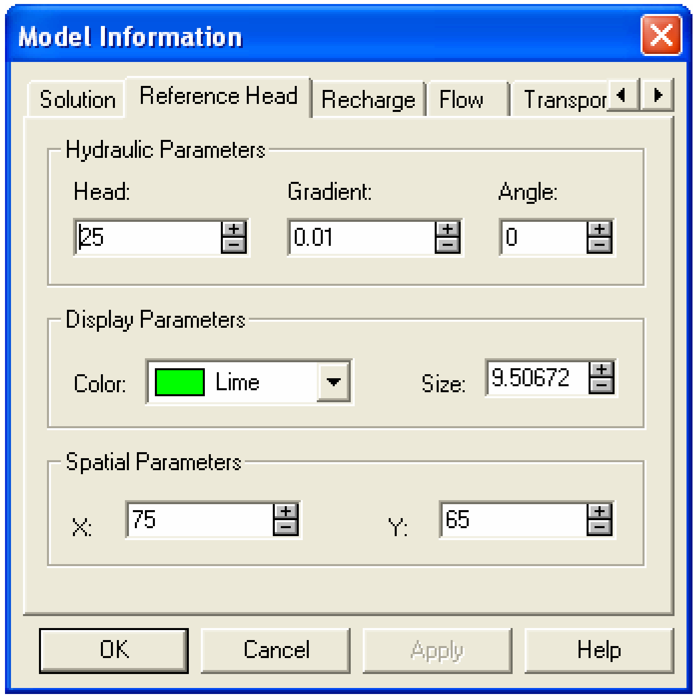


Solution Information

Analysis:	Defines the type of analytic solution to use when evaluating the current aquifer test

Convergence Information

Convergence Criteria:	The change in the sum of squared residuals that determines when the automated curve match algorithm is finished
Maximum Iterations:	The maximum number of nonlinear least-squares iterations allowed for each automated curve match.
Advanced:	This button activates the *Advanced Solution Information* dialog that provides more control of the nonlinear least-squares solution



The ***Reference Head*** tab is used to edit the characteristics of the reference head for simulation and flow model documents. The reference head represents one location within the aquifer where the head is known. Reference head characteristics that may be edited in this property sheet include the **X** and **Y** coordinates of the reference location, the head value, and the magnitude and direction of the regional hydraulic gradient. The angle of the regional gradient is entered in degrees with zero degrees representing East, 90 degrees is North, etc. The regional gradient is dimensionless (ft/ft).

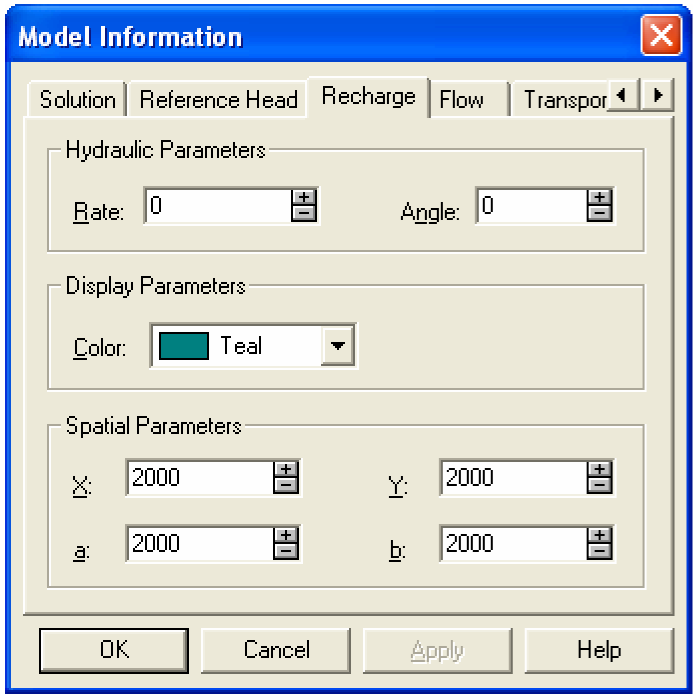
Head:	The value for hydraulic head at a point from which to calculate initial head matrix
Gradient:	The value for representing the change in hydraulic head with distance used to calculate the initial head matrix
Angle:	The angle of rotation used to calculate the initial head matrix.

Display Parameters

Color:	The color used to display the reference head symbol on the map
Size:	The size in map units of the reference head symbol on the map

Spatial Parameters

X:	The x-coordinate of the point at which the value of hydraulic head is being set
Y:	The y-coordinate of the point at which the value of hydraulic head is being set



The **Recharge** tab is used to change the characteristics of the recharge rate and location of the recharge ellipse for selected flow models. The property sheet allows you to specify the recharge rate (e.g., ft/d), the angle of the recharge ellipse axis (zero degrees is parallel to the X axis), the X and Y coordinates of the center of the ellipse, and the axes lengths (a and b) of the ellipse. Note that the size of the ellipse does not affect the ultimate shape of the water table mound that develops in response to recharge. Only the aspect ratio (a/b) effects the shape of the recharge mound.

Recharge is currently only used in the steady-state WinFlow model and the WinTran model. It is primarily for regional simulations in which model boundaries are simulated with wells and linesinks. In this case, the recharge then creates water table divides and mounds between these regional features. Recharge should not be used for small-scale models such as those used to simulate remediation systems around gasoline stations and other similar sites.

Hydraulic Parameters

Rate	Sets the recharge rate in units of length/time
Angle	Set the angle of the recharge ellipse axis (zero degrees is parallel to the X axis)

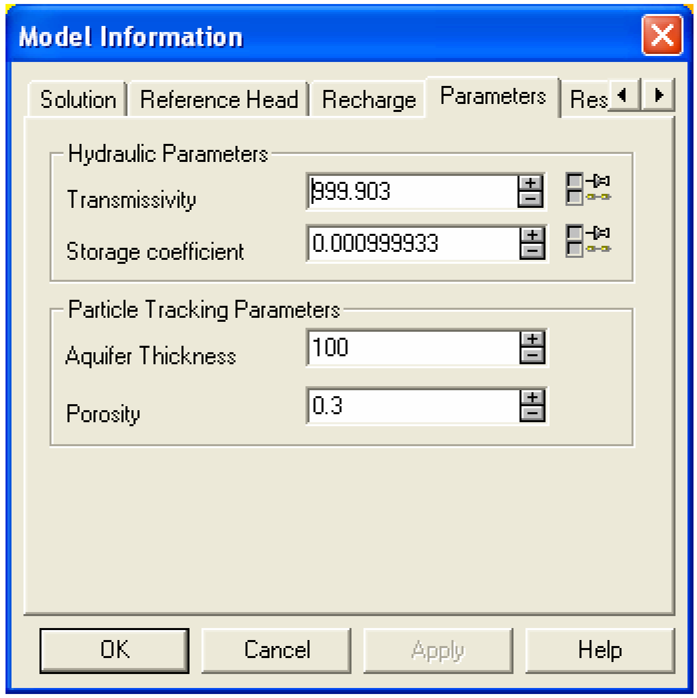
Display Parameters

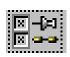
Color:	The color used to display the recharge ellipse on the map

Spatial Parameters

X:	The x-coordinate of the center point of the ellipse
Y:	The y-coordinate of the center point of the ellipse
a:	The x-axis length of the ellipse
b:	The y-axis length of the ellipse

NOTE: Only the aspect ratio (a/b) effects the shape of the recharge mound.



The  control adjacent to each edit field acts like two check-boxes. The upper check-box controls whether the adjacent parameter is fixed, held constant, during optimization. The lower check-box controls whether the particular parameter is linked to another parameter within the document. Linked parameters are not editable and share both value and units with the parameter to which they are linked. Furthermore, linked parameters are forced to be fixed.

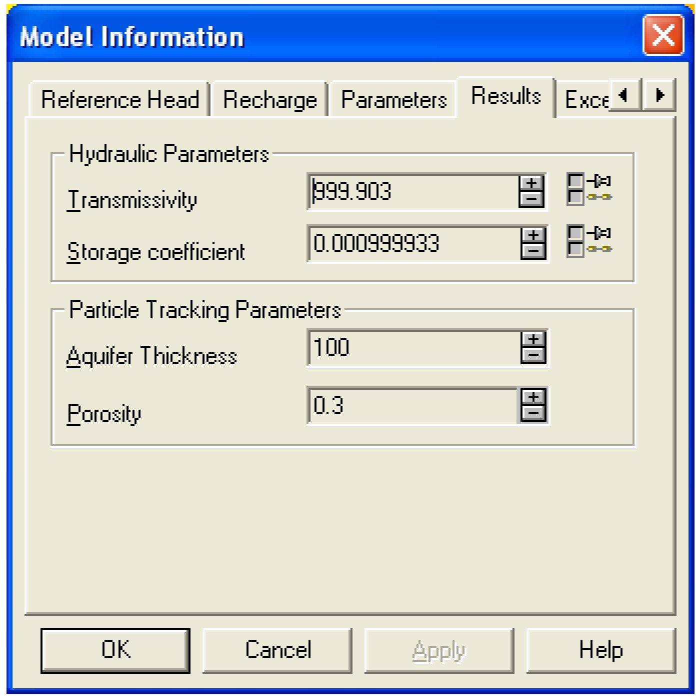
Hydraulic Parameters

Transmissivity	Value to use for transmissivity
Storage coefficient	Value to use for storage coefficient

Particle Tracking Parameters

Aquifer Thickness	Value to use when converting transmissivity to hydraulic conductivity for the purpose of calculating flow velocities for particle tracking
Porosity	Value to use for porosity for the purpose of calculating flow velocities for particle tracking

NOTE: These example parameters are specific to the Theis, 1935 analysis. Additional parameters and/or parameter pages may appear in this area depending upon the type of aquifer test analysis specified.



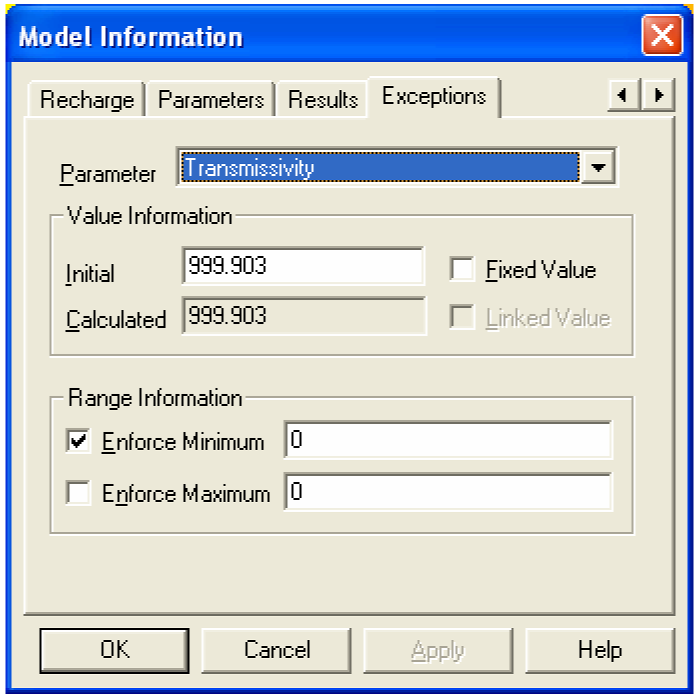
Hydraulic Parameters

Transmissivity	Calculated value for transmissivity
Storage coefficient	Calculated value for storage coefficient

Particle Tracking Parameters

Aquifer Thickness	Value used when converting transmissivity to hydraulic conductivity for the purpose of calculating flow velocities for particle tracking
Porosity	Value used for porosity for the purpose of calculating flow velocities for particle tracking

NOTE: **These example parameters are specific to the Theis, 1935 analysis. Additional parameters and/or parameter pages may appear in this area depending upon the type of aquifer test analysis specified.**

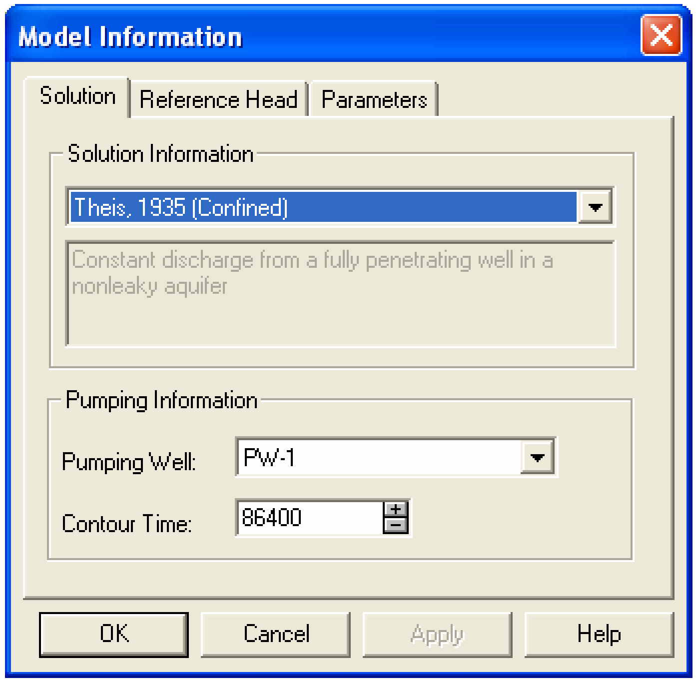


Parameter	Selects the parameter to edit
Value Information	
Initial	Initial guess to use for the selected parameter
Fixed Value	If checked, the value is fixed for the selected parameter during optimization
Calculated	Calculated value for the selected parameter
Linked Value	If checked, the value is linked to a parameter elsewhere in the document. Linked values are linked in both value and units and force the **Fixed Value** check-box to be checked.
Range Information	
Enforce Minimum	If checked, the adjacent value is the minimum value allowed during optimization
Enforce Maximum	If checked, the adjacent value is the maximum value allowed during optimization

NOTE: It is often necessary to enforce minimum and maximum values for parameters to achieve convergence on a solution.

Model

The **Model** menu accesses the *Model Information* property sheet which allows you to change the analysis type (i.e., confined versus leaky aquifers etc.), the pumping well, the reference head and the pertinent parameters for the simulation. The *Model Information* property sheet is dynamic and changes depending on the specific analysis selected. A general discussion of each tab of the property sheet is provided below.



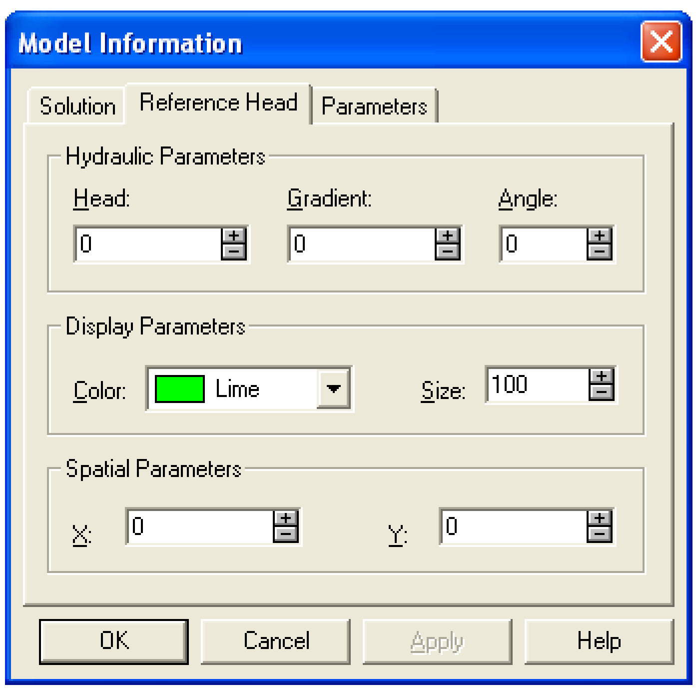
Solution Information

Analysis: Defines the type of analytic solution to use when simulating a pump test

Pumping Information

Pumping Well: Selects one of the available wells to serve as the pumping well for the simulation

Contour Time: The time at which a head matrix and contour map will be calculated



The ***Reference Head*** tab is used to edit the characteristics of the reference head for simulation documents. The reference head represents one location within the aquifer where the head is known. Reference head characteristics that may be edited in this property sheet include the **X** and **Y** coordinates of the reference location, the head value, and the magnitude and direction of the regional hydraulic gradient. The angle of the regional gradient is entered in degrees with zero degrees representing East, 90 degrees is North, etc. The regional gradient is dimensionless (ft/ft).

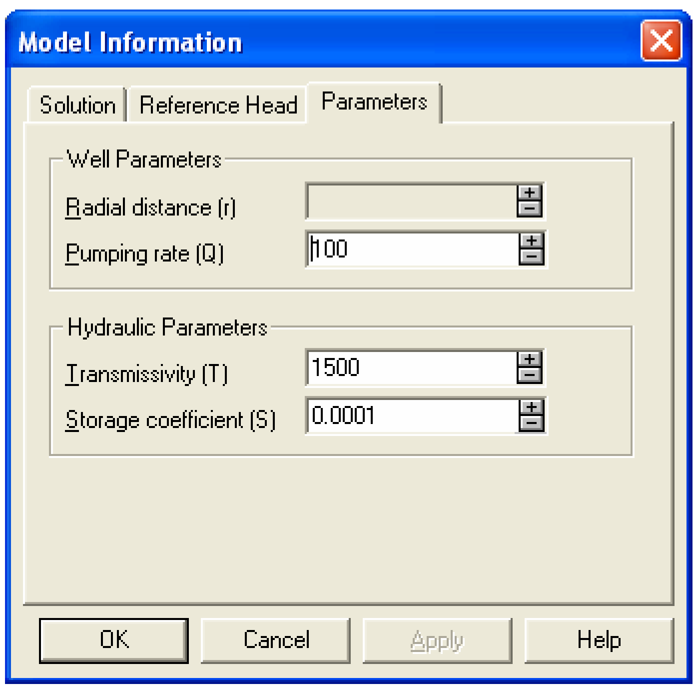
Head:	The value for hydraulic head at a point from which to calculate initial head matrix
Gradient:	The value for representing the change in hydraulic head with distance used to calculate the initial head matrix
Angle:	The angle of rotation used to calculate the initial head matrix.

Display Parameters

Color:	The color used to display the reference head symbol on the map
Size:	The size in map units of the reference head symbol on the map

Spatial Parameters

X:	The x-coordinate of the point at which the value of hydraulic head is being set
Y:	The y-coordinate of the point at which the value of hydraulic head is being set



Well Parameters

Radial distance (r)	In a simulation, this parameter is not used and is inactive

Pumping Rate (Q) — The constant pumping rate to use during the simulation

Hydraulic Parameters

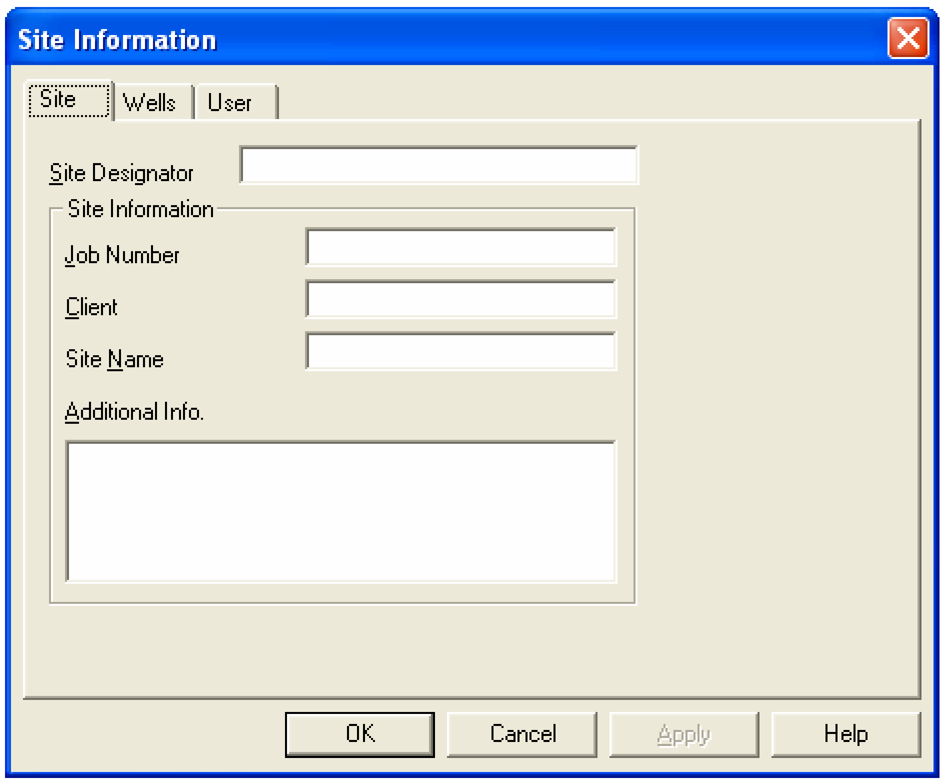
Transmissivity — Value to use for transmissivity

Storage coefficient — Value to use for storage coefficient

NOTE: These example parameters are specific to the Theis, 1935 analysis. Additional parameters and/or parameter pages may appear in this area depending upon the type of aquifer test analysis specified.

Site

The **Site** menu allows you to specify information describing the aquifer test site, including job number, client, etc. The menu item activates the *Site Information* property sheet. You may also specify the parameters describing each well used in the test, as described below.



Site Designator — User field that appears in the summary report and can be used to annotate a graph or map view.

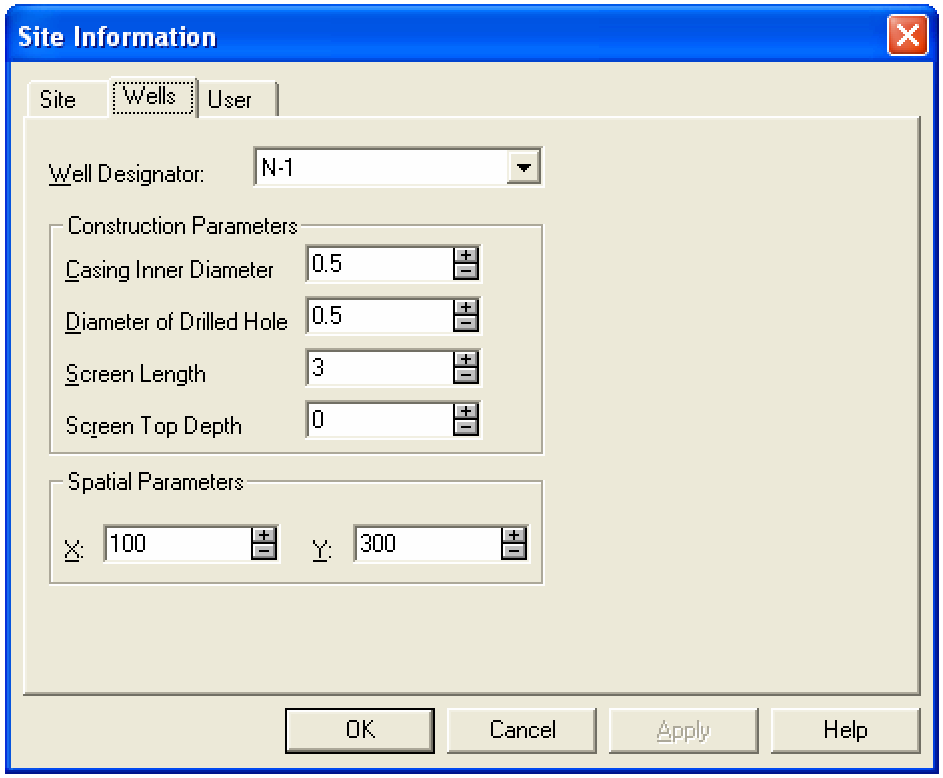
Site Information

Job Number — User field that appears in the summary report and can be used to annotate a graph or map view.

Client — User field that appears in the summary report and can be used to annotate a graph or map view.

Site Name — User field that appears in the summary report and can be used to annotate a graph or map view.

Additional Info:	User field that appears in the summary report and can be used to annotate a graph or map view.



Well Designator:	Specifies which well is being edited

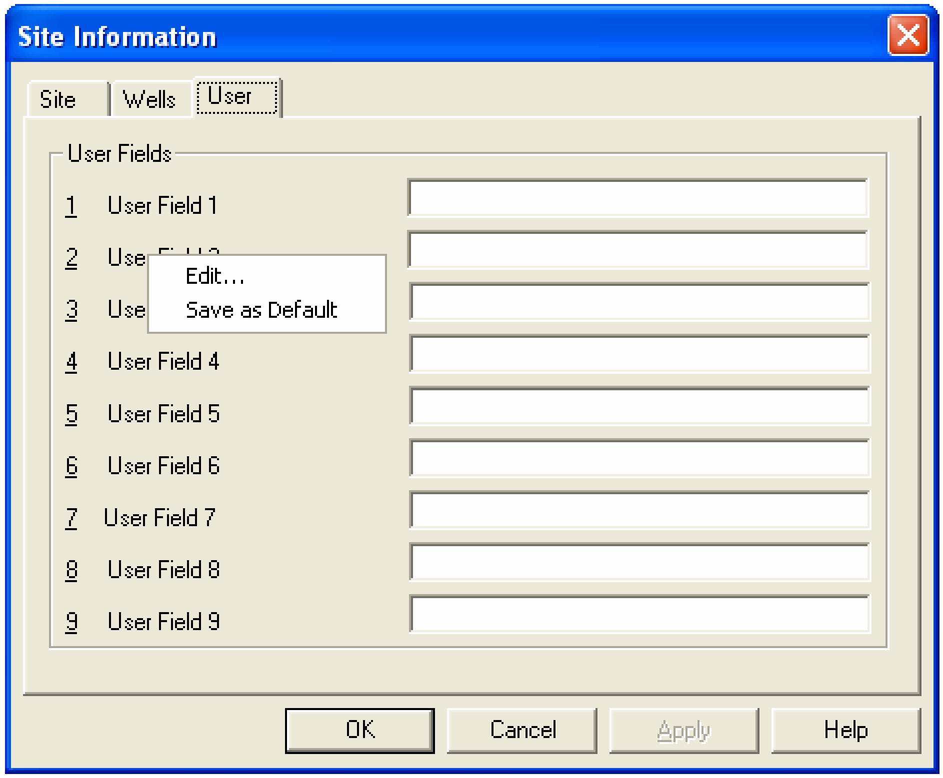
Construction Parameters

Casing Inner Diameter	The inner diameter of the well casing
Diameter of Drilled Hole	The diameter of the drilled hole
Screen Length	The length of the well screen
Screen Top Depth	The depth to top of the well screen relative to the top of the aquifer

Spatial Parameters

X:	The x-coordinate of the well location in map units
Y:	The y-coordinate of the well location in map units

NOTE: This tab is only shown when wells have been defined in the map view.

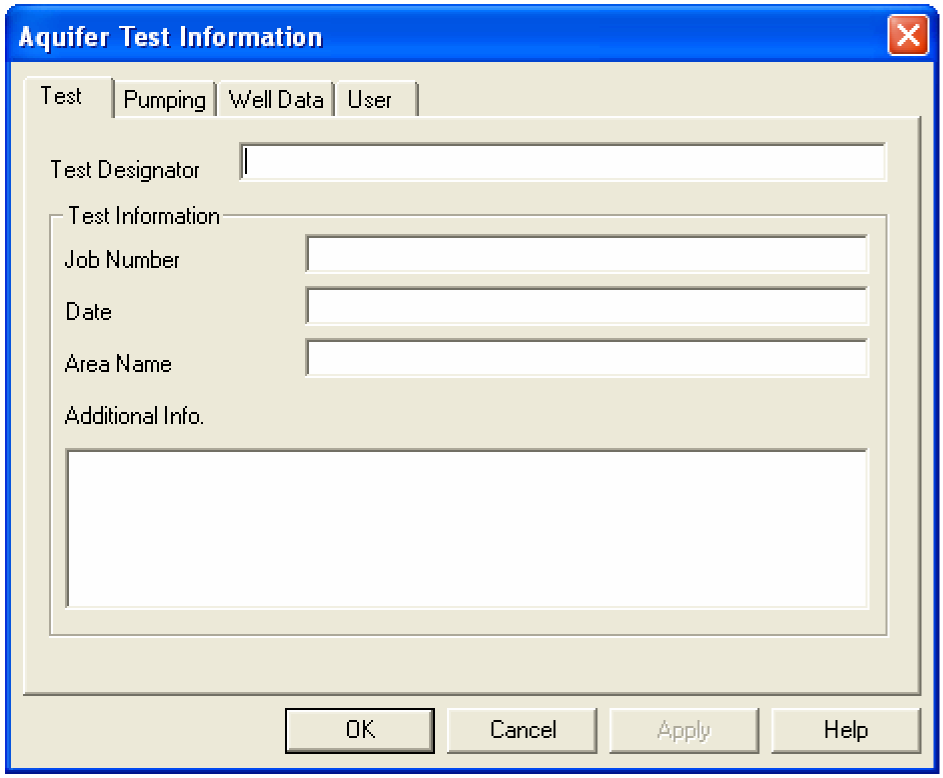


The **User** tab contains customizable parameter boxes which allow the input of a wide range of user specified data or text items. Each field initially has a default name which can be changed. Place the cursor over the field name to be changed and right click the mouse. On the context menu displayed, select **Edit** and enter the new name required. Click on **OK** and the new name will be saved. The required information can then be entered into the input section of each box.

Information entered in these fields can be accessed in footers/headers, figures and parameter boxes displayed on graphical output.

Aquifer Test

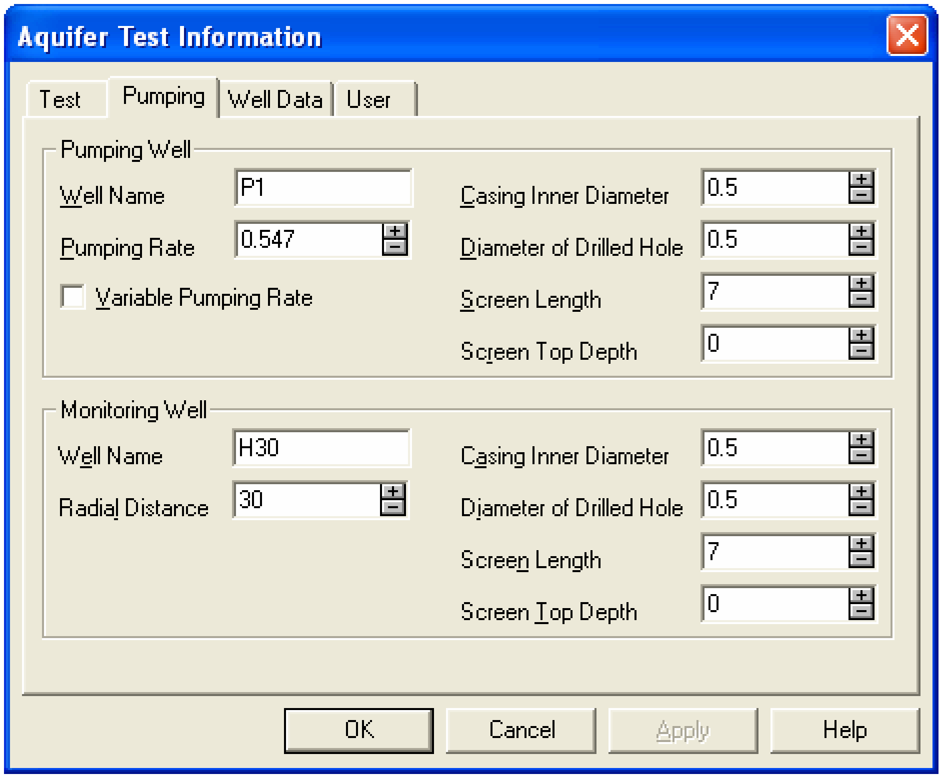
Selecting the **Aquifer Test** menu displays the *Aquifer Test Information* property sheet; this property sheet allows you to enter data defining the test you performed, including a description of the test, the pumping rate history for the test, wells that were monitored during the test, and the time-drawdown data for each well. A subset of the nine potential tabs will be displayed depending on the document type and data. The *Aquifer Test Information* property sheet is described below.



Test Designator:	User field that appears in the summary report and can be used to annotate a graph or map view.

Test Information

Job Number	User field that appears in the summary report and can be used to annotate a graph or map view.
Date	User field that appears in the summary report and can be used to annotate a graph or map view.
Area Name	User field that appears in the summary report and can be used to annotate a graph or map view.
Additional Info	User field that appears in the summary report and can be used to annotate a graph or map view.



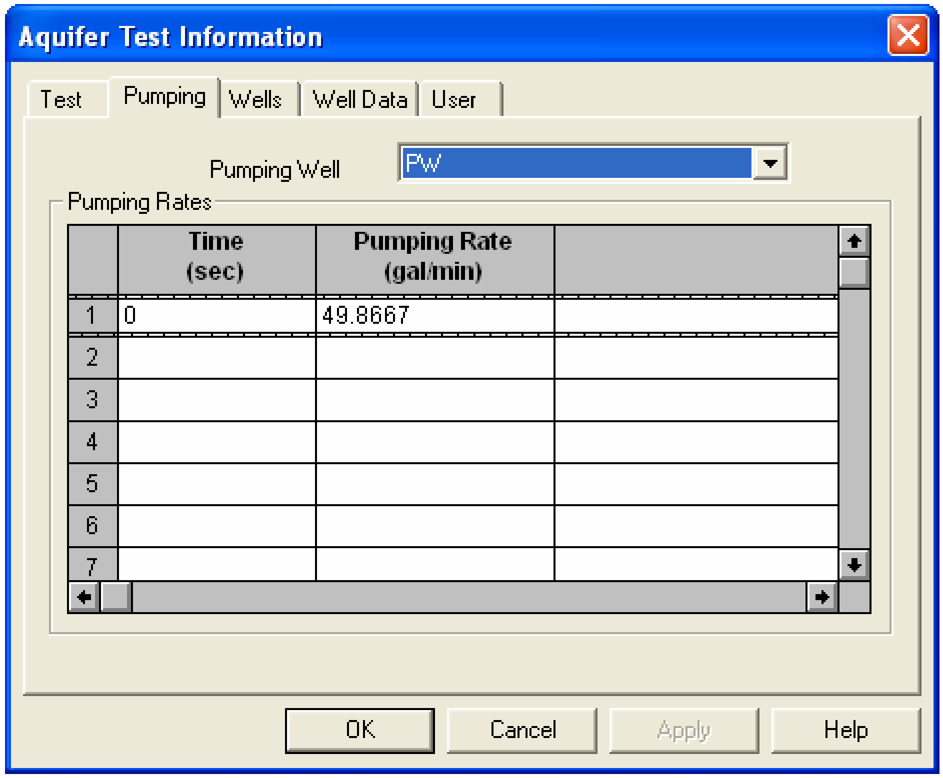
Pumping Well

Well Name	Name of the pumping well.
Pumping Rate	Well discharge rate.
Variable Pumping Rate	Set this check box to activate a variable pumping rate analysis. An additional **Rates** tab will be added to the property sheet for entering the pumping rates.
Casing Inner Diameter	The inner diameter of the pumping well casing.
Diameter of Drilled Hole	The diameter of the pumping well drilled hole.
Screen Length	The length of the pumping well screen.
Screen Top Depth	The depth to top of the pumping well screen relative to the top of the aquifer.

Monitoring Well

Well Name	Name of the monitoring well.
Radial Distance	Distance between the monitoring well and the pumping well.
Casing Inner Diameter	The inner diameter of the monitoring well casing.
Diameter of Drilled Hole	The diameter of the monitoring well drilled hole.
Screen Length	The length of the monitoring well screen.
Screen Top Depth	The depth to top of the monitoring well screen relative to the top of the aquifer.

NOTE: The *Pumping* tab is displayed as shown above when no wells have been defined in the site map.

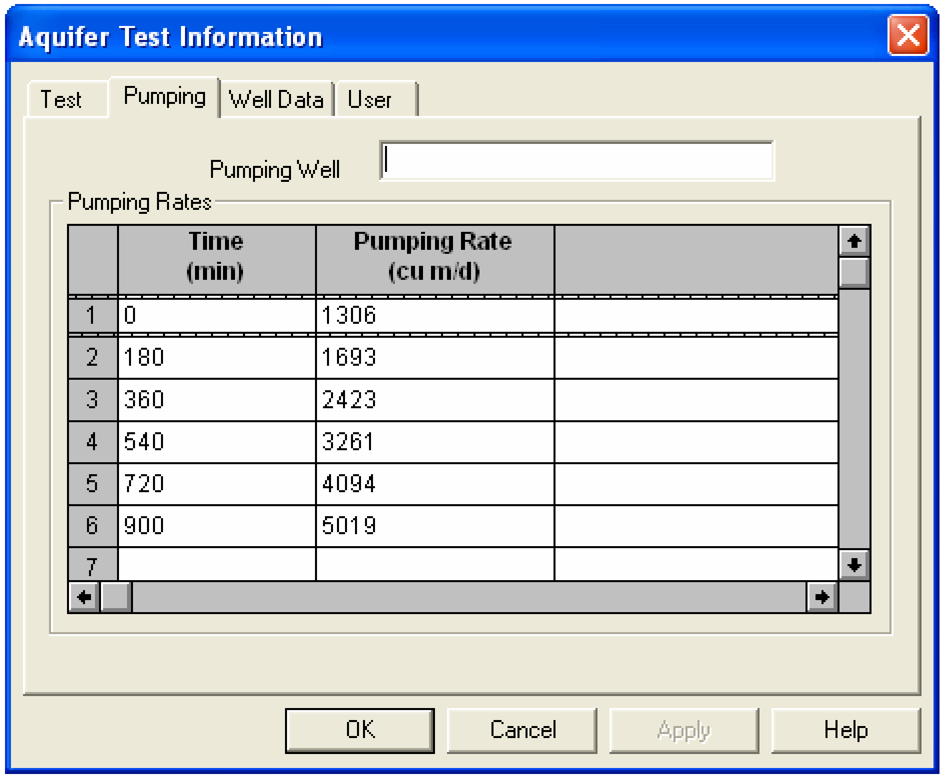


Pumping Well	Select the well pumped during the test

Pumping Rates

Time	The time coordinate of the pumping
Pumping Rate	The discharge rate at the specified time

NOTE: The *Pumping* tab is displayed as shown above when wells have been defined in the site map.



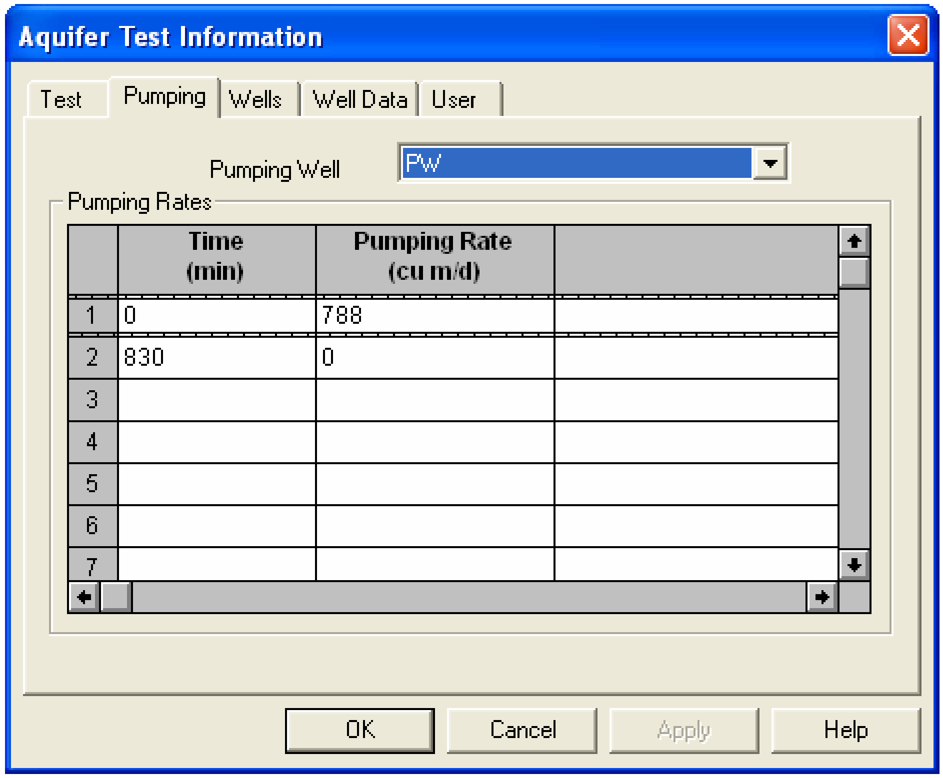
Pumping Well	The name of the well pumped during the test

Pumping Rates

Time	The starting time coordinate of the pumping step

Pumping Rate	The discharge rate during the pumping step

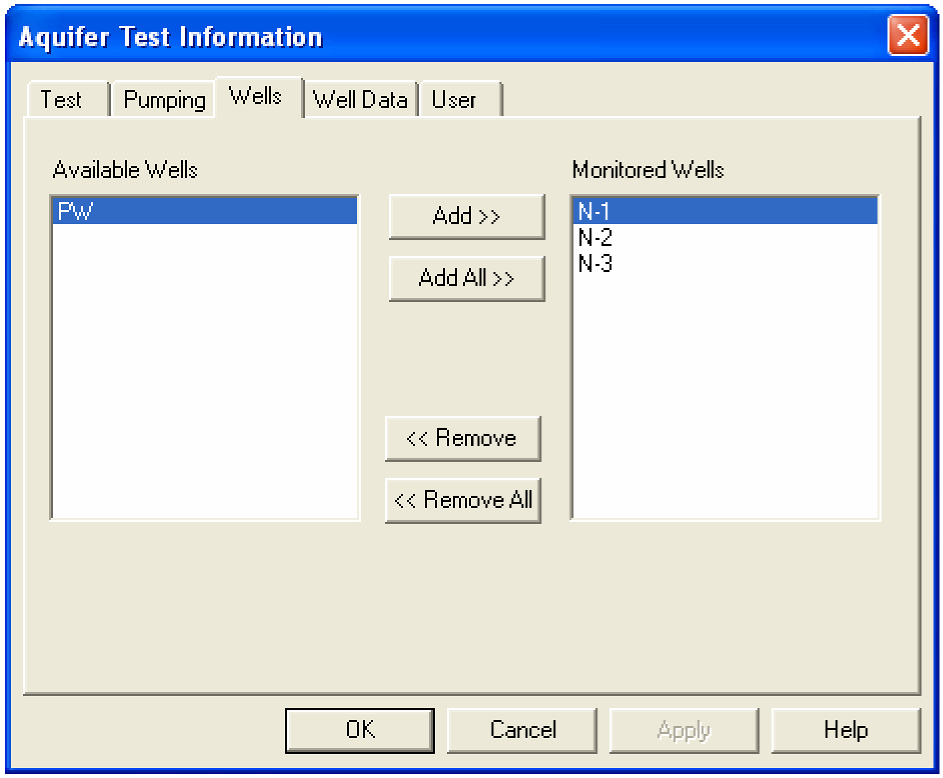
NOTE: The *Pumping* tab is displayed as shown above when the active document is a Step Test document.



Pumping Rates

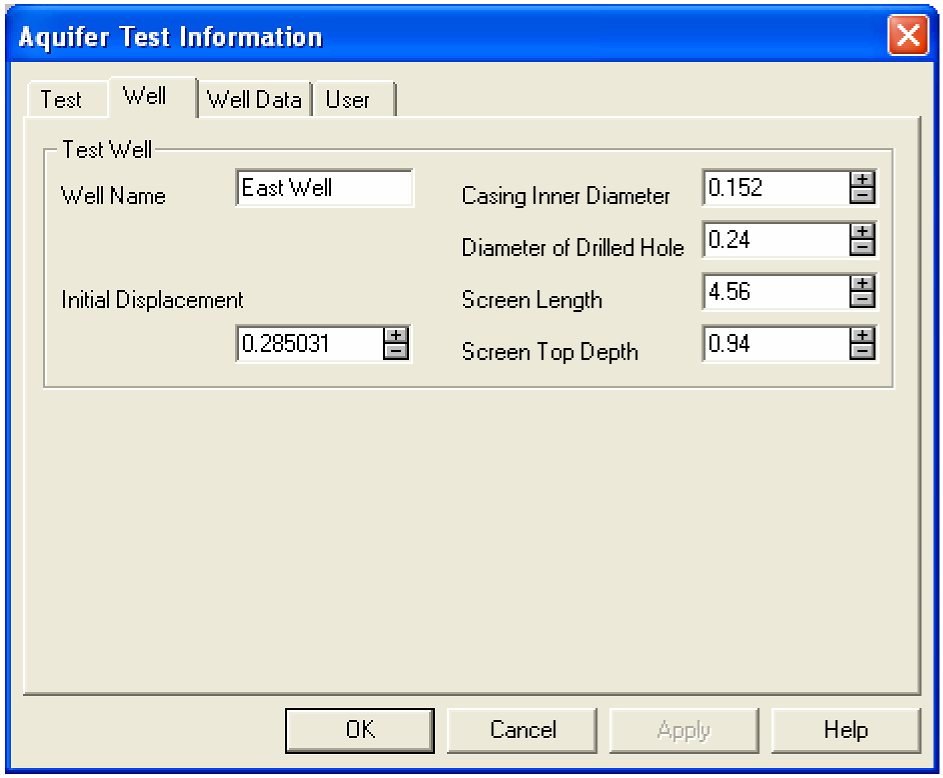
Time	The starting time of the pumping rate
Pumping Rate	The pumping rate at the specified time

NOTE: The *Rates* tab is displayed as shown above when no wells have been defined in the site map and Variable Pumping Rate has been checked on the Pumping tab.



Available Wells	Contains wells that are not being monitored
Monitored Wells	Contains wells that are being monitored
Add>>	Moves the currently selected well from the *Available Wells* list to the *Monitored Wells* list
Add All>>	Moves all wells from the *Available Wells* list to the *Monitored Wells* list
Remove>>	Moves the currently selected well from the *Monitored Wells* list to the *Available Wells* list
Remove All>>	Moves all wells from the *Monitored Wells* list to the *Available Wells* list

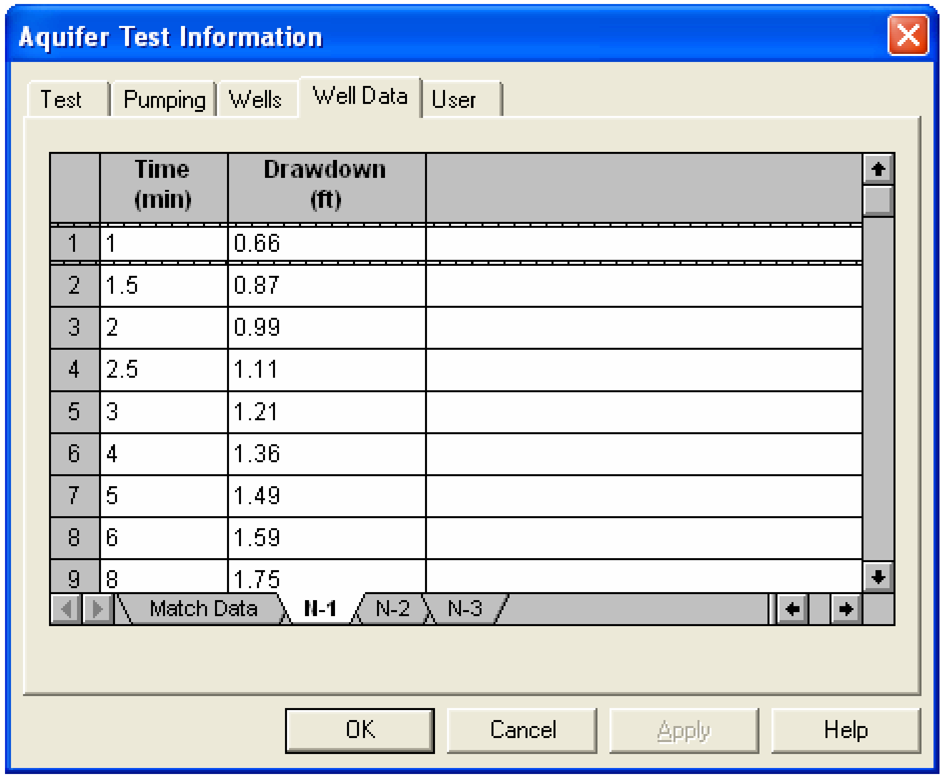
NOTE: The *Wells* tab is only shown when at least one well has been defined in the map view



Test Well

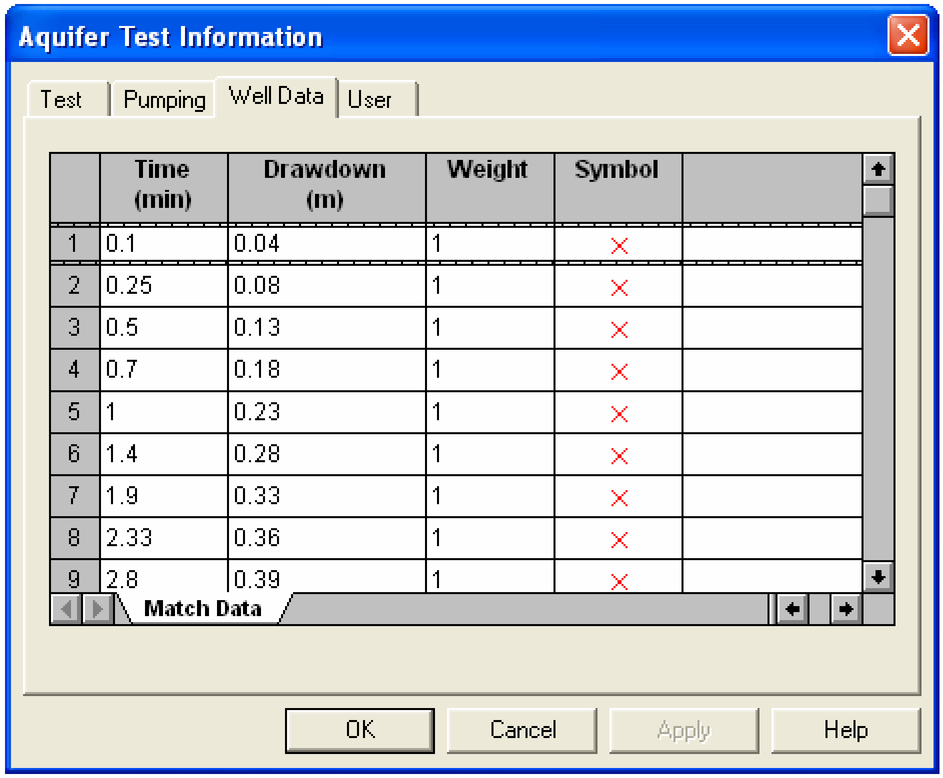
Well Name	Name of the well being tested
Initial Displacement	The maximum change in water level caused by the slug
Casing Inner Diameter	The inner diameter of the pumping well casing.
Diameter of Drilled Hole	The diameter of the pumping well drilled hole.
Screen Length	The length of the pumping well screen.
Screen Top Depth	The depth to top of the pumping well screen relative to the top of the aquifer.

NOTE: This tab is only displayed when a Slug Test document is active.



Time	The time coordinate of the monitoring well response
Drawdown	The drawdown of the monitoring well response

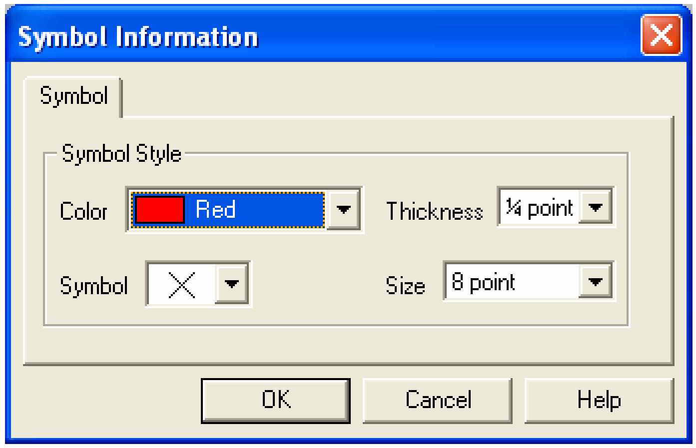
NOTE: The *Well Data* tab appears as shown above when at least one well has been defined as monitored in the *Wells* tab. If no wells are designated as monitored, only the Match Data tab is displayed.



The **Match Data** tab, shown above for a simple test, allows the user to alter the weighting and/or symbol on each data point. Increasing the weighting factor progressively reduces the influence of a data point when using the Marquardt (modified Gauss-Newton) nonlinear least-squares technique. Weighting is relative to the inverse square of the weight value. A value greater than or equal to 100 removes the data point from consideration in a linear regression analysis.

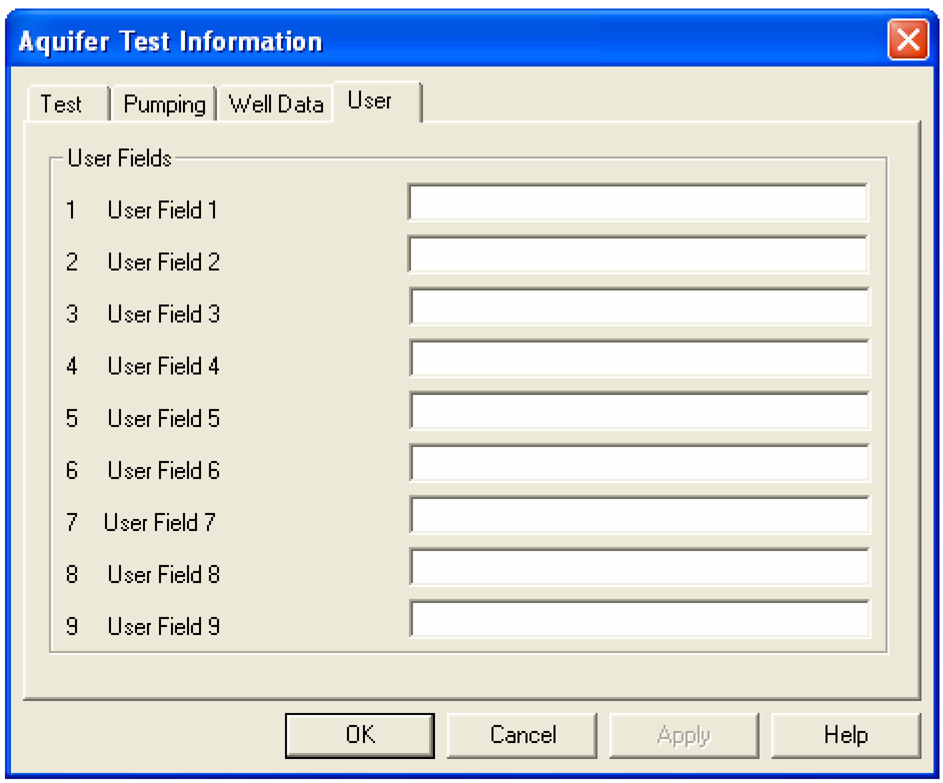
Time	The time coordinate of the monitoring well response
Drawdown	The drawdown of the monitoring well response
Weight	The weight applied to the data point as described above
Symbol	Displays the symbol used to display the data point on graphs

The symbol is edited by double clicking the mouse on the spreadsheet cell or setting focus on the cell and pressing the **Spacebar** key. The *Symbol Information* property sheet shown below is then displayed.



Symbol Style

Color	The color of the symbol displayed at each time/drawdown data point.
Thickness	The line thickness used to draw the symbol displayed at each time/drawdown data point.
Symbol	The symbol displayed at each time/drawdown data point.
Size	The size of the symbol displayed at each time/drawdown data point.



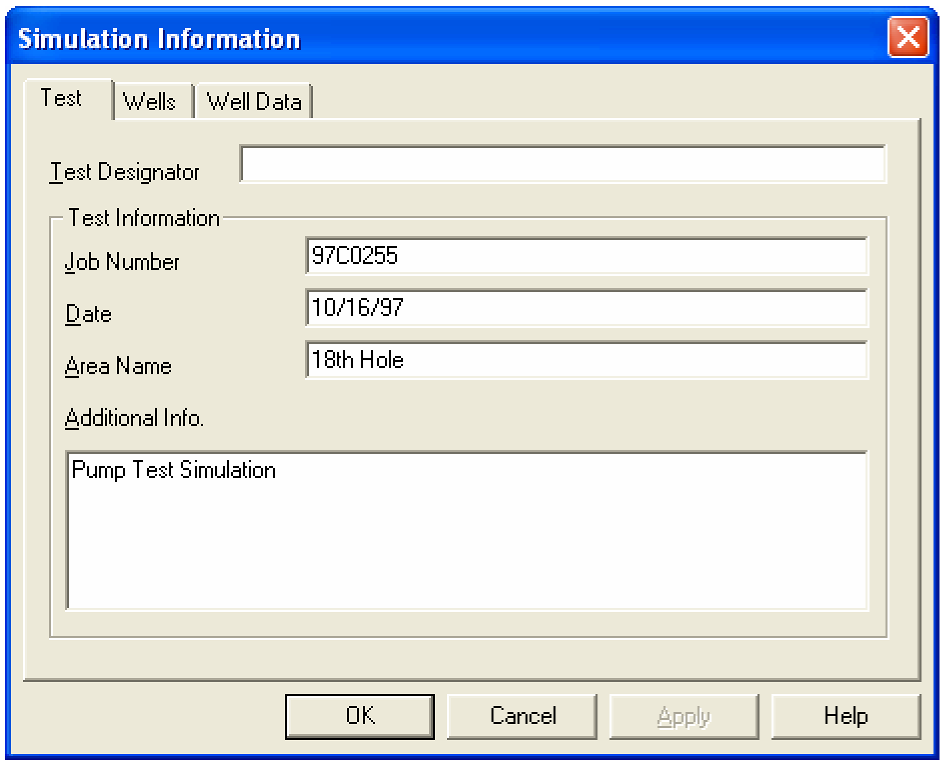
The **User** tab contains customizable parameter boxes which allow the input of a wide range of user specified data or text items. Each field initially has a default name which can be changed. Place the cursor over the field name to be changed and right click the mouse. On the context menu displayed, select **Edit** and enter the new name

required. Click on **OK** and the new name will be saved. The required information can then be entered into the input section of each box.

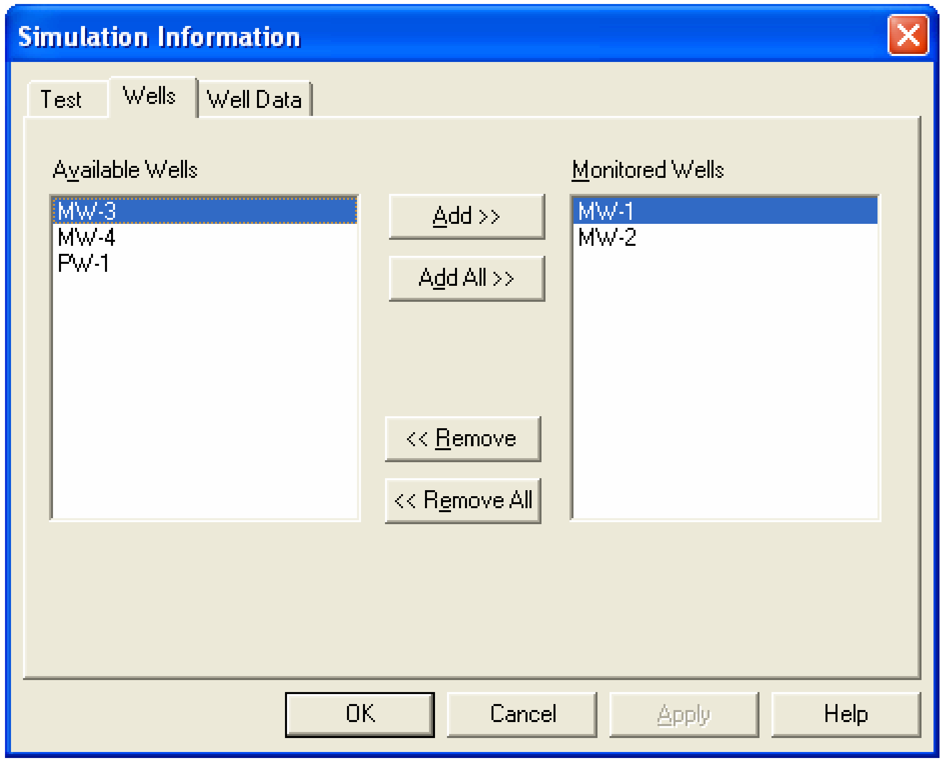
Information entered in these fields can be accessed in footers/headers, figures and parameter boxes displayed on graphical output.

Simulation

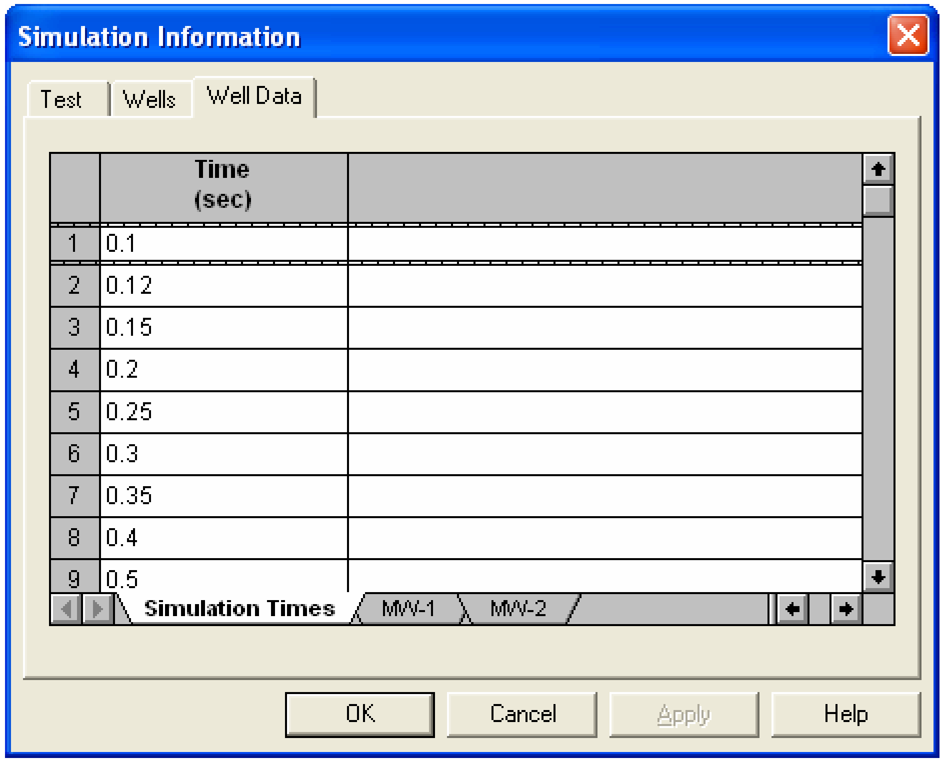
Selecting the **Simulation** menu displays the *Simulation Information* property sheet; this property sheet allows you to define monitoring wells for which you want to calculate time/drawdown data. The *Simulation Information* property sheet is described below.



Test Designator:	User field that appears in the summary report and can be used to annotate a graph or map view.
Test Information	
Job Number	User field that appears in the summary report and can be used to annotate a graph or map view.
Date	User field that appears in the summary report and can be used to annotate a graph or map view.
Area Name	User field that appears in the summary report and can be used to annotate a graph or map view.
Additional Info	User field that appears in the summary report and can be used to annotate a graph or map view.

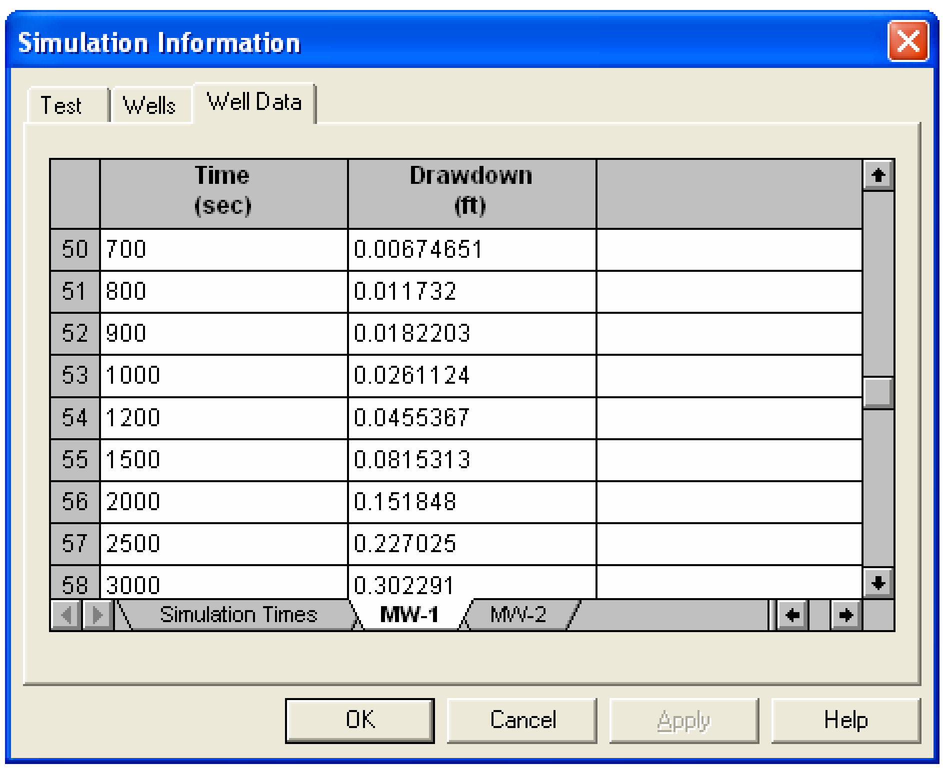


Available Wells	Contains wells that are not being simulated
Monitored Wells	Contains wells that are being simulated
Add>>	Moves the currently selected well from the *Available Wells* list to the *Monitored Wells* list
Add All>>	Moves all wells from the *Available Wells* list to the *Monitored Wells* list
Remove>>	Moves the currently selected well from the *Monitored Wells* list to the *Available Wells* list
Remove All>>	Moves all wells from the *Monitored Wells* list to the *Available Wells* list



Time — The time coordinate at which to calculate well responses

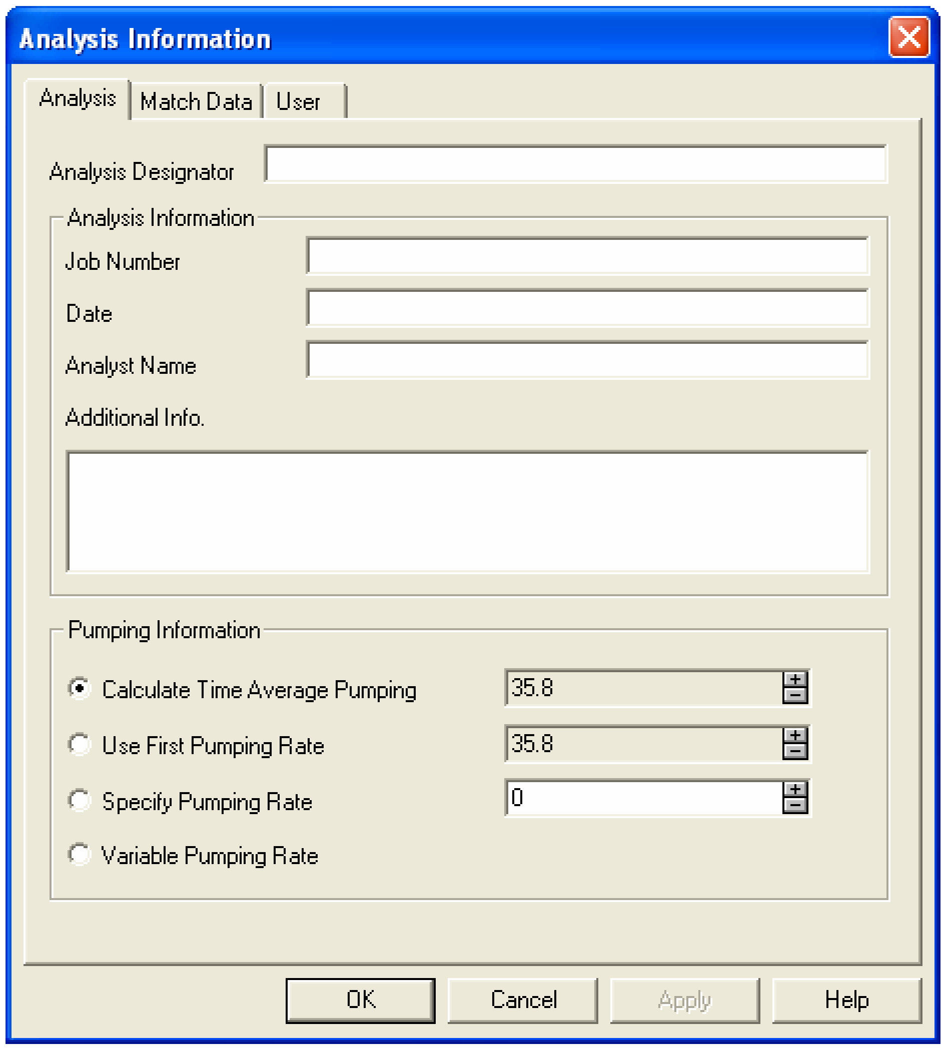
NOTE: The *Simulation Times* tab is used to define the specific times at which to calculate drawdown in all the simulated wells. Changes to the times take effect the next time you calculate the simulation.



Time	The time coordinate of the simulated well response
Drawdown	The drawdown of the simulated well response

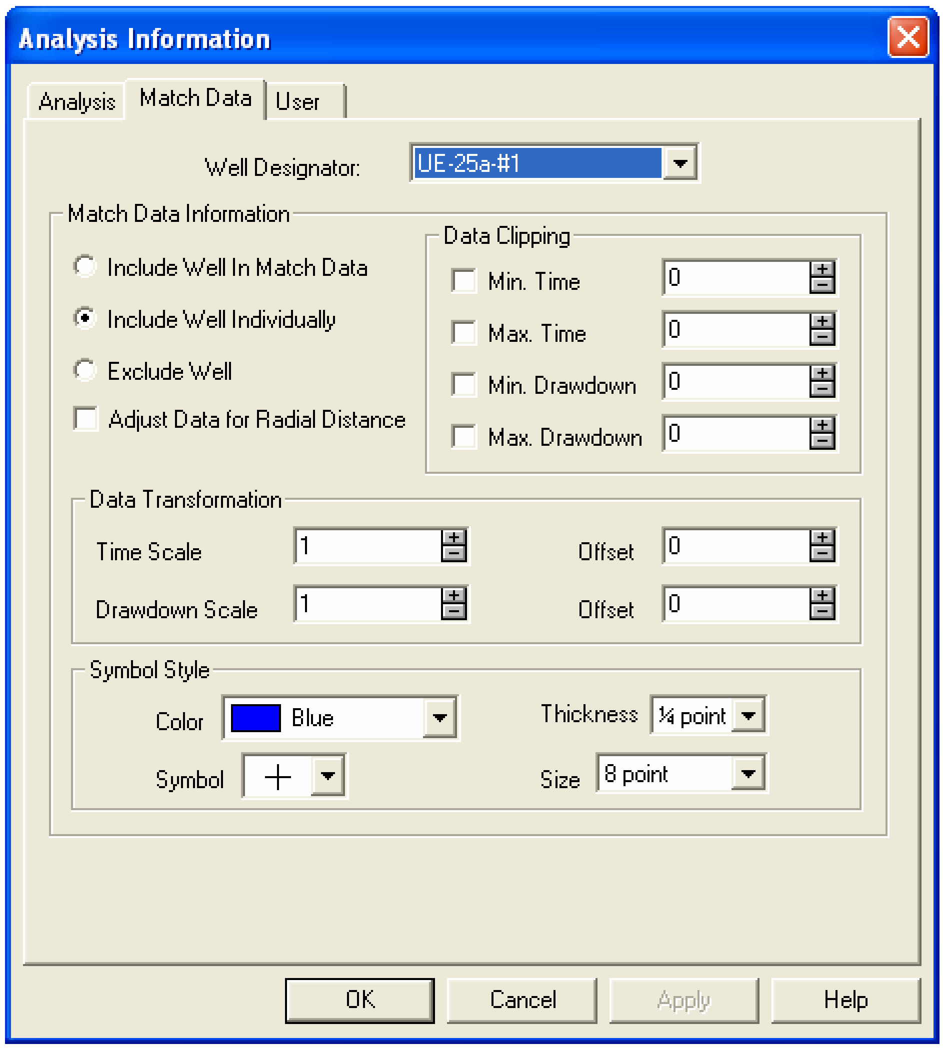
Analysis

Selecting the **Analysis** menu displays the *Analysis Information* property sheet that allows you to document the current analysis and to determine how data from monitoring wells is to be analyzed. The parameters you may modify are described below.



Analysis Designator	User field that appears in the summary report and can be used to annotate a graph or map view.
Analysis Information	
Job Number	User field that appears in the summary report and can be used to annotate a graph or map view.
Date	User field that appears in the summary report and can be used to annotate a graph or map view.
Analyst Name	User field that appears in the summary report and can be used to annotate a graph or map view.

Additional Info	User field that appears in the summary report and can be used to annotate a graph or map view.
Pumping Information	
Calculate Time Average Pumping	Pumping rate used in the analysis is calculated as a time average of the pumping schedule.
Use First Pumping Rate	The first pumping rate in the pumping schedule is used for the analysis.
Specify Pumping Rate	The pumping rate used in the analysis is specified in the adjacent field.
Variable Pumping Rate	The variable pumping rate defined for the pumping schedule is used



Well Designator:	Specifies the monitoring well being edited.
Match Data Information	
Include Well In Match Data	The drawdown data is added to the Match Data set

Include Well Individually	The well is included in the analysis independently
Exclude Well	The well is not included in the analysis.
Adjust Data for Radial Distance	The time coordinate is divided by the radial distance squared

Data Clipping

Min. Time	If checked, data points whose transformed time value is below the value in the adjacent field are excluded from the analysis.
Max. Time	If checked, data points whose transformed time value is above the value in the adjacent field are excluded from the analysis.
Min. Drawdown	If checked, data points whose transformed drawdown value is below the value in the adjacent field are excluded from the analysis.
Max. Drawdown	If checked, data points whose transformed drawdown value is above the value in the adjacent field are excluded from the analysis.

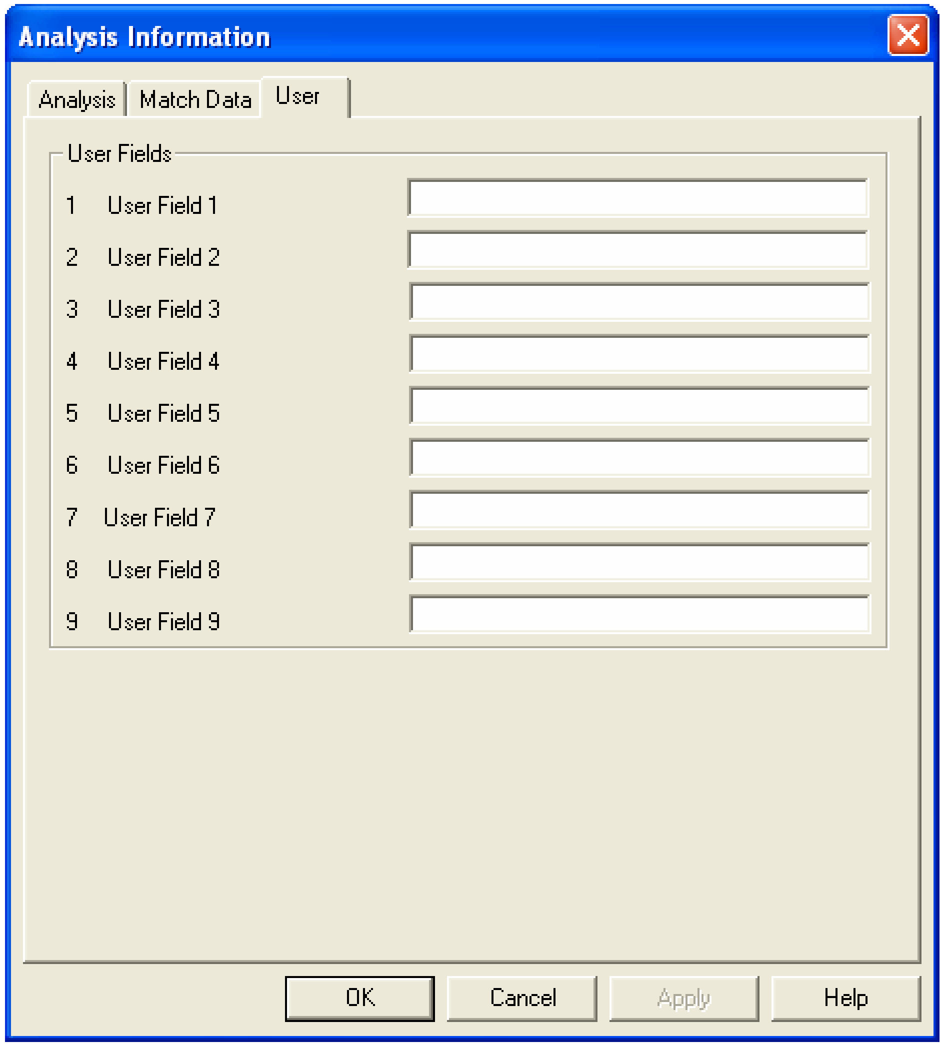
Data Transformation

Time Scale/Offset	Time values are multiplied by the scale value and added to the offset value.
Drawdown Scale/Offset	Drawdown values are multiplied by the scale value and added to the offset value.

Symbol Style

Color	The color of the symbol displayed at each time/drawdown data point.
Thickness	The line thickness used to draw the symbol displayed at each time/drawdown data point.
Symbol	The symbol displayed at each time/drawdown data point.
Size	The size of the symbol displayed at each time/drawdown data point.

NOTE: The *Match Data* tab appears as shown above when at least one well has been defined as monitored in the *Wells* tab.



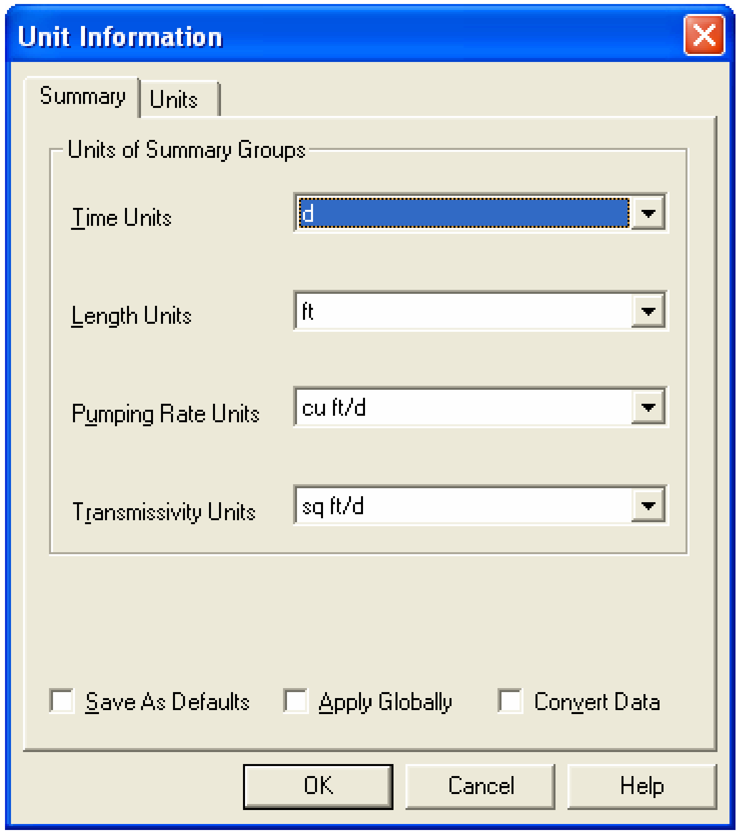
The **User** tab contains customizable parameter boxes which allow the input of a wide range of user specified data or text items. Each field initially has a default name which can be changed. Place the cursor over the field name to be changed and right click the mouse. On the context menu displayed, select **Edit** and enter the new name required. Click on **OK** and the new name will be saved. The required information can then be entered into the input section of each box.

Information entered in these fields can be accessed in footers/headers, figures and parameter boxes displayed on graphical output.

Units

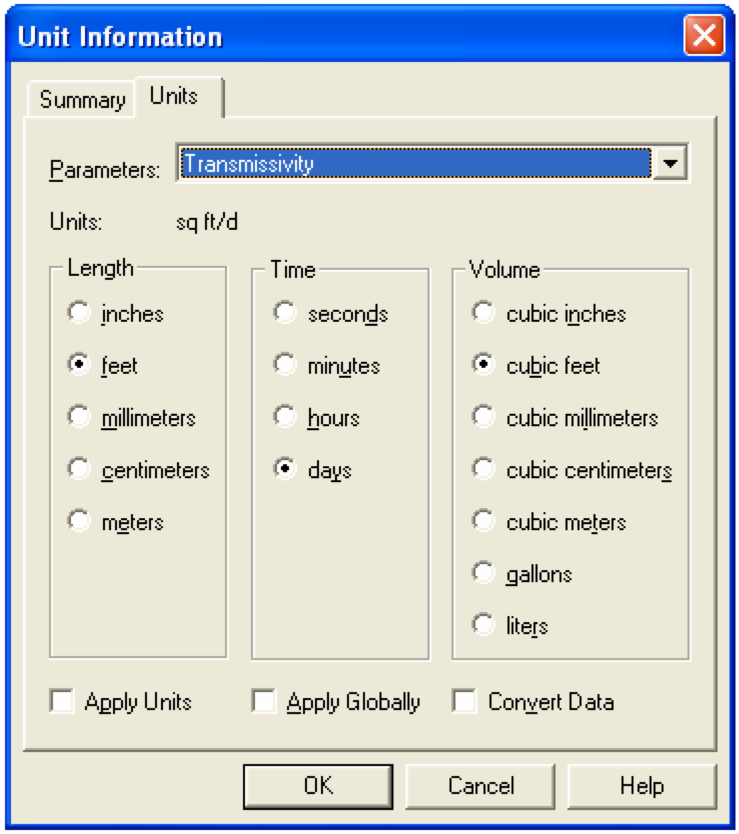
Selecting the **Units** menu displays the *Unit Information* property sheet with two tabs, as shown below. The first tab is labeled ***Summary*** and contains the four basic parameters and their respective units. The values represent either the program defaults or the values previously saved as default. Under normal circumstances, you would set the summary units once and save them as your defaults. If you always use the same types of units in your analyses, then you never need to change these again. Click the **Save As Default** check-box to do this and then click **OK**.

NOTE: If changes are made to both tabs of this property sheet, the values on the Summary tab take priority.



Units of Summary Groups

Time Units	Specifies the units to use for time
Length Units	Specifies the units to use for length
Pumping Rate Units	Specifies the units to use for pumping rate
Transmissivity Units	Specifies the units to use for transmissivity
Save As Defaults	The current selections are saved to the system registry and are used as the defaults
Apply Globally	The current selections are applied globally throughout the current document
Convert Data	The existing data values are converted from the old units to the new units

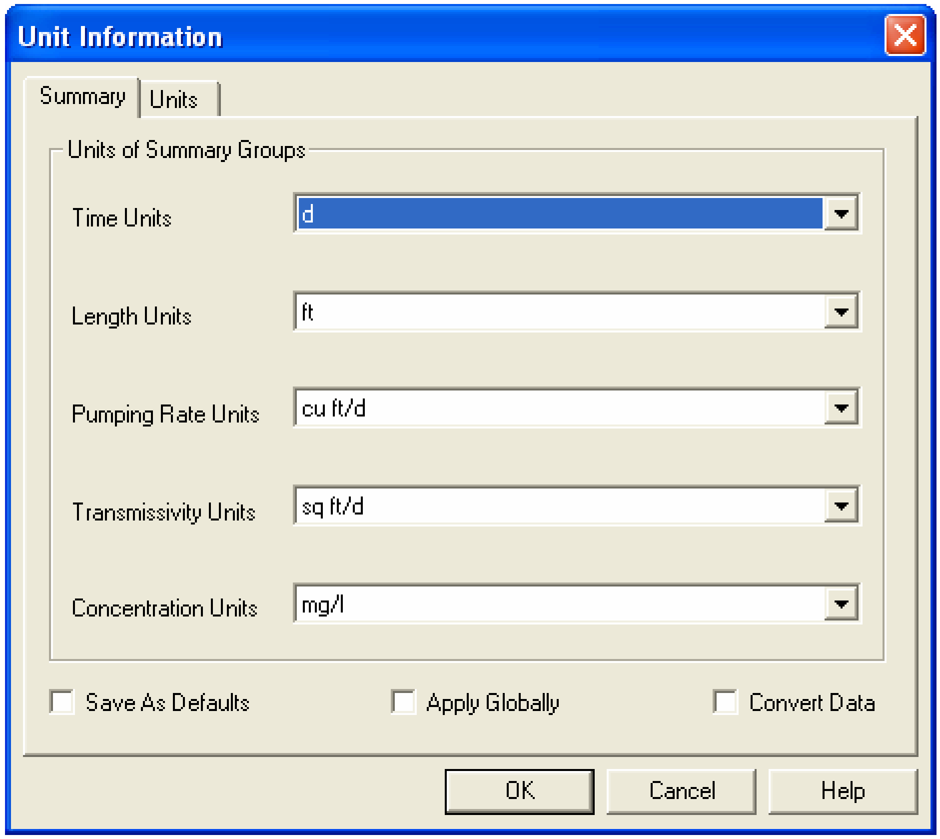


Parameters:	The parameter for which units are to be changed.
Units:	Units for the selected parameter as they will be displayed within the program
Length	Units of length
Time	Units of time
Volume	Units of volume
Apply Units	The current selections are applied to the currently selected parameter for the currently selected well
Apply Globally	The current selections are applied to the currently selected parameter globally throughout the active document
Convert Data	The existing data values are converted from the old units to the new units

Units - Mass

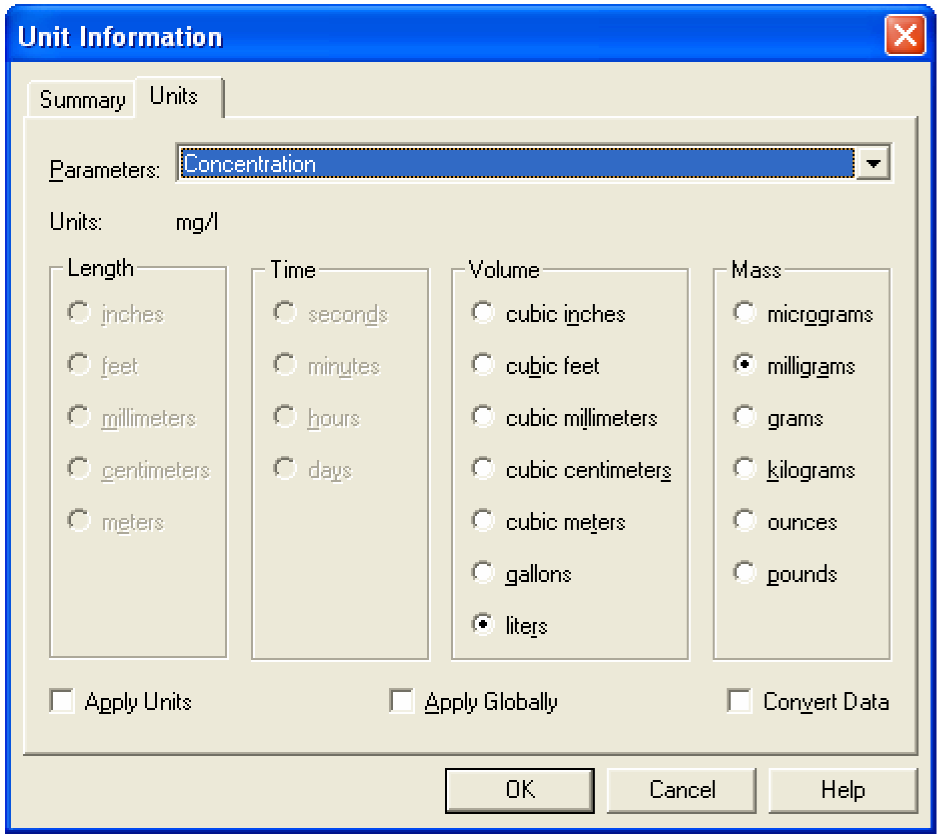
Selecting the **Units** menu displays the *Unit Information* property sheet with two tabs, as shown below. The first tab is labeled ***Summary*** and contains the four basic parameters and their respective units. The values represent either the program defaults or the values previously saved as default. Under normal circumstances, you would set the summary units once and save them as your defaults. If you always use the same types of units in your analyses, then you never need to change these again. Click the **Save As Default** check-box to do this and then click **OK**.

NOTE: If changes are made to both tabs of this property sheet, the values on the Summary tab take priority.



Units of Summary Groups

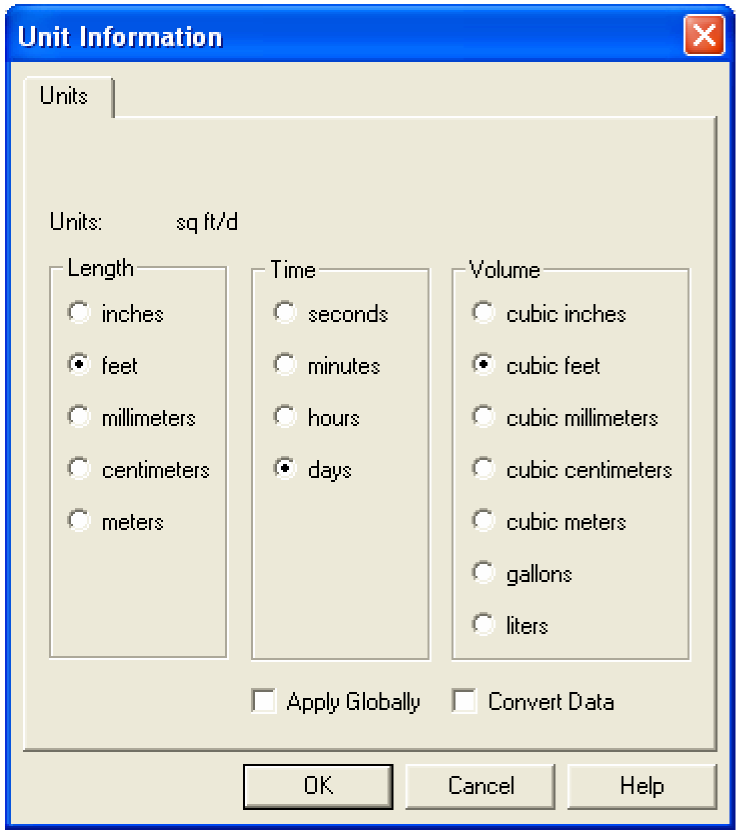
Time Units	Specifies the units to use for time
Length Units	Specifies the units to use for length
Pumping Rate Units	Specifies the units to use for pumping rate
Transmissivity Units	Specifies the units to use for transmissivity
Concentration Units	Specifies the units to use for concentration
Save As Defaults	The current selections are saved to the system registry and are used as the defaults
Apply Globally	The current selections are applied globally throughout the current document
Convert Data	The existing data values are converted from the old units to the new units



Parameters:	The parameter for which units are to be changed.
Units:	Units for the selected parameter as they will be displayed within the program
Length	Units of length
Time	Units of time
Volume	Units of volume
Mass	Units of mass
Apply Units	The current selections are applied to the currently selected parameter for the currently selected well
Apply Globally	The current selections are applied to the currently selected parameter globally throughout the active document
Convert Data	The existing data values are converted from the old units to the new units

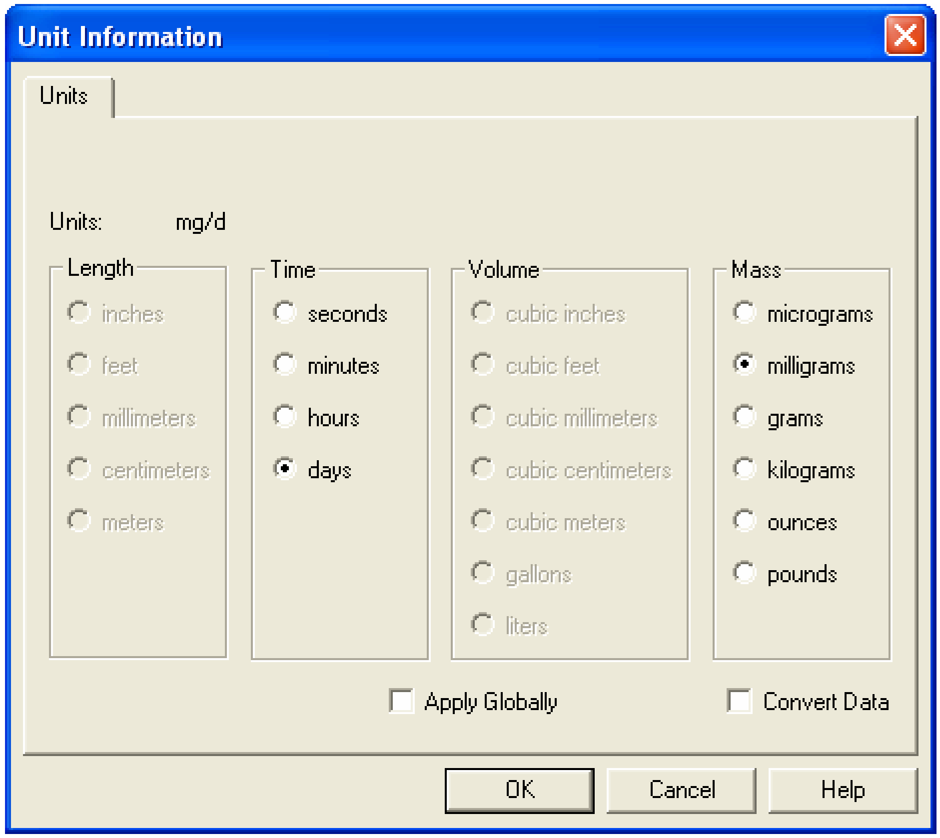
Units - Parameter

Selecting the **Units** menu from the edit field context menu displays the *Unit Information* property sheet for the specific parameter. If the parameter has no mass component, the property is as shown below.



Units:	Units for the selected parameter as they will be displayed within the program
Length	Units of length
Time	Units of time
Volume	Units of volume
Apply Globally	The current selections are applied to the currently selected parameter globally throughout the active document
Convert Data	The existing data values are converted from the old units to the new units

If the parameter has a mass component, the property is as shown below.



Units:	Units for the selected parameter as they will be displayed within the program
Length	Units of length
Time	Units of time
Volume	Units of volume
Mass	Units of mass
Apply Globally	The current selections are applied to the currently selected parameter globally throughout the active document
Convert Data	The existing data values are converted from the old units to the new units

3D Manipulation

Selecting the **3D Manipulation** menu from the edit field context menu allows the manipulation of the 3D view using the mouse and keyboard. Clicking and/or holding the left mouse button in the view causes the display to rotate in the direction toward the cursor location. Holding the **Shift** button while clicking and/or holding the left mouse button causes the display to move in the direction toward the cursor location.

Clicking and/or holding the right mouse button causes the display to zoom in and zoom out. If the cursor is in the upper half of the view, it will zoom out. If the cursor is in the lower half of the view, it will zoom in. The amount of the zoom is controlled by how far the cursor is from the vertical center of the view.

Holding the **Ctrl** button while clicking and/or holding the left mouse button causes the display to rotate around its center point. If the cursor is in the upper half of the view, it will rotate counter clockwise. If the cursor is in the lower half of the view it will rotate clockwise. The amount/speed of the rotation is controlled by how far the cursor is from the vertical center of the view.

Insert

The **Insert** menu is only available in the spreadsheet view. Selecting **Insert** from the **Edit** menu inserts a new data point before the currently selected row in the spreadsheet. Aquifer^{Win32} automatically assigns the time and drawdown values of 1.0 so you will need to change these values before continuing with the analysis.

Append

The **Append** menu adds a new row of data to the end of the spreadsheet. As with **Insert** above, time and drawdown are initially assigned values of 1.0.

Delete

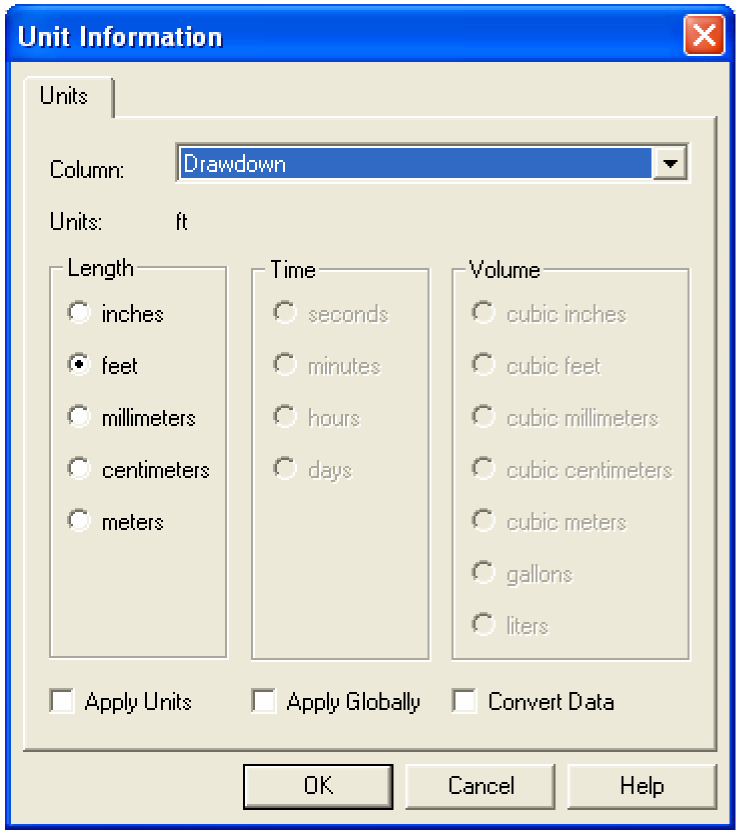
The **Delete** menu deletes the currently highlighted row of the spreadsheet or the currently selected block of rows. For a graph or map view, it deletes all selected elements.

Sort

The **Sort** menu sorts the spreadsheet relative to the Time value.

Column Conversion

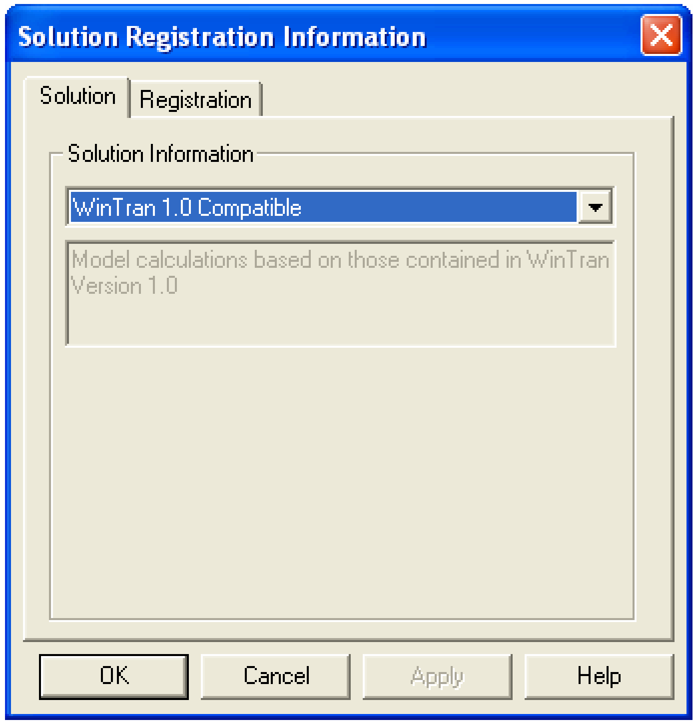
The **Column Conversion** menu displays the *Unit Information* property sheet that allows you to change the units on any column in the spreadsheet. Refer to the section entitled **Units** above for more information on this option.



Column:	The column for which units are to be changed
Units:	Units for the selected parameter as they will be displayed within the program
Length	Units of length
Time	Units of time
Volume	Units of volume
Apply Units	The current selections are applied to the currently selected parameter for the currently selected well
Apply Globally	The current selections are applied to the currently selected parameter globally throughout the current document
Convert Data	The existing data values are converted from the old units to the new units

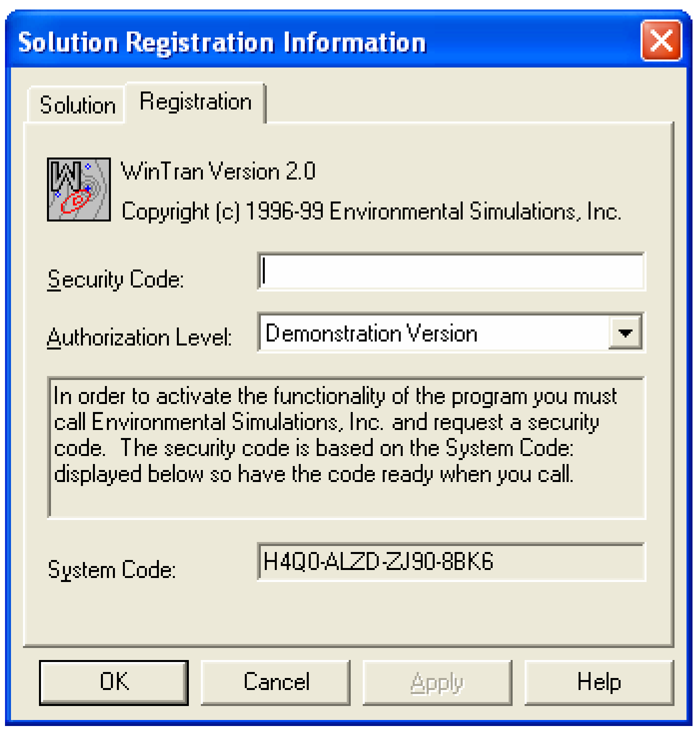
Solution Registration

The **Solution Registration** menu is only displayed for **Flow Model** documents and serves to identify each model available to the application. In the future, some models may require registration; if registration is required, the ***Registration*** tab will be displayed on the *Solution Registration Information* property sheet. Although WinTran is currently included without separate registration, it serves as an example of the screens.



Solution Information

Analysis: Defines the type of analytic solution to use when simulating a pump test

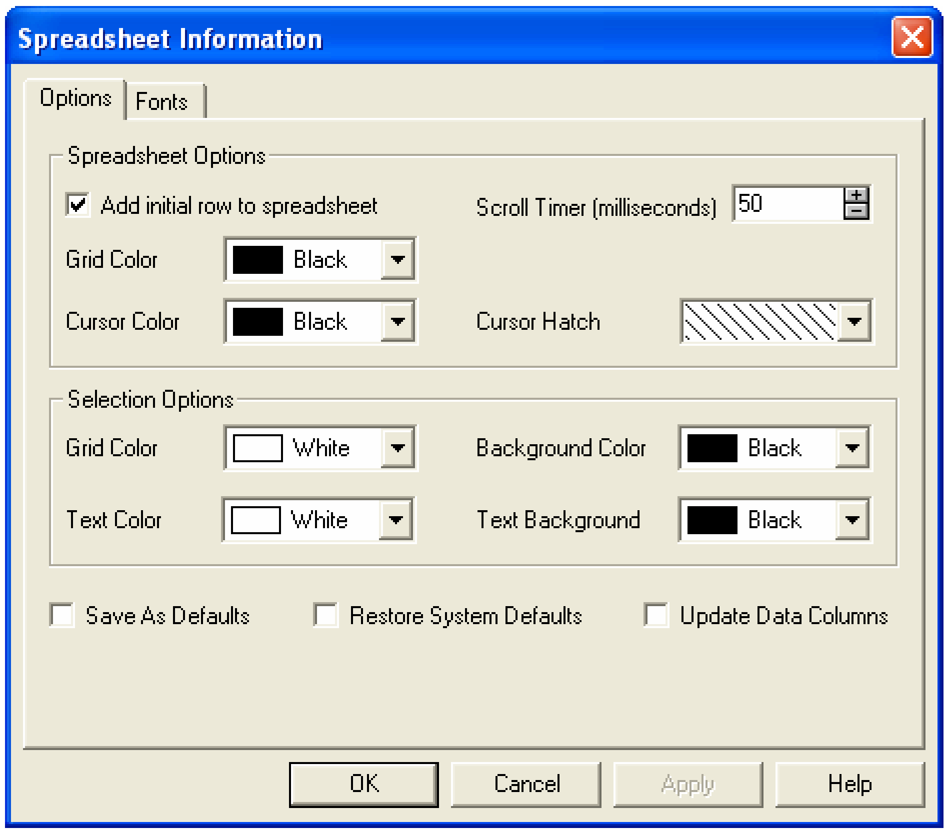


Security Code: This is an encrypted code containing information about your software license and the configuration of the computer used to validate the license

Authorization Level:	The level of functionality of the application granted by the license
System Code:	This is an encrypted code containing information about your hardware configuration used to generate a unique Security Code

Options

The **Options** menu is only displayed for the spreadsheet view and controls the default display characteristics of the spreadsheet and the columns of data contained therein. The specific options are on the *Spreadsheet Information* property sheet.are summarized below.



Spreadsheet Options

Add initial row to spreadsheet	When checked, a default initial data row is added to the spreadsheet when a new document is created. This row is initially active to help new users enter data for the first time.
Scroll Timer (milliseconds)	Sets the number of milliseconds to wait between automatic scrolling events in the spreadsheet
Grid Color	Sets the color to use for the grid of the spreadsheet
Cursor Color	Sets the color to use when displaying the cursor indicating the currently active line
Cursor Hatch	Sets the hatch pattern to use when displaying the cursor indicating the currently active line

Selection Options

Grid Color — Sets the grid color to use when a number of data lines have been selected

Background Color — Sets the background color to use when a number of data lines have been selected

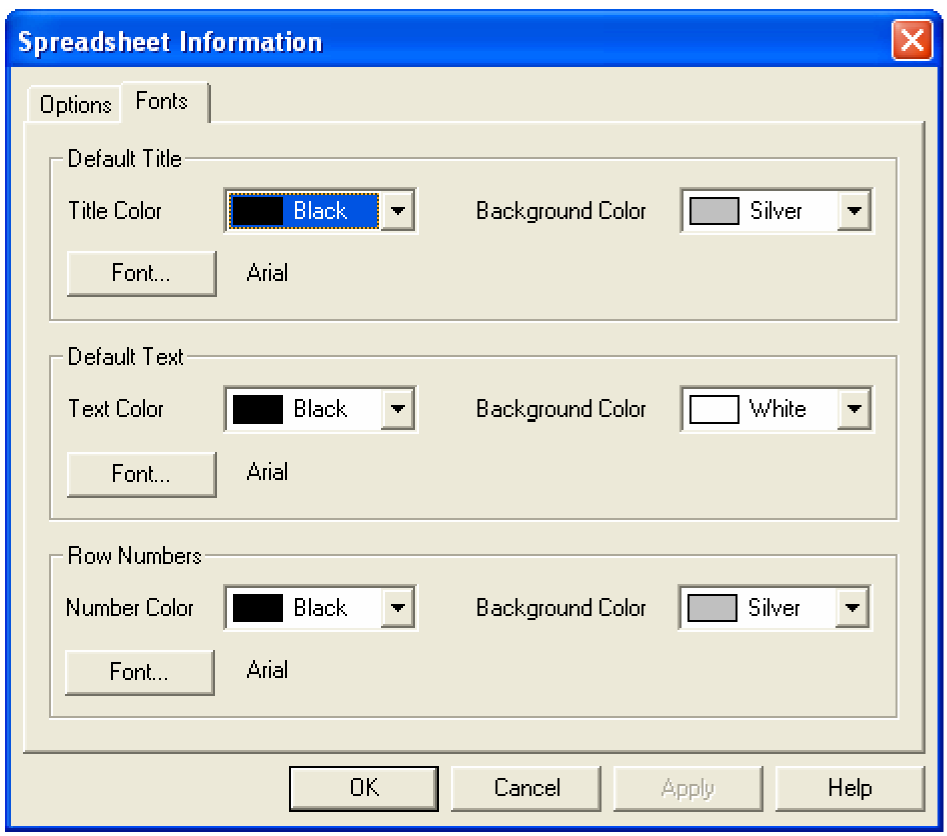
Text Color — Sets the text color to use when a number of data lines have been selected

Text Background — Sets the text background color to use when a number of data lines have been selected

Save As Defaults — When checked, the values contained in this property sheet are stored in the system registry when the property sheet is accepted and are used as the defaults when an new document is created

Restore System Defaults — When checked, the default system values are restored when the property sheet is accepted; this operation occurs before the **Save As Defaults** operation so having both checked resets both the document and the system registry

Update Data Columns — When checked, the default column characteristics defined on the ***Fonts*** tab are propagated to the existing columns in the spreadsheet



Default Title

Title Color — Sets the color to use when displaying the column title

Background Color	Sets the background color to use when displaying the column title
Font	Sets the font to use when displaying the column title
Default Text	
Text Color	Sets the color to use when displaying the column data values
Background Color	Sets the background color to use when displaying the column data values
Font	Sets the font to use when displaying the column data values
Row Numbers	
Number Color	Sets the color to use when displaying the spreadsheet row number
Background Color	Sets the background color to use when displaying the spreadsheet row number
Font	Sets the font to use when displaying the spreadsheet row number

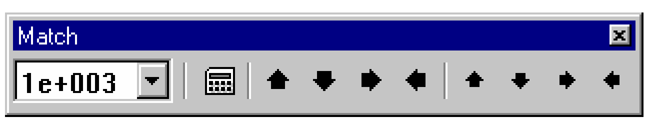
View Menu

Standard Tools

Selecting the **Standard Tools** menu toggles the display of the Standard toolbar. The display position of the toolbar is dependent on where it was previously located.

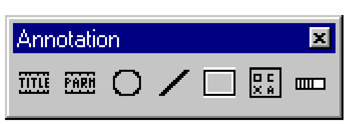
Match Tools

Selecting the **Match Tools** menu toggles the display of the Match toolbar. The display position of the toolbar is dependent on where it was previously located.



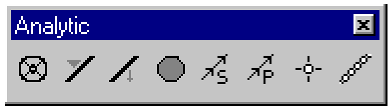
Annotation Tools

Selecting the **Annotation Tools** menu toggles the display of the Annotation toolbar. The display position of the toolbar is dependent on where it was previously located.



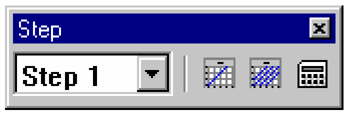
Analytic Tools

Selecting the **Analytic Tools** menu toggles the display of the Analytic toolbar. The display position of the toolbar is dependent on where it was previously located.



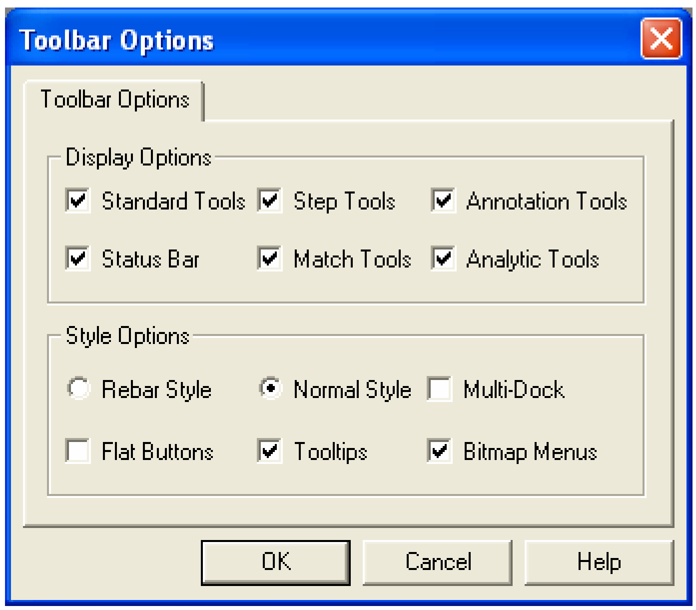
Step Tools

Selecting the **Step Tools** menu toggles the display of the Step toolbar. The display position of the toolbar is dependent on where it was previously located.



Toolbar Options

Selecting the **Toolbar Options** menu activates the *Toolbar Options* property sheet described below.



Display Options

Standard Tools	When checked, the Standard Tools toolbar is displayed
Step Tools	When checked, the Step Tools toolbar is displayed
Annotation Tools	When checked, the Annotation Tools toolbar is displayed
Status Bar	When checked, the Status Bar is displayed
Match Tools	When checked, the Match Tools toolbar is displayed
Analytic Tools	When checked, the Analytic Tools toolbar is displayed

Style Options

Display Parameters

Rebar Style Specifies Rebar Style toolbars which are aligned along the top adjacent to one another and wrap to additional lines as the window size changes

Normal Style Specifies individual, dockable toolbars

Multi-Dock When checked, multiple toolbars can be combined into on floating toolbar frame; Not available for Rebar style or with flat buttons

Flat Buttons Specifies using the new flat looking toolbar buttons

Tooltips When checked, tooltips are active on toolbar buttons

Bitmap Menus When checked, the corresponding toolbar button bitmaps are displayed adjacent to menu items

Status Bar

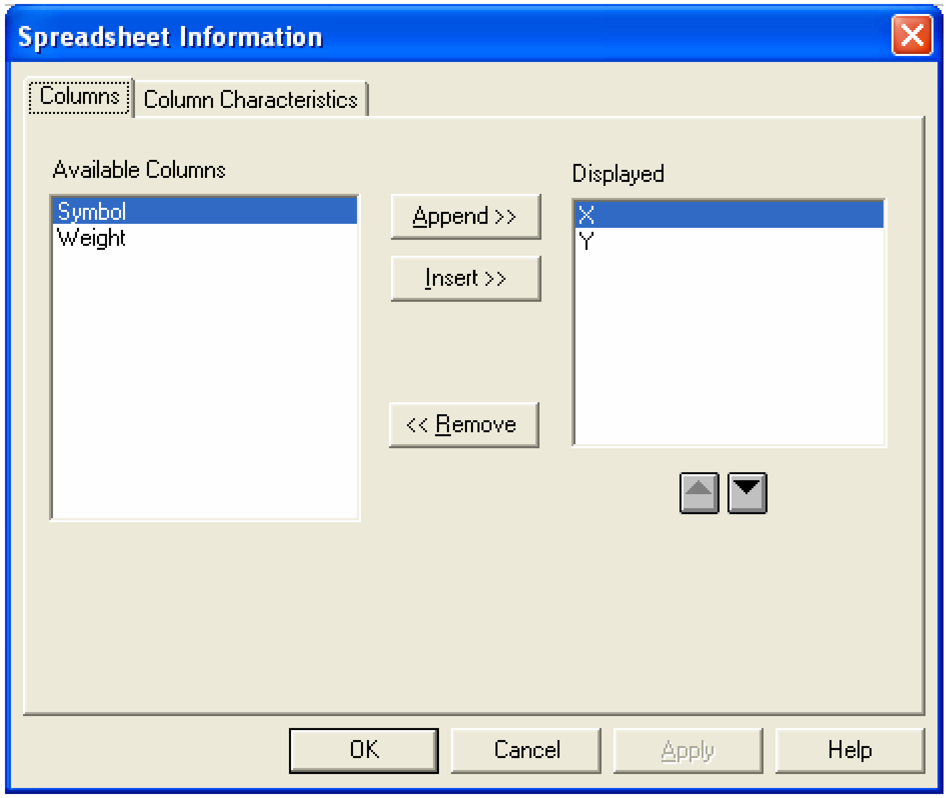
Selecting the **Status Bar** menu toggles the display of the status bar.

Optimize

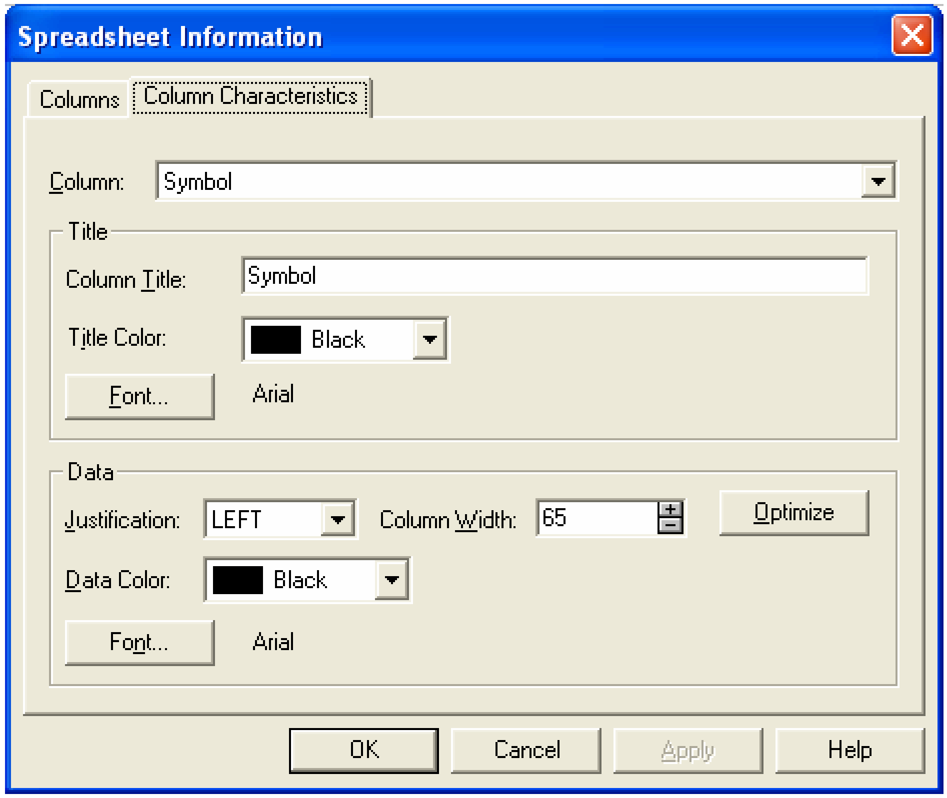
The **Optimize** menu is only displayed when the spreadsheet view is active. This menu causes the width of the spreadsheet columns to be optimized for best viewing. The width of each column is adjusted to be just wide enough to fully display the title and all data values in the column.

Columns

The **Columns** menu is only displayed when the spreadsheet view is active. This selection displays the *Spreadsheet Information* property sheet where you can determine which columns of the spreadsheet are displayed. Additionally, the properties of each column can be modified via the ***Column Characteristics*** tab.



Available Columns	Contains columns that are available to be displayed
Displayed	Contains columns that will be displayed
Add >>	Moves the currently selected column from the *Available Columns* list to the bottom of the *Displayed* list
Insert >>	Moves the currently selected column from the *Available Columns* list to the *Displayed* list located above the selected column in the *Displayed* list
<< **Remove**	Moves the currently selected column from the *Displayed* list to the *Available Columns* list
	Exchanges the currently selected column in the *Displayed* list with the one above it
	Exchanges the currently selected column in the *Displayed* list with the one below it



Column	Select the current column to be edited
Title	
Column Title	Sets the column title to be displayed on the spreadsheet
Title Color	Sets the color use when displaying the column title
Font	Defines the font, font style, size, effects and color for the column title
Data	
Justification	Sets justification of the data values within the column
Column Width	Sets the width in pixels of the column
Optimize	Automatically sets the width of the column to be just wide enough to display the longest data value and the column title
Data Color	Sets the color use when displaying the column data values
Font	Defines the font, font style, size, effects and color for the data values in the column

Well Data

The **Well Data** menu is only displayed when either the spreadsheet view or the Test Simulator view is active. This menu toggles whether the spreadsheet contains separate tabs for data from each of the monitored wells. In a **Simulation** document

the menu toggles whether separate view tabs are available displaying simulated drawdown versus time graphs for all wells simulated.

When you click on the tab containing the data for a particular well instead of the ***Match Data*** tab, Aquifer^{Win32} displays drawdown versus time data on the graph view. When the ***Match Data*** tab is clicked, however, the graph view changes to contain the data plotted over the type curve(s).

Clip Data

The **Clip Data** menu is only displayed when a graph view is active. This menu toggles whether data beyond the edges of the graph are clipped (not displayed). In the **Match Data** graph view, the data symbols are clipped. In the predicted graph, the observed data points and predicted drawdown versus time curves are clipped.

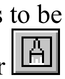
Automatic Refresh

NOTE: This menu is only shown for map views.

The **Automatic Refresh** menu selection determines whether Aquifer^{Win32} will redraw the window each time a dialog or property sheet is displayed or the window is resized, zoomed, or changed in any way. A checkmark appearing next to **Auto Refresh** means that the display is redrawn after every action that changes the display. Click on this menu to toggle the option on or off (with or without the checkmark). The default condition is **Auto Refresh** on. You may want to turn **Auto Refresh** off if you are making a lot of changes and your map is large. On a fast computer (e.g., Pentium), the refresh is fast enough that you probably will not care.

Refresh

NOTE: This menu is only shown for map views.

Selecting **Refresh** from the **View** menu causes the map window to be redrawn. You would use this feature if you have turned the **Auto Refresh** off (see last section) and the screen needs to be redrawn after a scroll or other action. You may also use the Standard toolbar  button.

Scroll and Page

NOTE: This menu is only shown in map views.

Selecting **Scroll** or **Page** from the **View** menu displays a third menu with the following selections: **Left**, **Right**, **Up**, and **Down**. Select which direction the current model window will be scrolled. The only difference between **Scroll** and **Page** is the distance that is scrolled. **Scroll** moves the display a short distance and **Page** moves the display a larger distance.

Zoom

NOTE: This menu is only shown in map views.

The **Zoom** menu allows you to magnify a point on the current display. After selecting **Zoom** from the **View** menu, you must click on a point with the mouse (or use the cursor keys followed by the **Enter** key). A dialog is displayed where you can change the coordinates of the magnification point and the degree of magnification.

A zoom value greater than 1 causes the display to be magnified and a value less than 1 causes more of the map to fit in the current window.

Window

NOTE: This menu is only shown in map views.

Select **Window** from the **View** menu to drag a rectangular region of the map that will be magnified to fit within the current display. After selecting **Window**, click on a point and drag a box to cover the area of interest. Release the mouse button and a dialog allows you to edit the coordinates of the corners of the box. Click **OK** to accept the values and redraw the display.

Full

NOTE: This menu is only shown in map views.

The **Full** menu contains two submenu choices, labeled **Screen** and **Printer**. Choose the **Screen** menu to display the full map window within the current window on your screen. By default the map window contains your full base map. Choose the **Printer** menu to make sure that the full map window will fit on a single printed page. This effect can also be achieved using the **File->Page Setup** menu and checking the **Size to Page** check-box. You may change the size of the current map window by selecting **Options->Map->Window**.

Display Bounds

NOTE: This menu is only shown in map views.

The **Display Bounds** menu selection indicates whether the coordinates of the current window will be displayed in the corners of the display.. A checkmark appearing on the menu next to the selection means that these coordinates will be displayed. Click on the menu selection to toggle the checkmark on and off.

Reference Head

NOTE: This menu is only shown in Simulation and Flow Model documents.

The **Reference Head** menu indicates whether the arrow for the reference head will be displayed. A checkmark appearing on the menu next to the selection means that the reference head will be displayed. Click on the menu selection to toggle the checkmark on and off.

Recharge Ellipse

NOTE: This menu is only shown in Flow Model documents.

The **Recharge Ellipse** menu indicates whether the recharge ellipse will be displayed. A checkmark appearing on the menu next to the selection means that the reference head will be displayed. Click on the menu selection to toggle the checkmark on and off.

Display Map

NOTE: This menu is only shown in map views.

The **Display Map** menu selection indicates whether the base map will be displayed. A checkmark appearing on the menu next to the selection means that the map will be displayed. Click on the menu selection to toggle the checkmark on and off.

Head Contours

NOTE: This menu is only shown in Flow Model documents.

The **Head Contours** menu selection indicates whether the head/drawddown contours will be displayed. A checkmark appearing on the menu next to the selection means that the contours will be displayed. Click on the menu selection to toggle the checkmark on and off.

Display Color Flood

NOTE: This menu is only shown in map views of Flow Model documents.

The **Display Color Flood** menu selection indicates whether a color flood map of drawdown or hydraulic head will be displayed. A checkmark appearing on the menu next to the selection means that the map will be displayed. Click on the menu selection to toggle the checkmark on and off.

Predicted

The **Predicted** menu is only displayed in a graph view and is only active in the ***Predicted*** tab view. This menu toggles whether the predicted time versus drawdown curves are calculated and displayed on the graph. In cases where it takes a long time to calculate the predicted drawdown versus time curves, it is convenient to turn this option off while manipulating the graph.

Default Legend

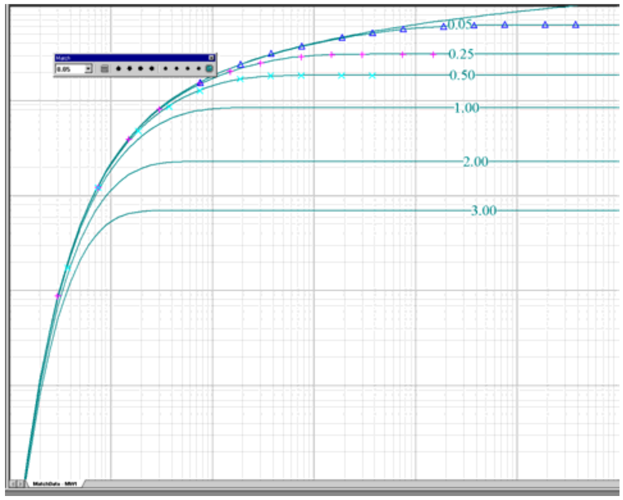
The **Default Legend** menu is used to toggle on/off the display of the default legend for the currently selected view/analysis combination. If no default legend has been saved, nothing will be displayed. Default legends are stored on the computer containing the software. If the document is moved to a different computer different default legends will be displayed. If you want to save the default legend with the document on any system, click on the legend to select it, click the **Edit->Copy** menu, click the **View->Default Legend** menu to turn off the default legend and click the **Edit->Paste** menu and save the document. Now the legend is part of the document.

Reset 3D

The **Reset 3D** menu is used recenter the 3D perspective view. This is sometimes required when changes have been made via the Contour Options property sheet or on screen editing.

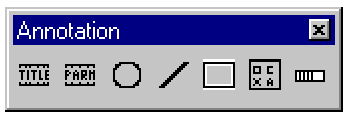
Full Screen

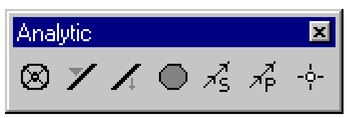
The **Full Screen** menu is only displayed in a graph view and is only active in the match view. It is used to blow up the match view to use the entire computer screen. Everything is removed except a version of the match toolbar that contains a full screen button to go back to normal viewing.



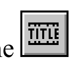
Add Menu

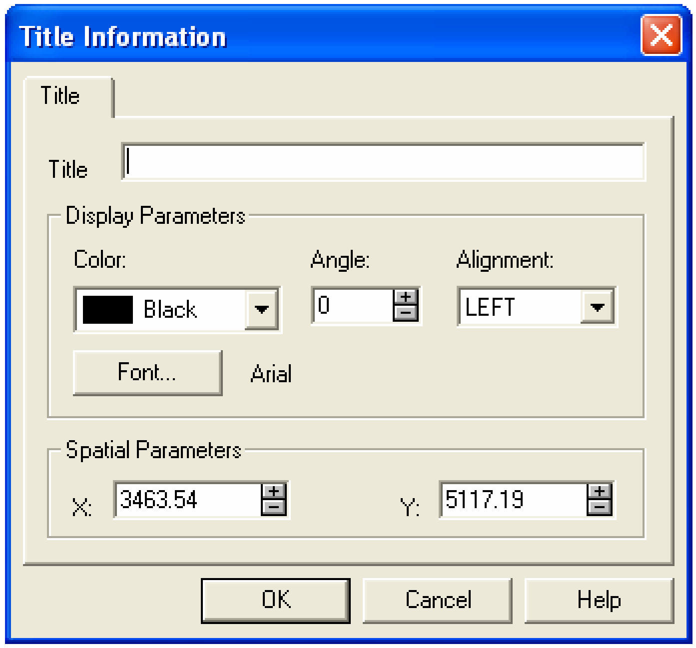
The **Add** menu is for all views except the spreadsheet. These menus can also be selected from the Annotation and Analytic toolbars shown below.





Title

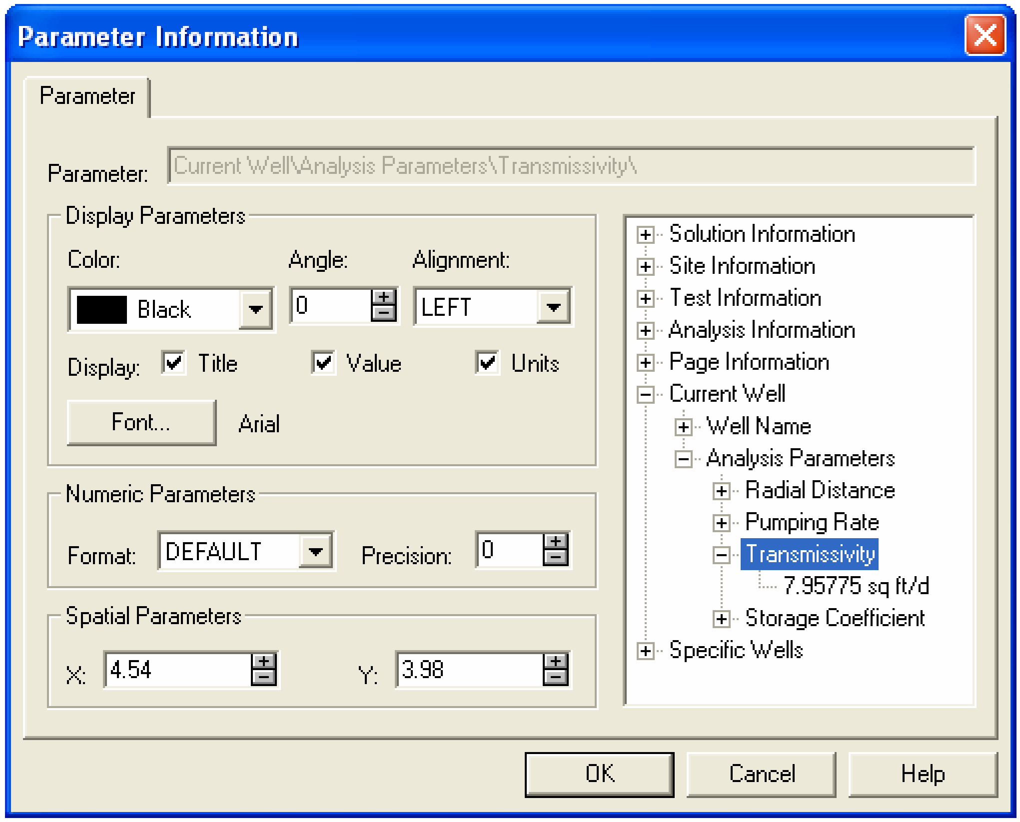
Select the **Title** menu or choose the  button on the Annotation toolbar to add text to the view. Move the cursor to the point where the text will be inserted and click the left mouse button or press the **Enter** key. The *Title Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in map views, the coordinates are in map units. Any number of titles may be added. Titles may also be embedded in Frames and Legends.



Title	The text to display
Display Parameters	
Color	The color to use when displaying the text
Angle	The angle to rotate the text around the location point
Alignment	The alignment of the label relative to the location point
Font	Defines the font, font style and size of the text
Spatial Parameters	
X:	The x-coordinate of the title location in the proper units
Y:	The y-coordinate of the title location in the proper units

Parameter

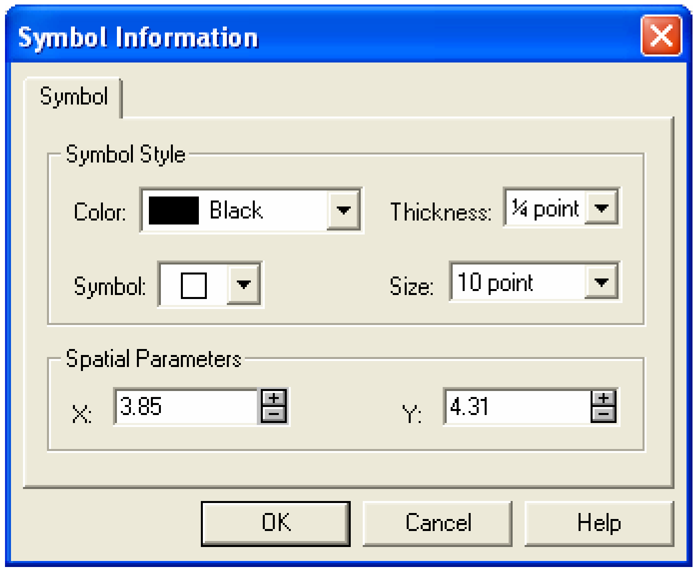
Select the **Parameter** menu or choose the  button on the Annotation toolbar to add parameter text to the view. Move the cursor to the point where the text will be inserted and click the left mouse button or press the **Enter** key. The *Parameter Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in map views, the coordinates are in map units. Any number of parameters may be added. Parameters may also be embedded in Frames and Legends.



Parameter:	A reference to the parameter to be displayed determined by the current selection in the tree control
Display Parameters	
Color	The color to use when displaying the text
Angle	The angle to rotate the text around the location point
Alignment	The alignment of the label relative to the location point
Display Title	If checked, the title of the parameter is displayed
Display Value	If checked, the value of the parameter is displayed
Display Units	If checked, the unit string of the parameter is displayed
Font	Defines the font, font style and size of the text
Numeric Parameters	
Format:	The format to use when displaying numeric values
Precision:	The number of digits after the decimal point
Spatial Parameters	
X:	The x-coordinate of the parameter location in the proper units
Y:	The y-coordinate of the parameter location in the proper units

Symbol

Select the **Symbol** menu or choose the  button on the Annotation toolbar to add a symbol to the view.   Move the cursor to the point where the symbol will be inserted and click the left mouse button or press the **Enter** key. The *Symbol Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in map views, the coordinates are in map units. Any number of symbols may be added. Symbols may also be embedded in Frames, Legends and Header/Footer elements.



Symbol Style

Color	The color to use when displaying the symbol
Thickness	The line thickness to use when displaying the symbol
Symbol	The symbol to display
Size	The size of the symbol in points

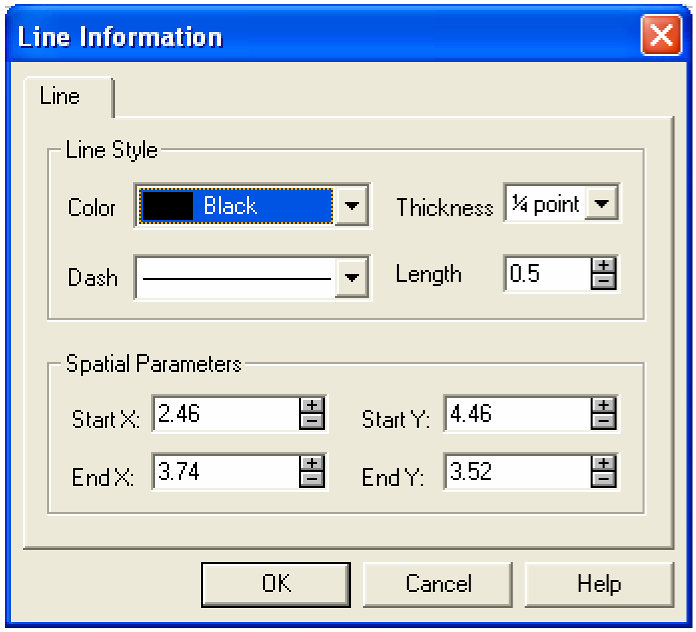
Spatial Parameters

X:	The x-coordinate of the symbol location in the proper units
Y:	The y-coordinate of the symbol location in the proper units

Line

Select the **Line** menu or choose the  button on the Annotation Toolbar to add a line to the view. Move the cursor to the point where the line will start, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the line should end and release the left mouse button or the **Shift** key. The *Line Information* property sheet is presented prompting for the pertinent display characteristics. The

x-coordinate and y-coordinate of both end points are also provided in case you would like to override the values obtained from dragging the line with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in map views, the coordinates are in map units. Any number of lines may be added. Lines may also be embedded in Frames, Legends and Header/Footer elements.



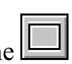
Line Style

Color	The color used to display the line
Thickness	The thickness in points of the line
Dash	The dash pattern to use for the line
Length	The length in inches of the dash pattern for the line

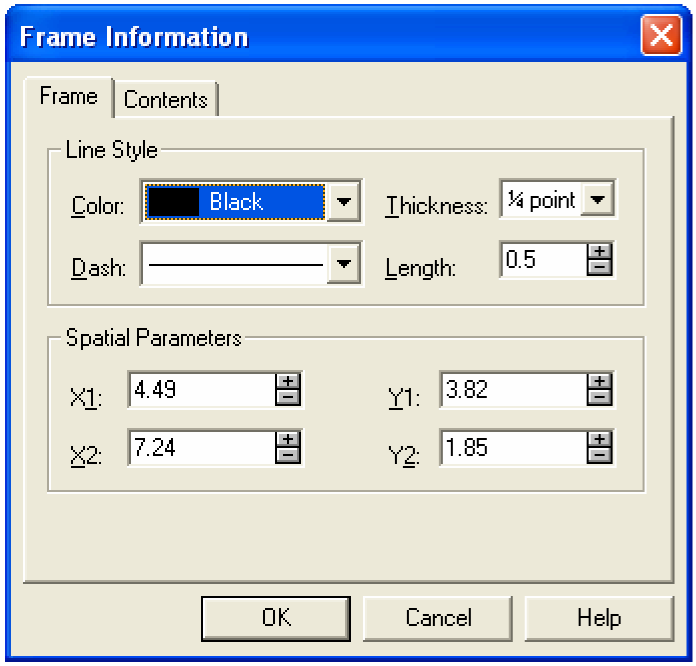
Spatial Parameters

X1:	The x-coordinate of the first end point of the line
Y1:	The y-coordinate of the first end point of the line
X2:	The x-coordinate of the second end point of the line
Y2:	The y-coordinate of the second end point of the line

Frame

Select the **Frame** menu or choose the  button on the Annotation Toolbar to add a frame to the view. Move the cursor to the first corner of the frame, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the second corner of the frame and release the left mouse button or the **Shift** key. The *Frame Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of both corners are also provided in case you would like to override the values obtained from dragging the frame with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in

map views, the coordinates are in map units. Any number of frames may be added. Frames may also be embedded in Legends and Header/Footer elements.

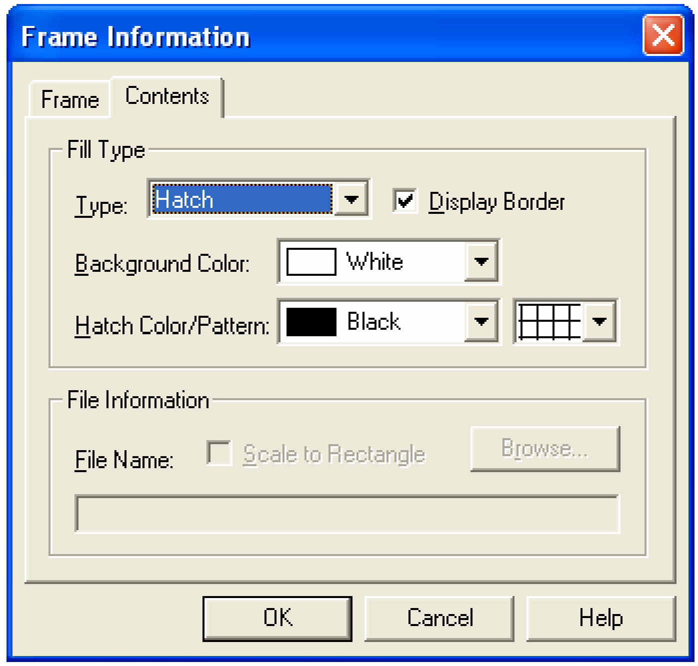


Line Style

Color	The color used to display the optional border
Thickness:	The thickness in points of the optional border
Dash	The dash pattern to use for the optional border
Length	The length in inches of the dash pattern for the optional border

Spatial Parameters

X1:	The x-coordinate of the first corner of the frame
Y1:	The y-coordinate of the first corner of the frame
X2:	The x-coordinate of the second corner of the frame
Y2:	The y-coordinate of the second corner of the frame



Fill Type

Type		Sets the type of fill to use within the frame
	None	The frame is not filled
	Solid	The frame is filled with a solid color
	Hatch	The frame is filled with a hatch pattern
	Bitmap	The frame is filled with a bitmap
	Metafile	The frame is filled with a metafile
Display Border		If checked, a border line is displayed around perimeter of the frame
Background Color		Selects the background color with which to fill the frame
Hatch Color/Pattern		Selects the hatch color and hatch patter to use to fill Hatch type frames

File Information

File Name:	The name of the file that was loaded for Bitmap and Metafile type frames

NOTE: The contents of the file are loaded and stored within the document so the file is no longer required and the name is for historical reference only.

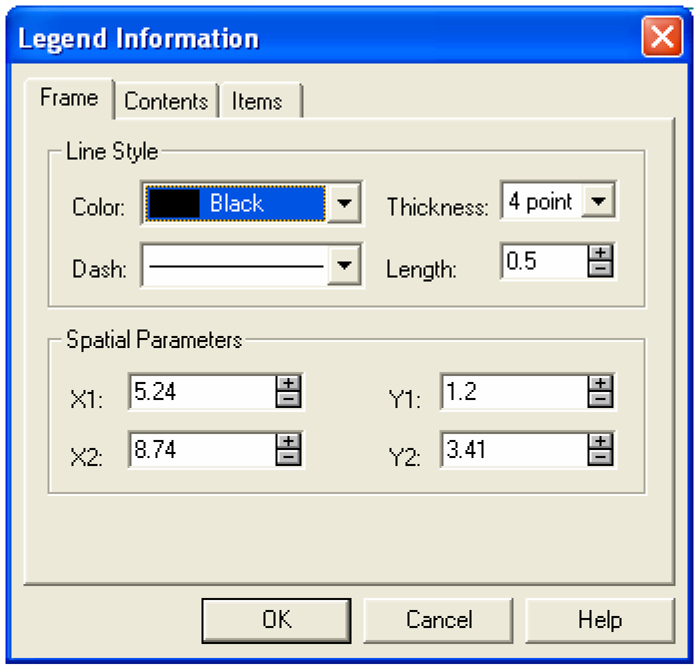
Scale to Rectangle	If checked, the bitmap is expanded/compressed to fill the frame

NOTE: This check-box is usually checked, otherwise, the screen display and printed output will not be the same.

Browse	Click this button to display a standard File Open dialog used to locate the Bitmap or Metafile file stored on the computer.

Legend

Select the **Legend** menu or choose the  button on the Annotation toolbar to add a frame to the view. Move the cursor to the first corner of the frame, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the second corner of the frame and release the left mouse button or the **Shift** key. The *Legend Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of both corners are also provided in case you would like to override the values obtained from dragging the frame with the mouse or keyboard. In graph views, the coordinates are in inches from the lower left corner of the plot; in map views, the coordinates are in map units. Any number of legends may be added. Legends may also be embedded in Header/Footer elements.

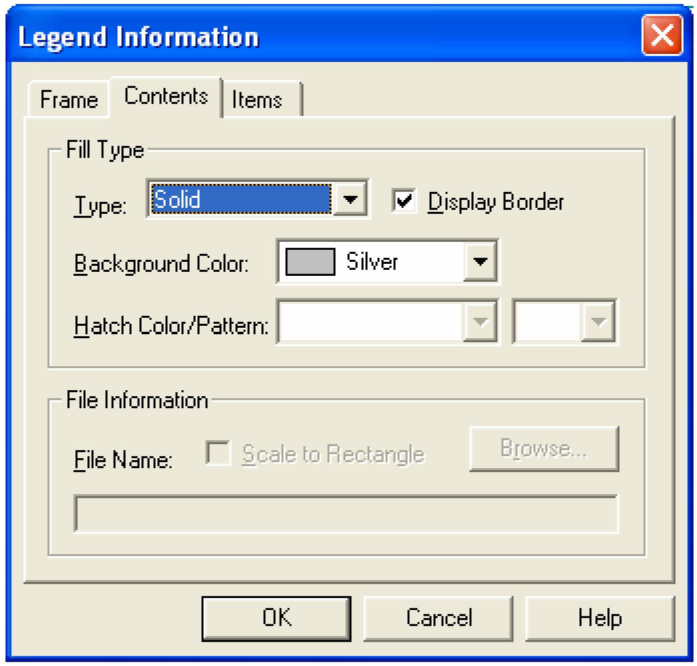


Line Style

Color	The color used to display the optional border
Thickness	The thickness in points of the optional border
Dash	The dash pattern to use for the optional border
Length	The length in inches of the dash pattern for the optional border

Spatial Parameters

X1:	The x-coordinate of the first corner of the legend
Y1:	The y-coordinate of the first corner of the legend
X2:	The x-coordinate of the second corner of the legend
Y2:	The y-coordinate of the second corner of the legend



Fill Type

Type		Sets the type of fill to use within the frame
	None	The frame is not filled
	Solid	The frame is filled with a solid color
	Hatch	The frame is filled with a hatch pattern
	Bitmap	The frame is filled with a bitmap
	Metafile	The frame is filled with a metafile
Display Border		If checked, a border line is displayed around perimeter of the frame
Background Color		Selects the background color with which to fill the frame
Hatch Color/Pattern		Selects the hatch color and hatch patter to use to fill Hatch type frames

File Information

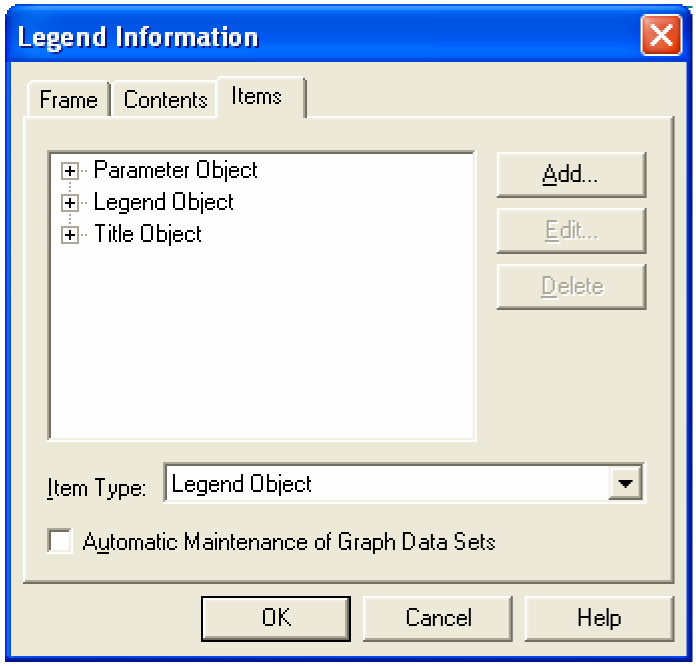
File Name: The name of the file that was loaded for Bitmap and Metafile type frames

NOTE: The contents of the file are loaded and stored within the document so the file is no longer required and the name is for historical reference only.

Scale to Rectangle If checked, the bitmap is expanded/compressed to fill the frame

NOTE: This check-box is usually checked, otherwise, the screen display and printed output will not be the same.

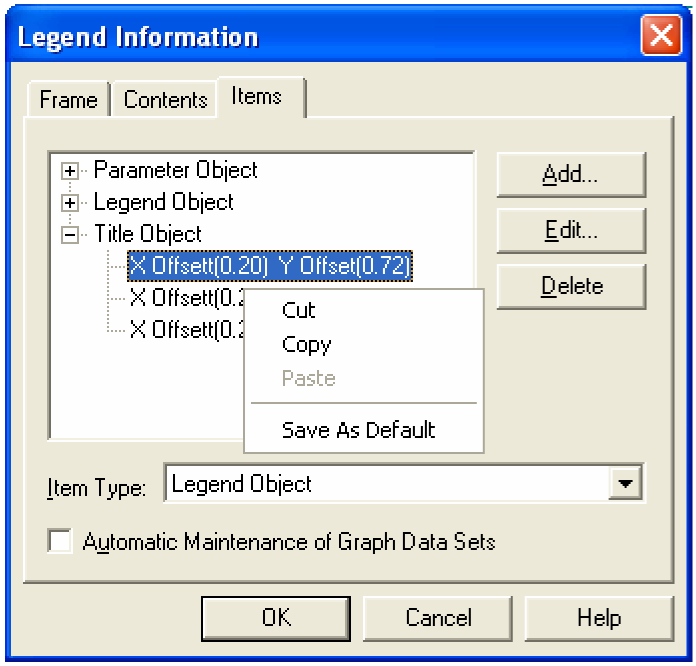
Browse Click this button to display a standard File Open dialog used to locate the Bitmap or Metafile file stored on the computer.



Add:	Adds an item, of the type selected in *Item Type*, to the legend
Edit:	Presents the property sheet for the item selected in the tree control
Delete:	Deletes the selected item currently selected in the tree control
Item Type:	Specifies the currently active item to be added when the **Add** button is clicked

The **Automatic Maintenance of Graph Data Sets** check-box only appears if the parent view of the legend is a graph. If this check-box is checked, symbols and labels for each data set in the graph will be automatically displayed within the legend. If the legend is higher than it is wide, the symbol/label items will be spaced evenly vertically and centered horizontally within the legend. If the legend is wider than it is high, the symbol/label items will be spaced evenly horizontally and centered vertically within the legend. Typically, a legend set up using *Automatic Maintenance of Graph Data Sets* does not contain any other elements although it is likely embedded in another legend element.

The **Automatic Maintenance of Color Flood Legend** check-box only appears if the parent view of the legend is a map view of a **Flow Model** document. If this check-box is checked, the legend will contain a color flood legend. The color flood legend will fill the legends frame so if you want to add other text, embed this legend within a larger one. The colors and label characteristics of this color flood legend are controlled via the *Contour Options* property sheet on the ***Flood Colors*** and ***Flood Legend*** tabs.

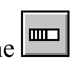


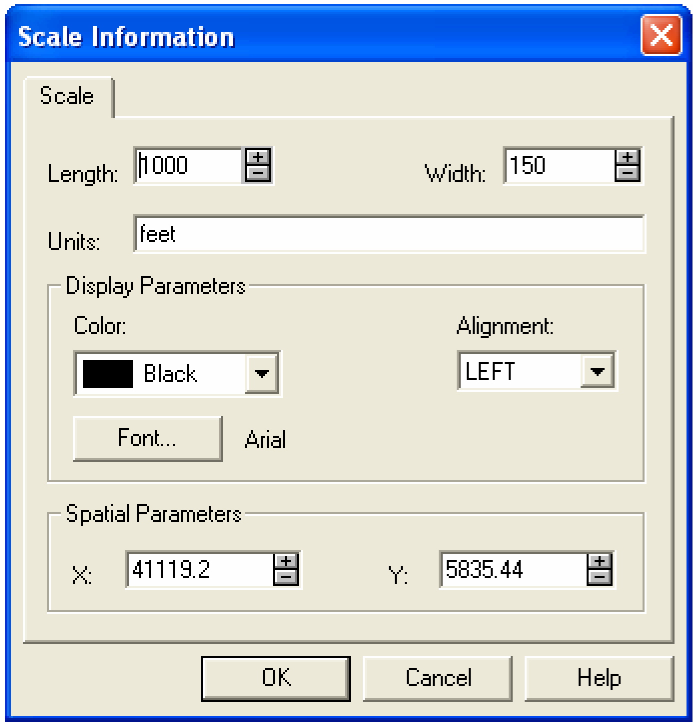
Context Menu:

The tree control on the ***Items*** tab has a context menu that allows you to Cut/Copy/Paste items within and across legend objects. Also, the **Save As Default** menu allows you to save the legend to be used as the default whenever another document is created to use this type of analysis. Refer to the Default Legend section for more details.

Scale Bar

NOTE: This menu is only shown in map views.

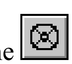
Select the **Scale Bar** menu or choose the  button on the Annotation toolbar to add a scale bar to the view. Move the cursor to the point where the scale bar will be inserted and click the left mouse button or press the **Enter** key. The *Scale Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In map views, the coordinates are in map units. Any number of scale bars may be added.



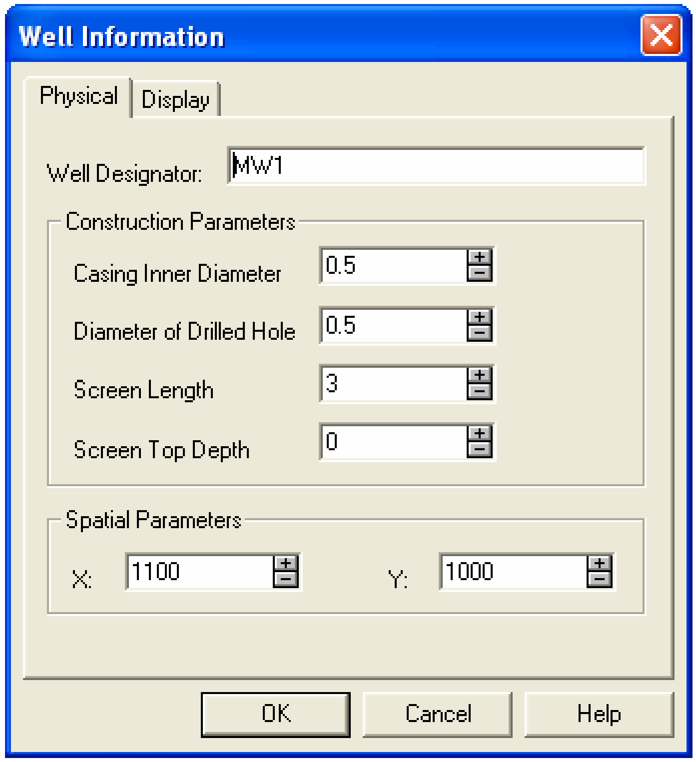
Length:	The length of the scale bar in map units
Width:	The width of the scale bar in map units
Display Parameters	
Color:	The color to use when displaying the symbol
Alignment:	The alignment of the scale bar relative to the location point
Font	Defines the font, font style, size for the scale bar
Spatial Parameters	
X:	The x-coordinate of the well location in map units
Y:	The y-coordinate of the well location in map units

Well

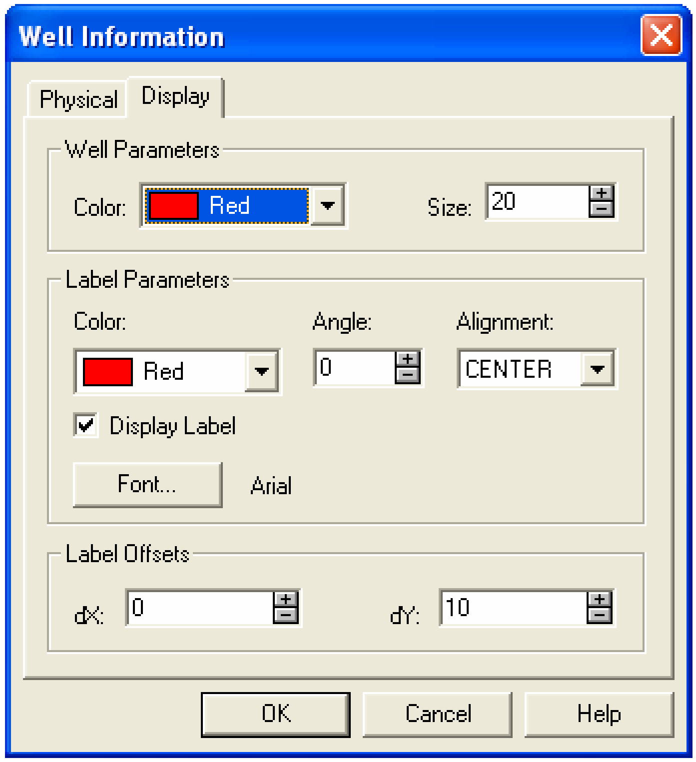
NOTE: This menu is only shown in map views.

Select the **Well** menu or choose the  button on the Analytic toolbar to add a well to the view. Move the cursor to the point where the well will be inserted and click the left mouse button or press the **Enter** key. The *Well Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In map views, the coordinates are in map units. Any number of wells may be added.

The following property sheet is for wells in other than flow model documents.



Well Designator:	Unique name for the well
Construction Parameters	
Casing Inner Diameter	The inner diameter of the well casing
Diameter of Drilled Hole	The diameter of the drilled hole
Screen Length	The length of the well screen
Screen Top Depth	The depth to top of the well screen relative to the top of the aquifer
Spatial Parameters	
X:	The x-coordinate of the well location in map units
Y:	The y-coordinate of the well location in map units



Well Parameters

Color	The color to use when displaying the well symbol
Size	The display size in map units of the well symbol

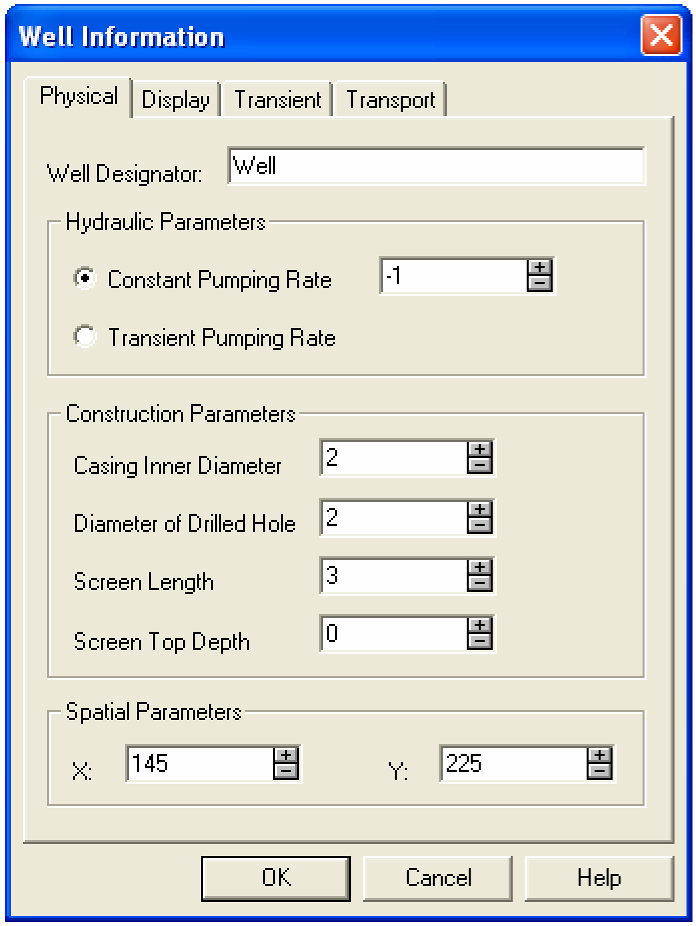
Label Parameters

Color	The color to use when displaying the well label
Angle	The angle to rotate the text when displaying the well label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the well designator is used to label the well
Font	Defines the font, font style, size and effects for the label

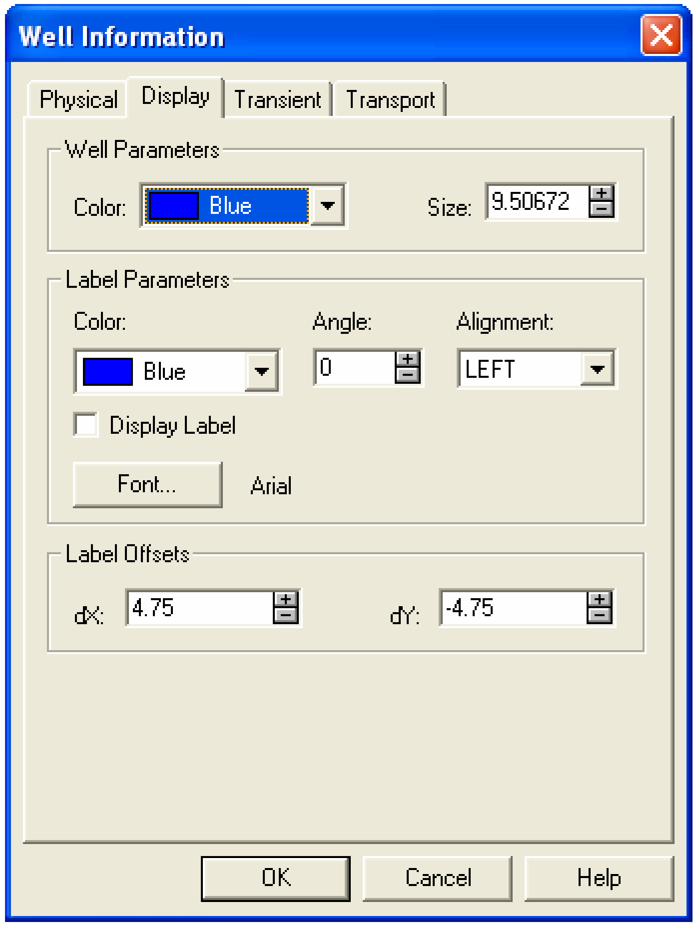
Label Offsets

dX:	The distance in map units to offset the label in x from the x-coordinate of the well location
dY:	The distance in map units to offset the label in y from the y-coordinate of the well location

The following property sheet is for flow model documents.



Well Designator:	Name for the well
Hydraulic Parameters	
Constant Pumping Rate	If set, this well pumps at a constant rate whose value is contained in the adjacent data field
Transient Pumping Rate	If set, this well pumps at a variable rate and the pumping schedule is contained on the ***Transient*** tab
Construction Parameters	
Casing Inner Diameter	The inner diameter of the well casing
Diameter of Drilled Hole	The diameter of the drilled hole
Screen Length	The length of the well screen
Screen Top Depth	The depth to top of the well screen relative to the top of the aquifer
Spatial Parameters	
X:	The x-coordinate of the well location in map units
Y:	The y-coordinate of the well location in map units



Well Parameters

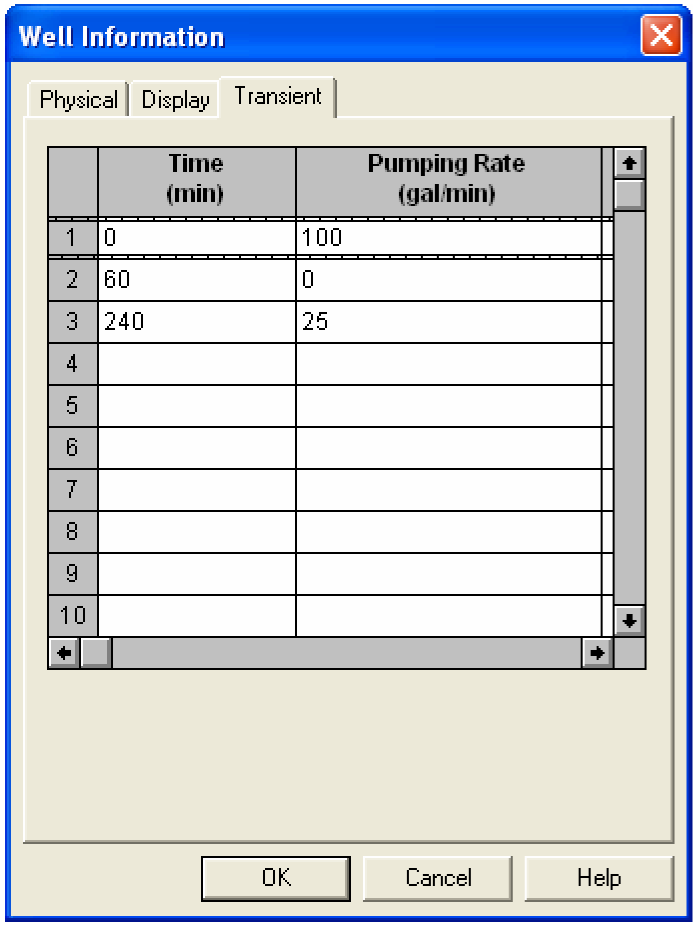
Color	The color to use when displaying the well symbol
Size	The display size in map units of the well symbol

Label Parameters

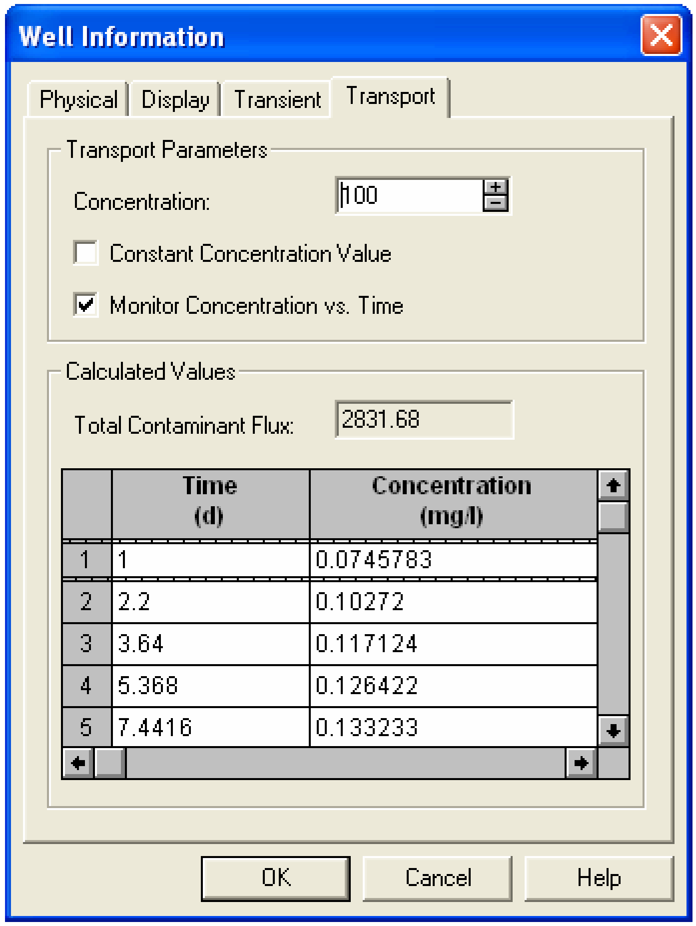
Color	The color to use when displaying the well label
Angle	The angle to rotate the text when displaying the well label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the well designator is used to label the well
Font	Defines the font, font style, size and effects for the label

Label Offsets

dX:	The distance in map units to offset the label in x from the x-coordinate of the well location
dY:	The distance in map units to offset the label in y from the y-coordinate of the well location



This spreadsheet defines the pumping schedule used for this well. This pumping schedule is only used if the **Transient Pumping Rate** radio button has been set on the ***Physical*** tab and the solution supports variable pumping rates.



Transport Parameters

Concentration:	The concentration of injected water
Constant Concentration Value	The concentration value is maintained as constant at the injection well
Monitor Concentration vs. Time	Concentration versus time data is calculated for the well based on the transport time stepping scheme

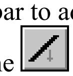
Calculated Values

Total Contaminant Flux:	The total contaminant flux rate contributed by the well

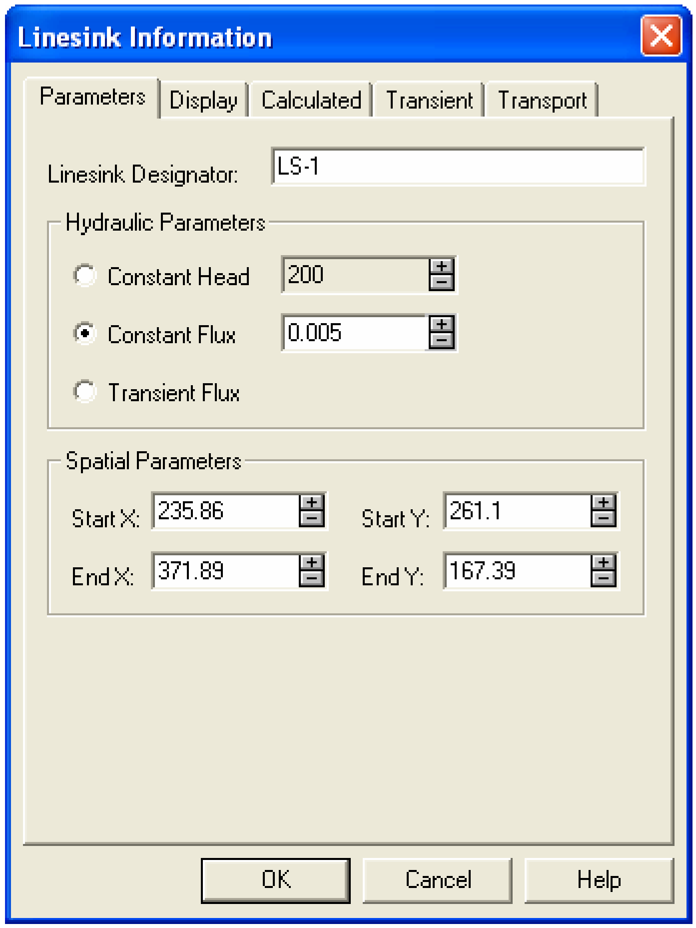
The spreadsheet presents the concentration versus time values if the **Monitor Concentration Vs. Time** checkbox has been checked and the model run..

Head/Flux Linesink

NOTE: This menu is only shown in map views of Flow Model documents.

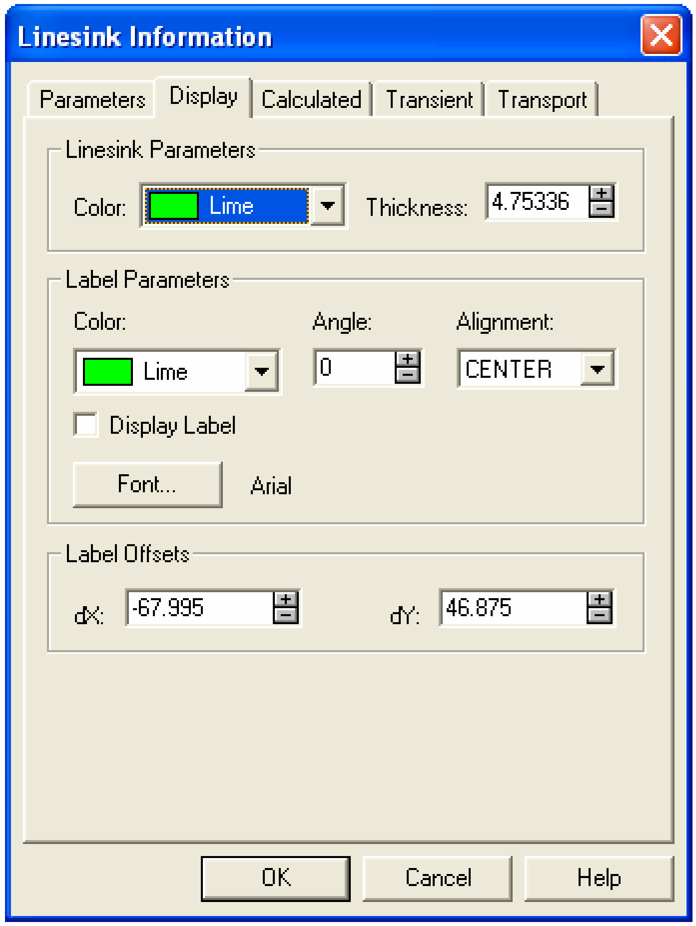
Select the **Head Linesink** menu or the  button on the Analytic toolbar to add a head linesink to the model. Select the **Flux Linesink** menu or choose the  button on the Analytic toolbar to add a flux linesink to the model. Move the cursor to the point where the linesink will start, click and hold the left mouse button or press

and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the linesink should end and release the left mouse button or the **Shift** key. The *Linesink Information* property sheet is presented prompting for the pertinent display and hydraulic characteristics. The x-coordinate and y-coordinate of both end points are also provided in case you would like to override the values obtained from dragging the line with the mouse or keyboard. The coordinates are in map units and any number of linesinks may be added.



Linesink Designator:	Name for the linesink
Hydraulic Parameters	
Constant Head	If set, this linesink is a head linesink whose head value is contained in the adjacent data field
Constant Flux	If set, this linesink is a flux linesink whose flux rate is contained in the adjacent data field
Transient Flux	If set, this linesink is a flux linesink whose flux rate is variable and the rate schedule is contained on the ***Transient*** tab
Spatial Parameters	
Start X:	The x-coordinate of the starting point of the linesink
Start Y:	The y-coordinate of the starting point of the linesink
End X:	The x-coordinate of the ending point of the linesink

End Y:	The y-coordinate of the ending point of the linesink



Linesink Parameters

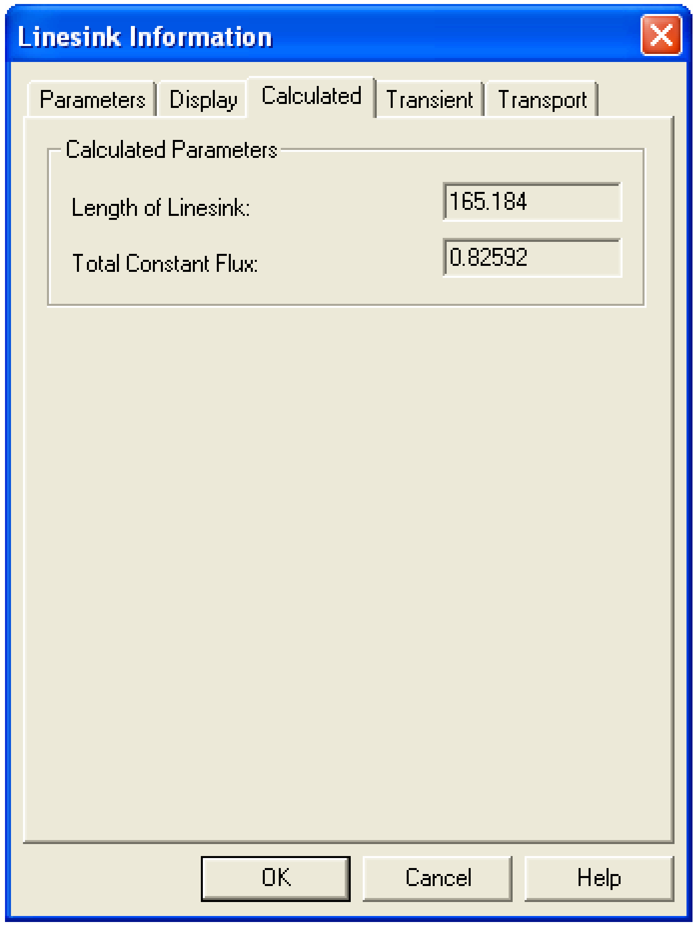
Color	The color used to display the linesink
Thickness	The thickness in map units of the linesink

Label Parameters

Color	The color to use when displaying the well label
Angle	The angle to rotate the text when displaying the linesink label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the linesink designator is used to label the linesink
Font	Defines the font, font style, size and effects for the label

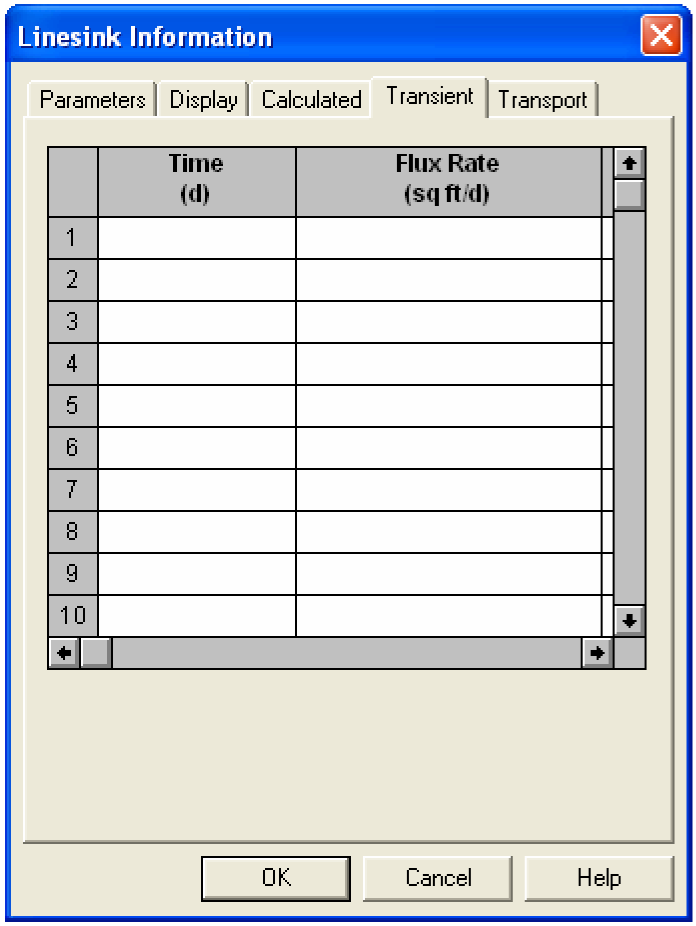
Label Offsets

dX:	The distance in map units to offset the label in x from the x-coordinate of the center of the linesink
dY:	The distance in map units to offset the label in y from the y-coordinate of the center of the linesink

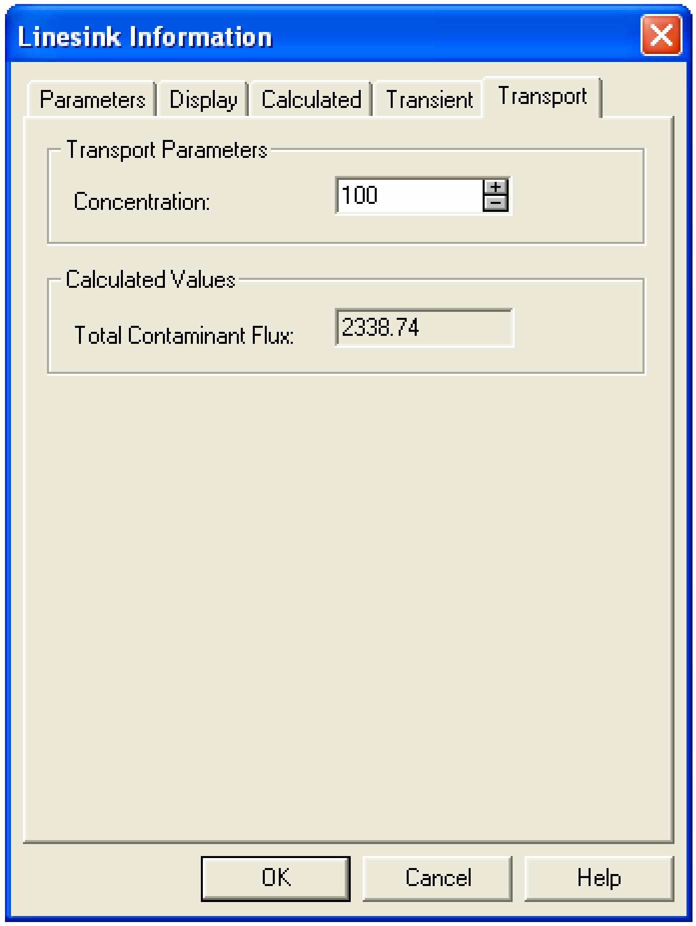


Calculated Parameters

Length of Linesink	The total length of the linesink in map units
Total Constant Flux	The total flux rate generated by the linesink



This spreadsheet defines the pumping schedule used for this linesink. This pumping schedule is only used if the **Transient Flux** radio button has been set on the ***Physical*** tab.



Transport Parameters

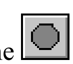
Concentration	The concentration of water injected by the linesink

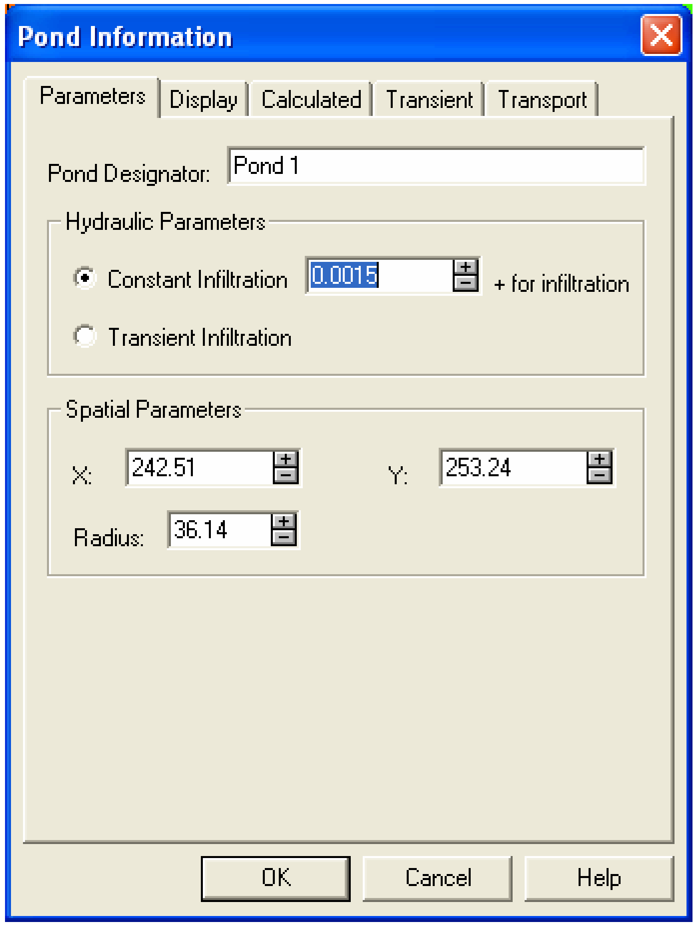
Calculated Values

Total Contaminant	The total contaminant flux rate generated by the linesink

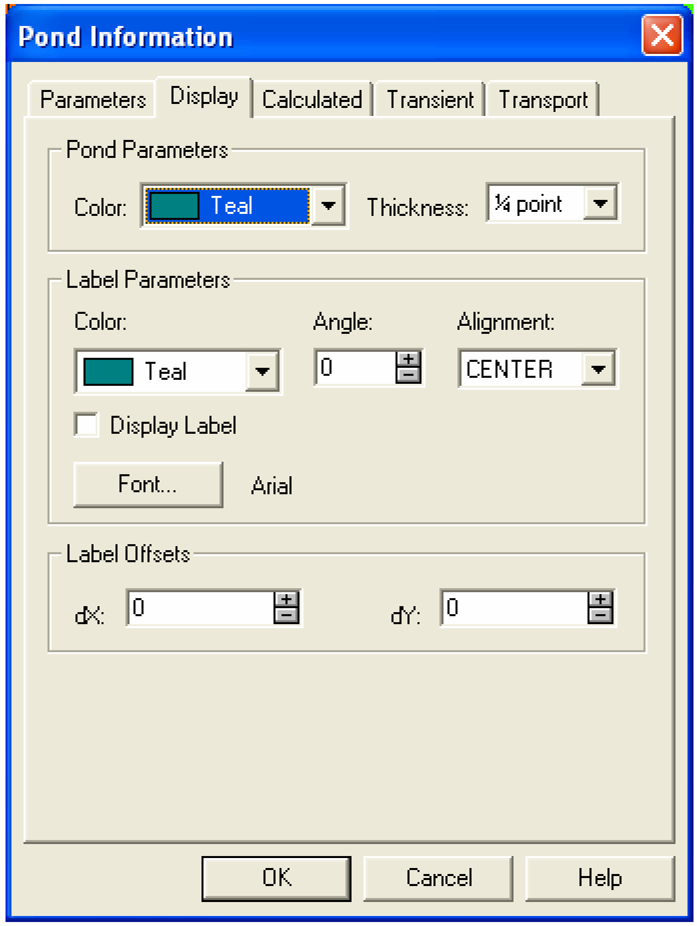
Pond

NOTE: This menu is only shown in map views of Flow Model documents.

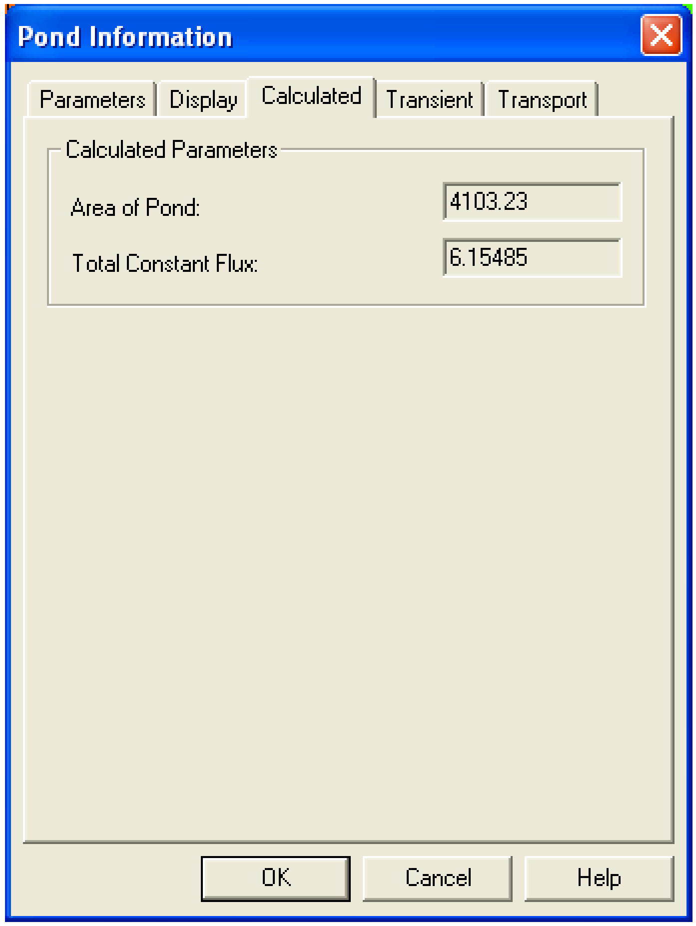
Select the **Pond** menu or the  button on the Analytic toolbar to add a pond to the model. Move the cursor to the point at the center of the pond, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the pond should end and release the left mouse button or the **Shift** key. The *Pond Information* property sheet is presented prompting for the pertinent display and hydraulic characteristics. The x-coordinate and y-coordinate of center and the radius are also provided in case you would like to override the values obtained from dragging the pond with the mouse or keyboard. The coordinates are in map units and any number of ponds may be added.



Pond Designator:	Name for the pond
Hydraulic Parameters	
Constant Infiltration	If set, this pond has a constant infiltration rate whose value is contained in the adjacent data field
Transient Infiltration	If set, this pond has a variable infiltration rate and the infiltration schedule is contained on the ***Transient*** tab
Spatial Parameters	
X:	The x-coordinate of the center of the pond in map units
Y:	The y-coordinate of the center of the pond in map units
Radius:	The radius of the pond in map units

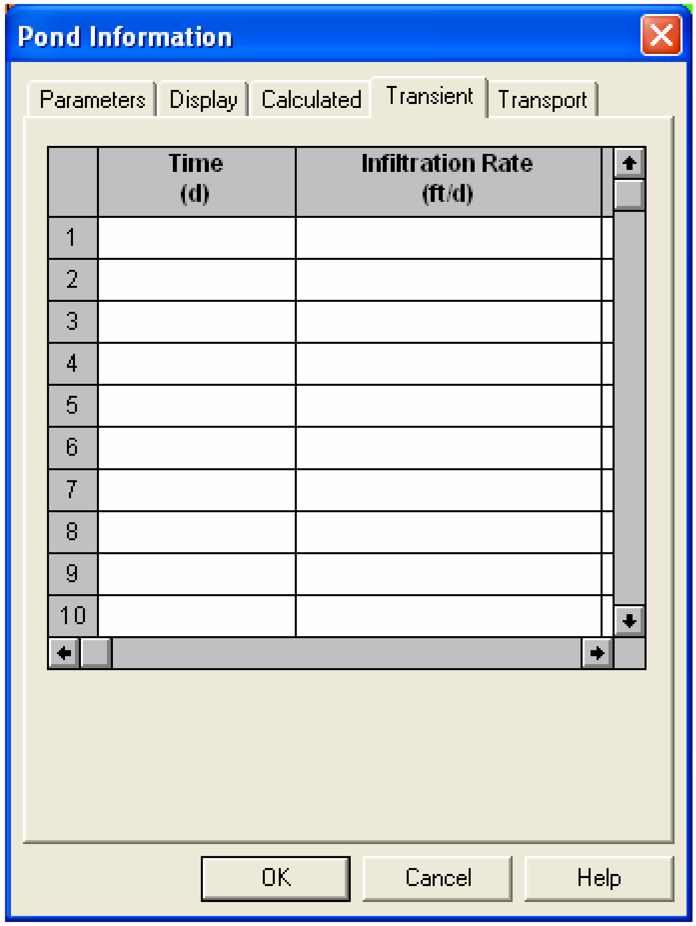


Pond Parameters	
Color	The color used to display the pond
Thickness	The thickness of the line used to display the perimeter of the pond in points
Label Parameters	
Color	The color to use when displaying the pond label
Angle	The angle to rotate the text when displaying the pond label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the pond designator is used to label the pond
Font	Defines the font, font style, size and effects for the label
Label Offsets	
dX:	The distance in map units to offset the label in x from the x-coordinate of the center of the pond
dY:	The distance in map units to offset the label in y from the y-coordinate of the center of the pond

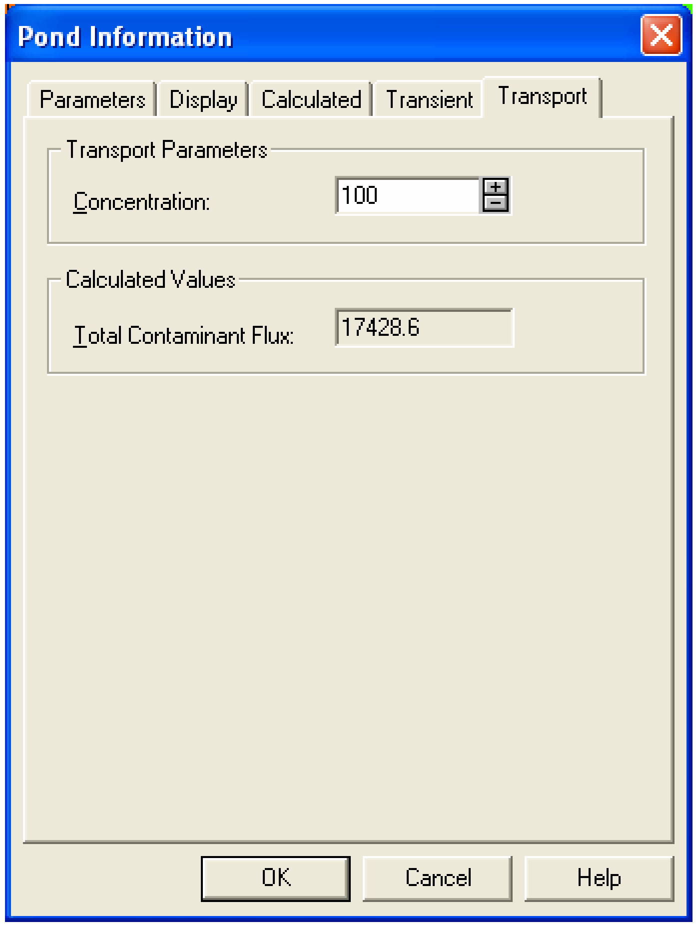


Calculated Parameters

Area of Pond:	The total of the pond in map units
Total Constant Flux:	The total flux rate generated by the pond



This spreadsheet defines the infiltration schedule used for this pond. This infiltration schedule is only used if the **Transient Infiltration** radio button has been set on the ***Physical*** tab.



Transport Parameters

Concentration:	The concentration of water added to the model by the pond.

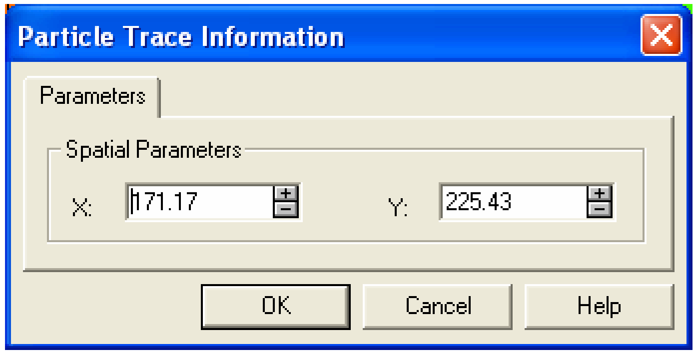
Calculated Values

Total Contaminant Flux:	The total contaminant rate generated by the pond

Particle

NOTE: This menu is only shown in map views of Flow Model documents.

Select the **Particle->Single** menu or choose the  button on the Analytic toolbar to add a particle trace to the model. Move the cursor to the point where the particle trace will be inserted and click the left mouse button or press the **Enter** key. The *Particle Trace Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. The coordinates are in map units and any number of particle traces may be added.



Spatial Parameters

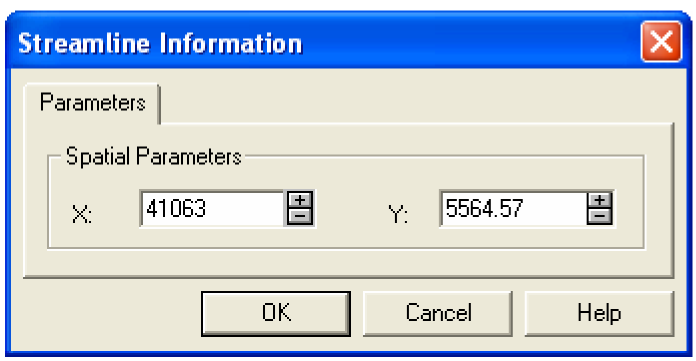
X: The x-coordinate of the start of particle trace in map units

Y: The y-coordinate of the start of particle trace in map units

Streamline

NOTE: This menu is only shown in map views of Flow Model documents.

Select the **Streamline->Single** menu or choose the  button on the Analytic toolbar to add a streamline to the model. Move the cursor to the point where the streamline will be inserted and click the left mouse button or press the **Enter** key. The *Streamline Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. The coordinates are in map units and any number of streamlines may be added.



Spatial Parameters

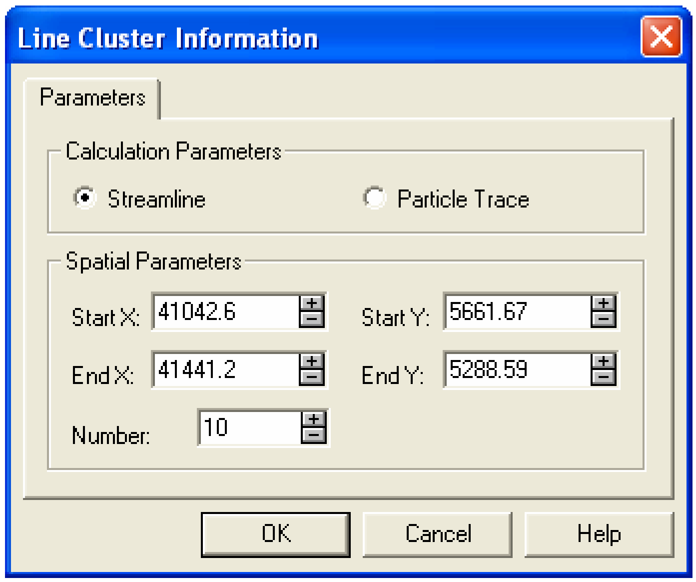
X: The x-coordinate of the start of streamline in map units

Y: The y-coordinate of the start of streamline in map units

Line Cluster

NOTE: This menu is only shown in map views of Flow Model documents.

Select the **Streamline->Line** or **Particle->Line** menu to add a line of streamlines or particle traces to the model. Move the cursor to the point where the line cluster will start, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the line cluster should end and release the left mouse button or the **Shift** key. The *Line Cluster Information* property sheet is presented prompting for the pertinent display and calculation characteristics. The x-coordinate and y-coordinate of both end points are also provided in case you would like to override the values obtained from dragging the line with the mouse or keyboard. The coordinates are in map units and any number of linesinks may be added.



Calculation Parameters

Streamline	If set, the line cluster contains streamlines
Particle Trace	If set, the line cluster contains particle traces

Spatial Parameters

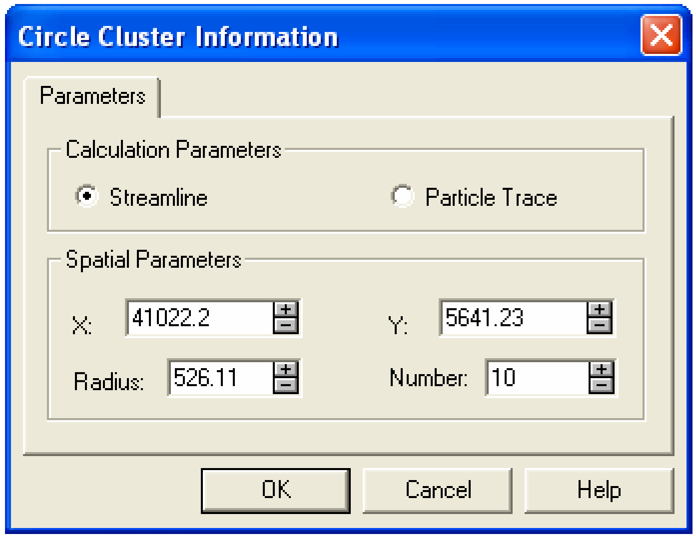
Start X:	The x-coordinate of the starting point of the line cluster
Start Y:	The y-coordinate of the starting point of the line cluster
End X:	The x-coordinate of the ending point of the line cluster
End Y:	The y-coordinate of the ending point of the line cluster
Number:	The number of equally spaced streamlines or particle traces contained in the line cluster

Circle Cluster

NOTE: This menu is only shown in map views of Flow Model documents.

Select the **Streamline->Circle** or **Particle->Circle** menu to add a circle of streamlines or particle traces to the model. Move the cursor to the point at the center

of the circle, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the circle should end and release the left mouse button or the **Shift** key. The *Circle Cluster Information* property sheet is presented prompting for the pertinent display and calculation characteristics. The x-coordinate and y-coordinate of center and the radius are also provided in case you would like to override the values obtained from dragging the circle with the mouse or keyboard. The coordinates are in map units and any number of circle clusters may be added.



Calculation Parameters

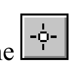
Streamline	If set, the circle cluster contains streamlines
Particle Trace	If set, the circle cluster contains particle traces

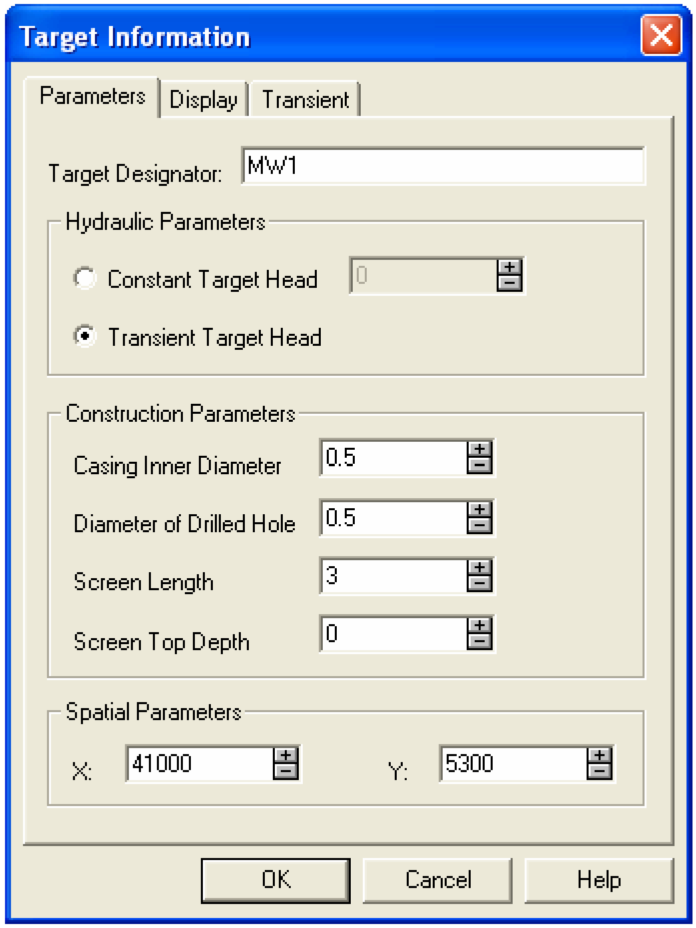
Spatial Parameters

X:	The x-coordinate of the center of the circle cluster in map units
Y:	The y-coordinate of the center of the circle cluster in map units
Radius:	The radius of the circle cluster in map units
Number:	The number of equally spaced streamlines or particle traces contained around the perimeter of the circle cluster

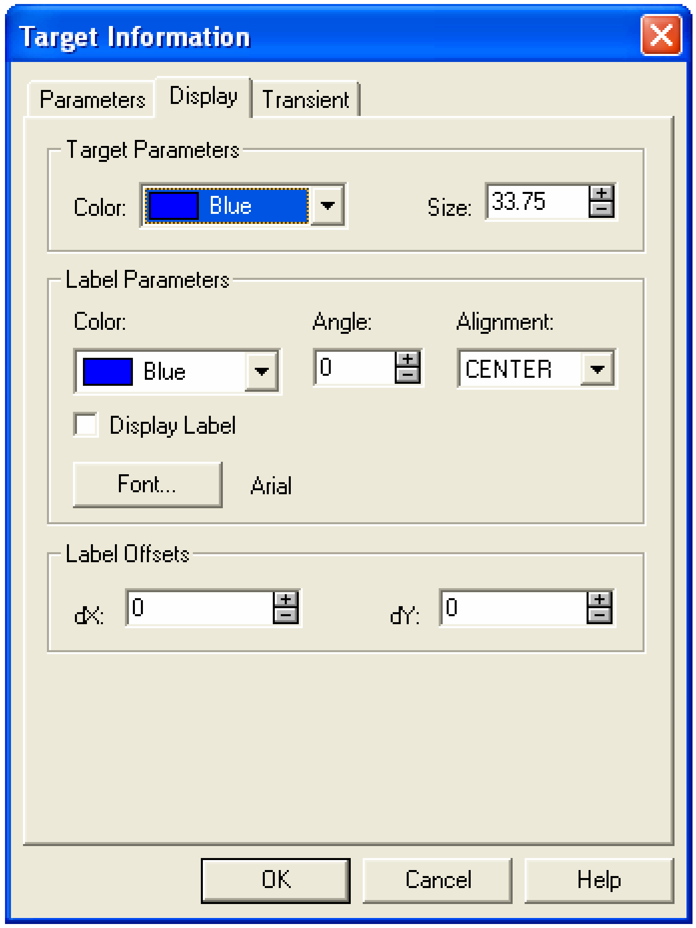
Target

NOTE: This menu is only shown in map views of Flow Model documents.

Select the **Target** menu or choose the  button on the Analytic toolbar to add a target to the view. Move the cursor to the point where the well will be inserted and click the left mouse button or press the **Enter** key. The *Target Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of the point are also provided in case you would like to override the values obtained from setting the location with the mouse or keyboard. In map views, the coordinates are in map units. Any number of targets may be added.



Target Designator:	Name for the target
Hydraulic Parameters	
Constant Target Head	If set, the target head is constant rate with a value contained in the adjacent data field
Transient Target Head	If set, this target is variable in time and the values are contained on the ***Transient*** tab
Construction Parameters	
Casing Inner Diameter	The inner diameter of the target well casing
Diameter of Drilled Hole	The diameter of the drilled hole
Screen Length	The length of the target well screen
Screen Top Depth	The depth to top of the target well screen relative to the top of the aquifer
Spatial Parameters	
X:	The x-coordinate of the target location in map units
Y:	The y-coordinate of the target location in map units



Target Parameters

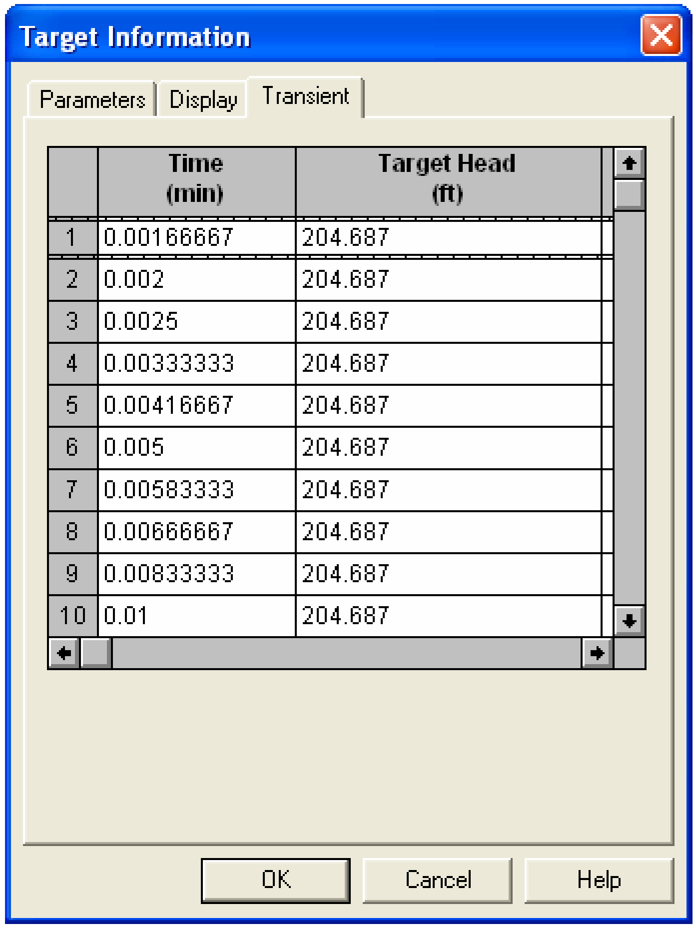
Color	The color to use when displaying the target symbol
Size	The display size in map units of the target symbol

Label Parameters

Color	The color to use when displaying the target label
Angle	The angle to rotate the text when displaying the target label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the target designator is used to label the target
Font	Defines the font, font style, size and effects for the label

Label Offsets

dX:	The distance in map units to offset the label in x from the x-coordinate of the target location
dY:	The distance in map units to offset the label in y from the y-coordinate of the target location

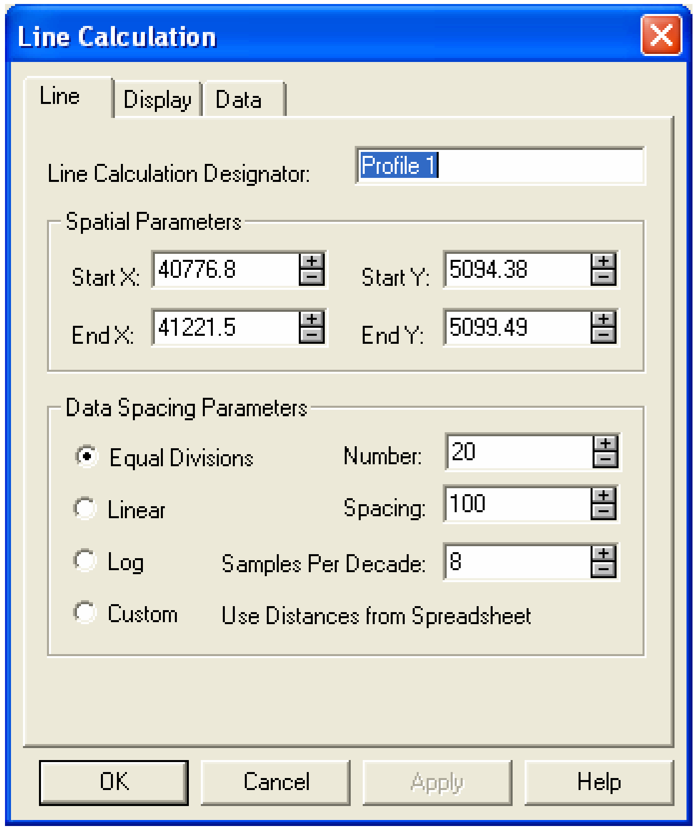


This spreadsheet defines the head versus time data for this target. These heads are only used if the **Transient Target Head** radio button has been set on the ***Physical*** tab.

Line Calculation

NOTE: This menu is only shown in map views of Simulation and Flow Model documents.

Select the **Line Calculation** menu or choose the  button on the Analytic toolbar to add a line calculation to the view. Move the cursor to the point where the line will start, click and hold the left mouse button or press and hold the **Shift** key and press the **Enter** key. Drag the mouse or use the arrow keys to move the cursor to the point where the line should end and release the left mouse button or the **Shift** key. The *Line Calculation Information* property sheet is presented prompting for the pertinent display characteristics. The x-coordinate and y-coordinate of both end points are also provided in case you would like to override the values obtained from dragging the line with the mouse or keyboard. The coordinates are in map units.



Line Calculation Designator:	Each line calculation must have a unique name so that it can be identified on view tabs.

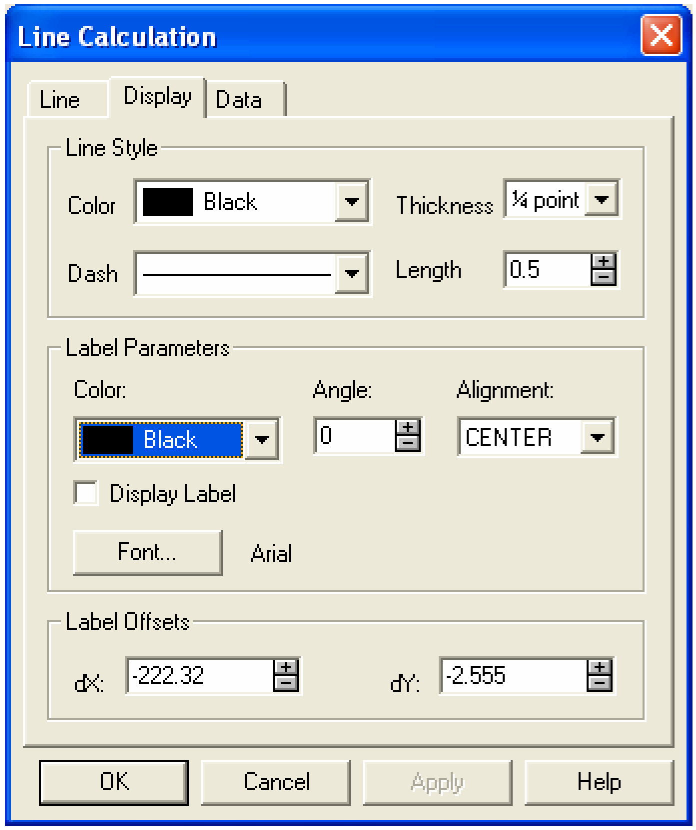
Spatial Parameters

Start X:	The x-coordinate of the starting point for the line calculation in map coordinates. Distances are calculated relative to this point.
Start Y:	The y-coordinate of the starting point for the line calculation in map coordinates. Distances are calculated relative to this point.
End X:	The x-coordinate of the ending point for the line calculation in map coordinates.
End Y:	The y-coordinate of the ending point for the line calculation in map coordinates.

Data Spacing Parameters

Equal Divisions	Data points will be calculated at equal intervals between the start and end points of the line calculation.
Number:	The number of equal intervals to calculate between the start and end points of the line calculations.
Linear	Data points will be calculated at specified intervals starting at the start point of the line calculation.
Spacing:	The spacing, in map units, between adjacent points in the line calculation.

Log	Data points will be calculated at logarithmic intervals starting at the start point of the line calculation.
Samples Per Decade:	The number of logarithmic intervals between adjacent points in the line calculation.
Custom	Distances, in map units, entered into the spreadsheet on the ***Data*** tab will be used for the calculations.



Line Style

Color:	Sets the color of the line connecting the data points
Thickness:	Sets the thickness in points of the line connecting the data points
Dash:	Sets the dash pattern to use for the line connecting the data points
Length:	Sets the length in inches of the dash pattern for the line connecting the data points

Label Parameters

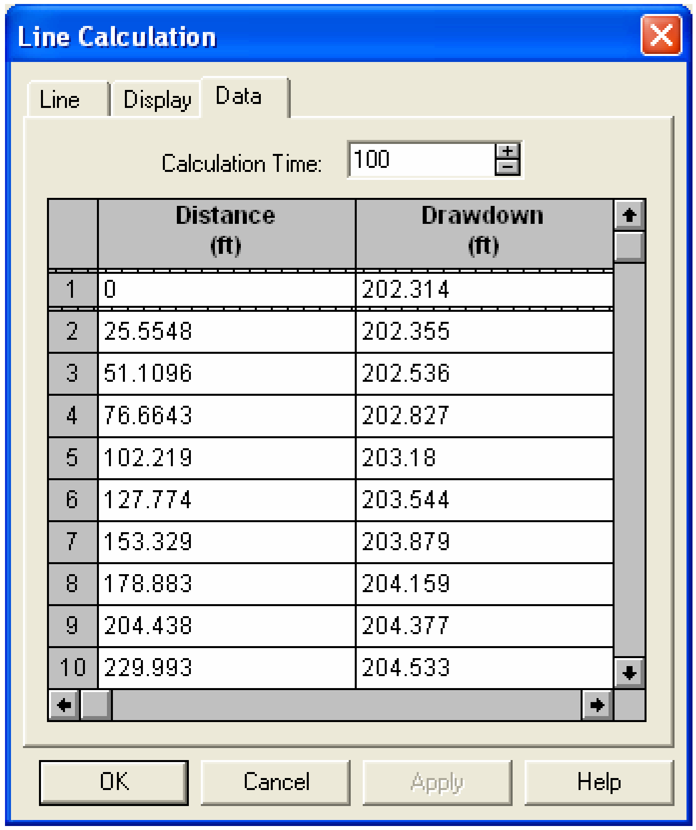
Color	The color to use when displaying the line calculation label
Angle	The angle to rotate the text when displaying the line calculation label
Alignment	The alignment of the label relative to the label location
Display Label	If checked, the line calculation designator is used to label the well

Font Defines the font, font style, size and effects for the label

Label Offsets

dX: The distance, in map units, to offset the label in x from the x-coordinate of the line calculation center point.

dY: The distance, in map units, to offset the label in y from the y-coordinate of line calculation center point.



Calculation Time: The time at which to calculate head/drawdown versus distance data.

Spreadsheet The data in the spreadsheet reflect the calculated data points. If **Custom** has been selected on the ***Line*** tab, you can edit the number lines and the values of the distances at which to calculate; if **Custom** was not selected, any changes made will be ignored.

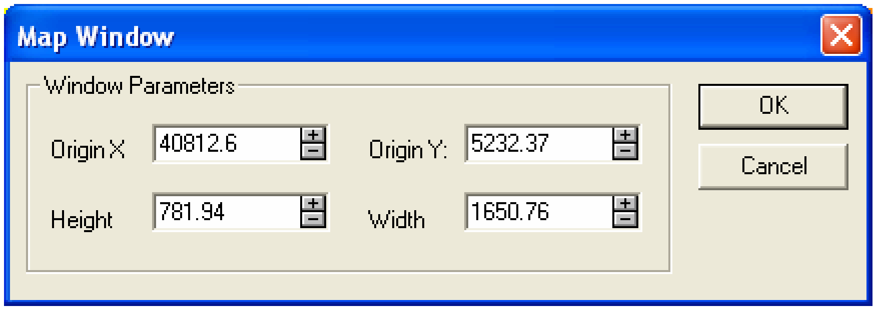
Options Menu

NOTE: The Options menu is only displayed in map views.

The **Options** menu contains selections for customizing the Aquifer^{Win32} simulation model and associated map. You may change the appearance of the site map and/or contour map using these menus; each of these menus is described below.

Map

The **Map** menu selection contains three submenus, **Window, Parameters** and **Locate**. The **Window** option allows you to change the size and location of the map window. The map window by default ranges from 0 to 10000 in both the x and y axes. If a base map is imported, the size of the map window is determined from the base map. The map window is the region that is displayed when you choose **View->Full->Screen** and the area printed. Selecting **Options->Map->Window** allows you to drag a rectangle on the screen to redefine the new map window. Doing so confirms the boundaries using the *Map Window* dialog as shown below.

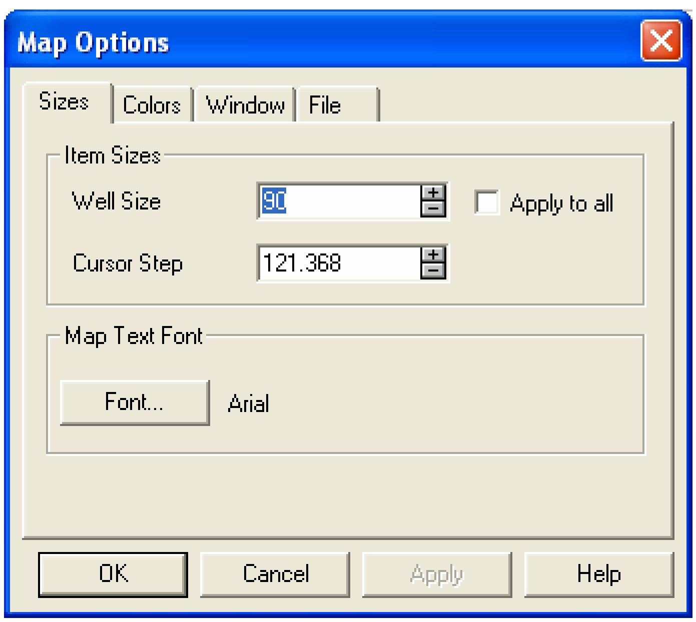


Window Parameters

Origin X	The x-coordinate of the origin of the map in map units
Origin Y	The y-coordinate of the origin of the map in map units
Height	The height of the map in map units
Width	The width of the map in map units

The **Parameters** menu selection displays the *Map Options* property sheet described below.

The following property sheet is for map views in other than flow model documents.

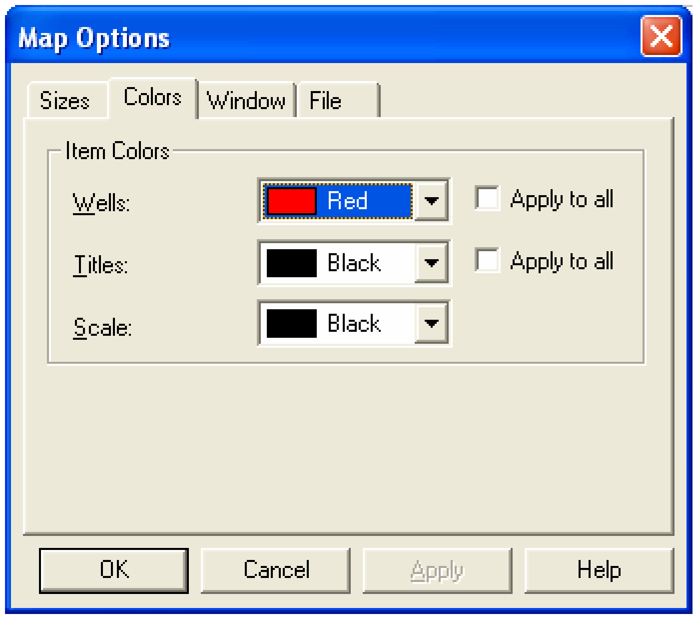


Item Sizes

Well Size	The default size for well symbols when they are created in map units
Apply to all wells	If checked, the *Well Size* will be applied to all existing well symbols
Cursor Step	The distance, in map units, the cursor is moved in the map window when an arrow key is pressed

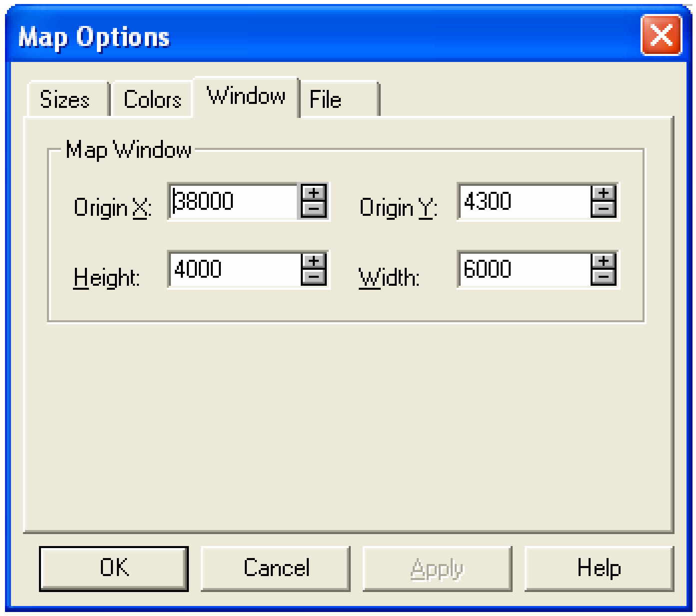
Map Text Font

Font	Defines the font, font style and size the text strings defined in the optional base map



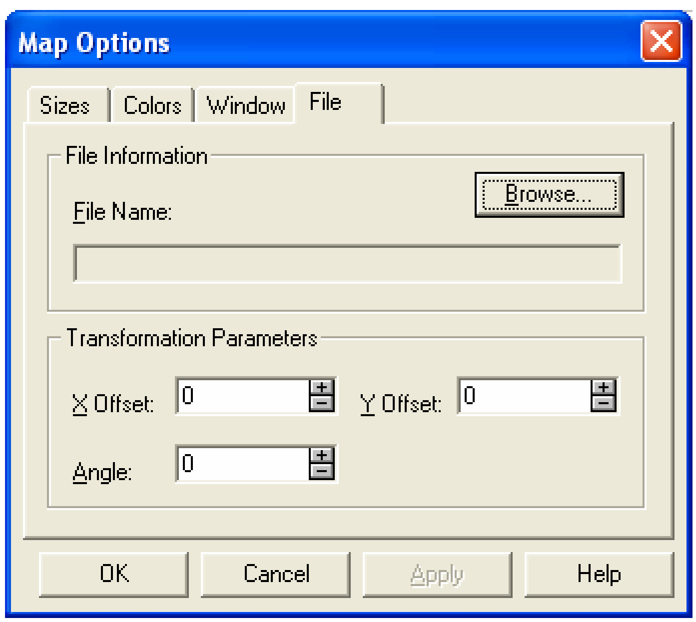
Item Colors

Wells:	The default color for well symbols when they are initially created
Apply to all wells	If checked, the *Wells* color will be applied to all existing well symbols
Titles:	The default color for titles and parameters when they are initially created on the map
Apply to all titles	If checked, the *Titles* color will be applied to all existing titles and parameters on the map
Scale:	The default color for scale bars when they are initially created on the map



Map Window

Origin X:	The x-coordinate of the origin of the map in map units
Origin Y:	The y-coordinate of the origin of the map in map units
Height:	The height of the map in map units
Width:	The width of the map in map units



File Information

File Name	The name of the base map file

NOTE: The contents of the file are loaded and stored within the document so the file is no longer required and the name is for historical reference only.

Browse | Click this button to display a standard *File Open* dialog used to locate the base map file stored on the computer.

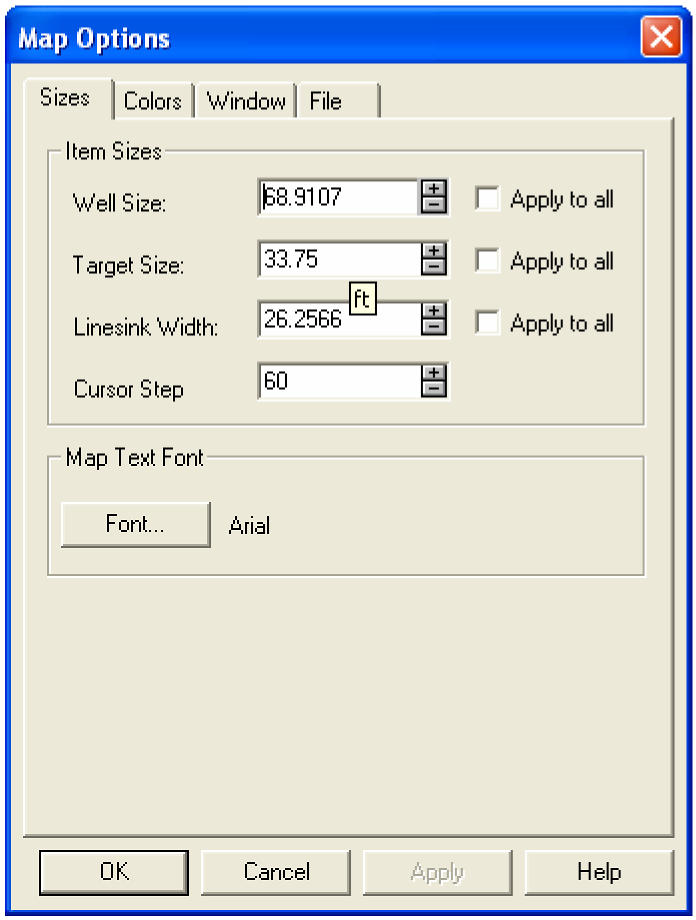
Transformation Parameters

X Offset: The value, in map units, to add to the x-coordinates of items in the base map file

Y Offset: The value, in map units, to add to the y-coordinates of items in the base map file

Angle: The angle, in degrees, to rotate the base map

The following property sheet is for map views in flow model documents.



Item Sizes

Well Size: The default size for well symbols when they are created, in map units

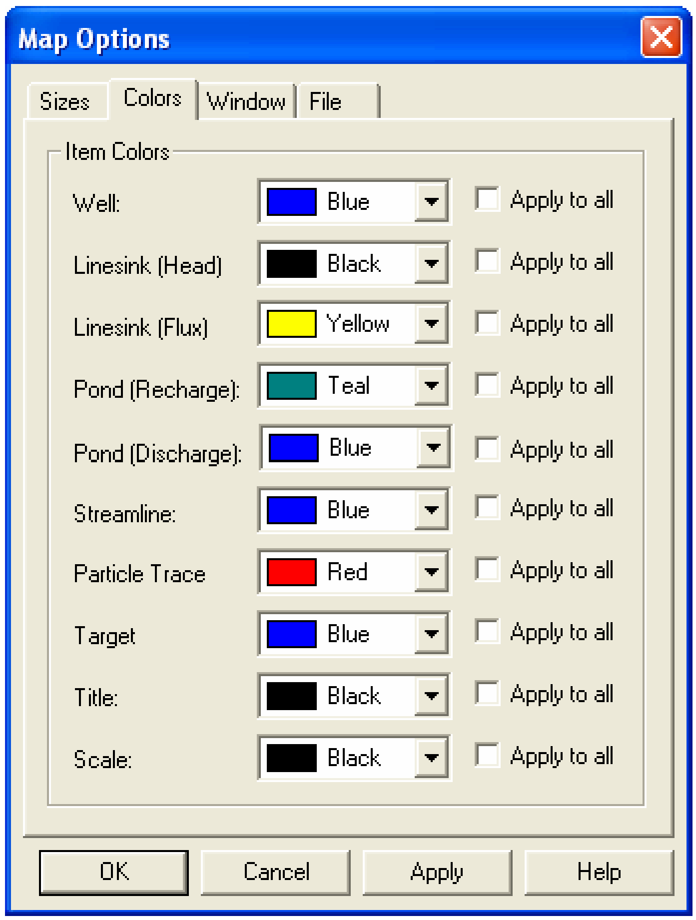
Apply to all: If checked, the *Well Size* will be applied to all existing well symbols

Target Size: The default size for target symbols when they are created, in map units

Apply to all	If checked, the *Target Size* will be applied to all existing target symbols
Linesink Width	The default width for linesinks when they are created, in map units
Apply to all	If checked, the *Linesink Width* will be applied to all existing linesinks
Cursor Step	The distance, in map units, the cursor is moved in the map window when an arrow key is pressed

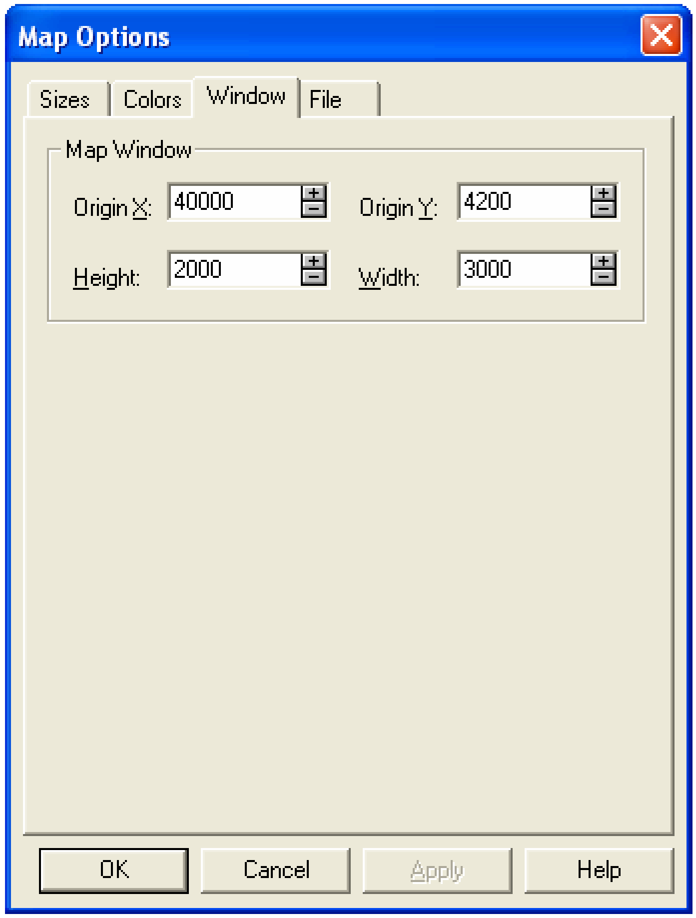
Map Text Font

Font	Defines the font, font style and size the text strings defined in the optional base map



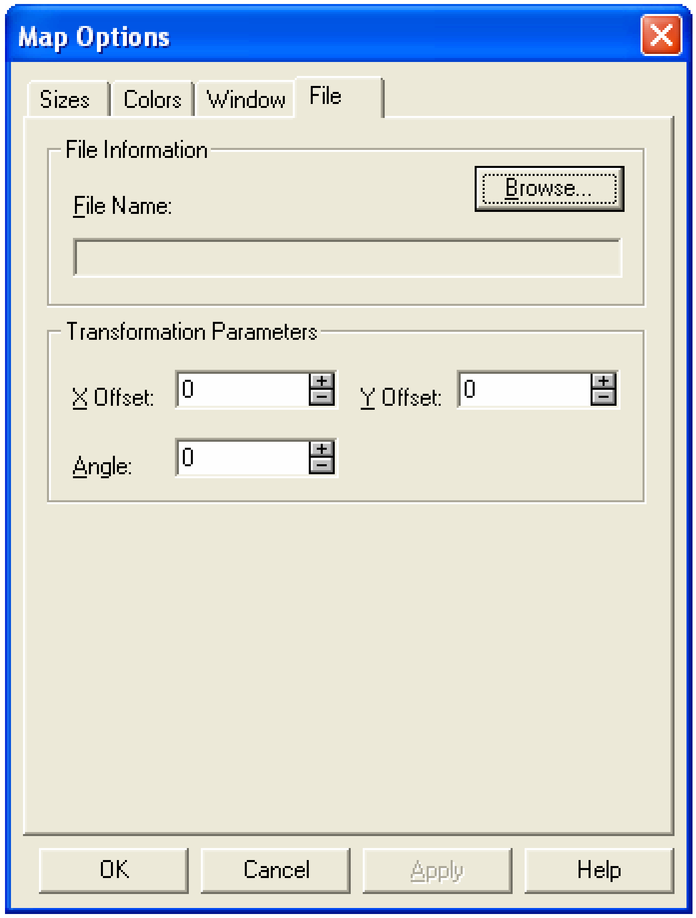
Item Colors

Wells:/Linesink (Head);/…	The default color for particular item when it is initially created
Apply to all	If checked, the corresponding item color will be applied to all existing items



Map Window

Origin X:	The x-coordinate of the origin of the map in map units
Origin Y:	The y-coordinate of the origin of the map in map units
Height:	The height of the map in map units
Width:	The width of the map in map units



File Information

File Name	The name of the base map file

NOTE: The contents of the file are loaded and stored within the document so the file is no longer required and the name is for historical reference only.

Browse	Click this button to display a standard *File Open* dialog used to locate the base map file stored on the computer.

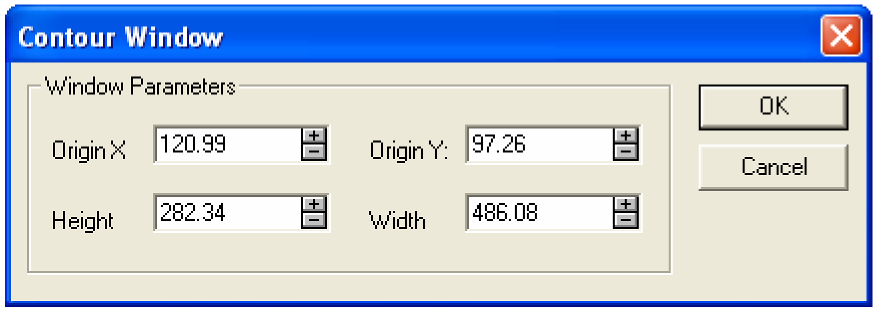
Transformation Parameters

X Offset:	The value, in map units, to add to the x-coordinates of items in the base map file
Y Offset:	The value, in map units, to add to the y-coordinates of items in the base map file
Angle:	The angle, in degrees, to rotate the base map

Use the **Locate** menu when you have imported a non-bitmap map file (.map, .dxf, .shp) and you cannot see the map. An attempt will be made to find the center of the map and estimate an appropriate scale. After the operation is complete, use the **Options/Map/Window** menu to select the part of the map you want to use.

Contour

The **Contour** menu selection contains two submenus, **Window** and **Parameters**. The **Window** option allows you to change the size and location of the contour window. The contour window by default ranges from 0 to 10000 in both the x and y axes. If a base map is imported, the size of the contour window is determined from the base map. Selecting **Options->Contour->Window** allows you to drag a rectangle on the screen to redefine the new contour window. Doing so confirms the boundaries using the *Contour Window* dialog as shown below.

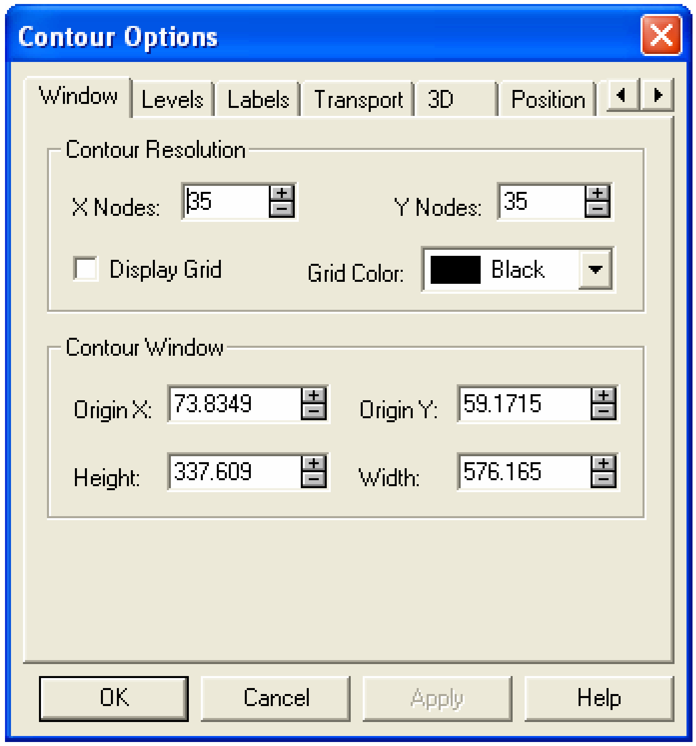


Window Parameters

Origin X	The x-coordinate of the origin of the map in map units
Origin Y	The y-coordinate of the origin of the map in map units
Height	The height of the map in map units
Width	The width of the map in map units

The **Parameters** menu selection displays the *Contour Options* property sheet described below. The contour matrix is made up of a series of equally spaced points where head is computed by Aquifer^{Win32}. You may choose the number of points in both the X and Y directions. The upper limit is 180 points in each direction. Increasing the number of nodes increases the time required to calculate the simulation and to contour the results. Although the contours will appear smoother with a larger contour matrix, there is no difference in the accuracy of the results.

The following property sheet adapts depending on the document type and not all tabs will appear in all circumstances.

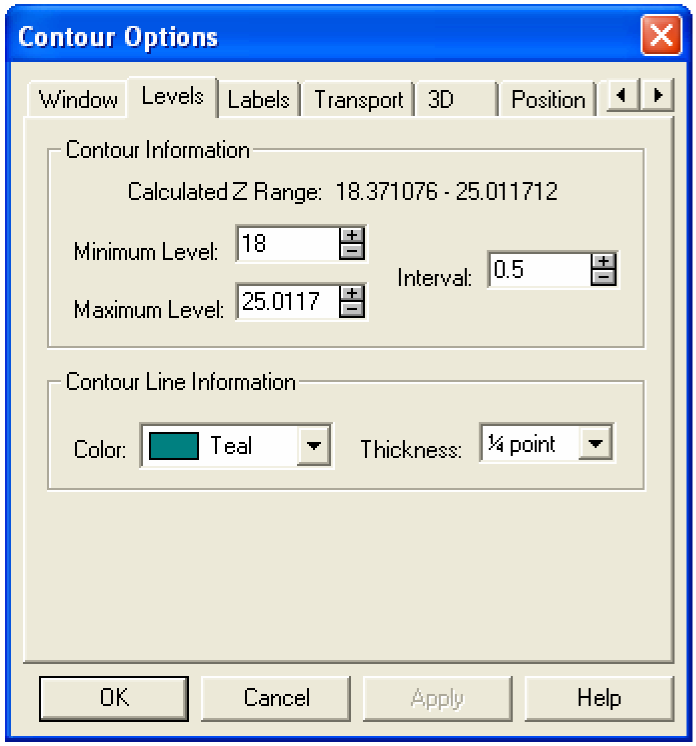


Contour Resolution

X Nodes:	The number of equal subdivisions along the x-axis of the contour window to calculate data values for contouring
Y Nodes:	The number of equal subdivisions along the y-axis of the contour window to calculate data values for contouring
Display	If checked, the contour grid is displayed on the map window
Grid Color:	The color to use when displaying the contour grid

Contour Window

Origin X	The x-coordinate of the origin of the contour window in map units
Origin Y	The y-coordinate of the origin of the contour window in map units
Height	The height of the contour window in map units
Width	The width of the contour window in map units

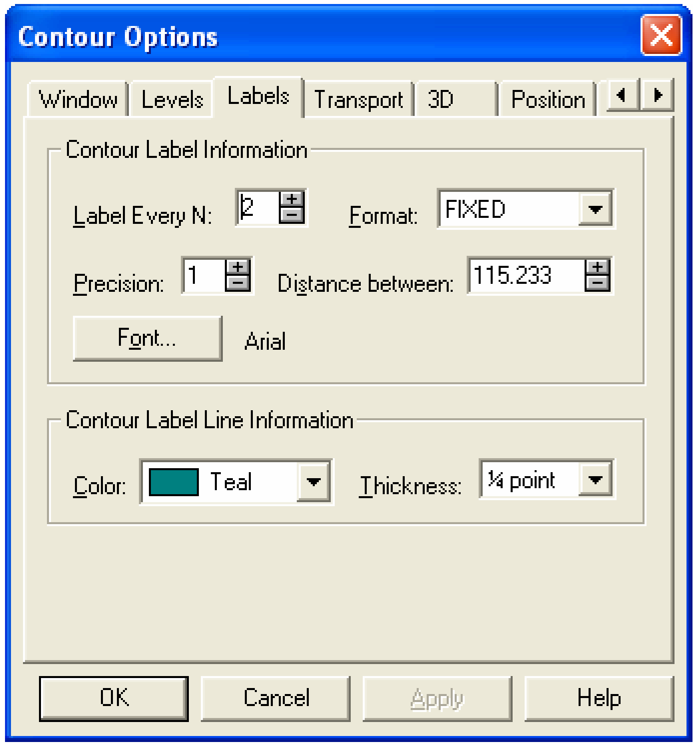


Contour Information

Calculated Z Range:	Displays the calculated minimum and maximum head/drawdown values over the contour window
Minimum Level:	Defines the minimum contour level to display
Maximum Level:	Defines the maximum contour level to display
Interval:	Defines the head/drawdown interval between adjacent contour lines

Contour Line Information

Color:	The color to use when displaying non-labeled contour lines
Thickness:	The thickness, in points, of the line used when displaying non-labeled contour lines

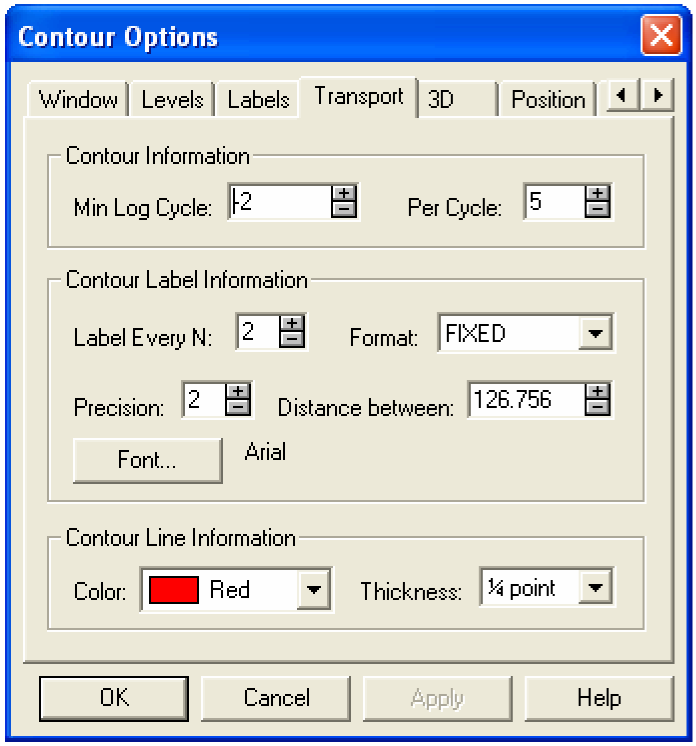


Contour Label Information

Label Every N:	The number of non-labeled contour lines between labeled contour lines is one less than this value
Format:	The numeric format to use when displaying the label on labeled contour lines
Precision:	The number of digits to the right of the decimal point to use when displaying the label on labeled contour lines
Distance between:	The linear distance, in map units, between adjacent labels on a labeled contour line
Font	Defines the font, font style and size for contour label

Contour Label Line Information

Color:	The color to use when displaying labeled contour lines
Thickness:	The thickness, in points, of the line used when displaying labeled contour lines



Contour Label Information

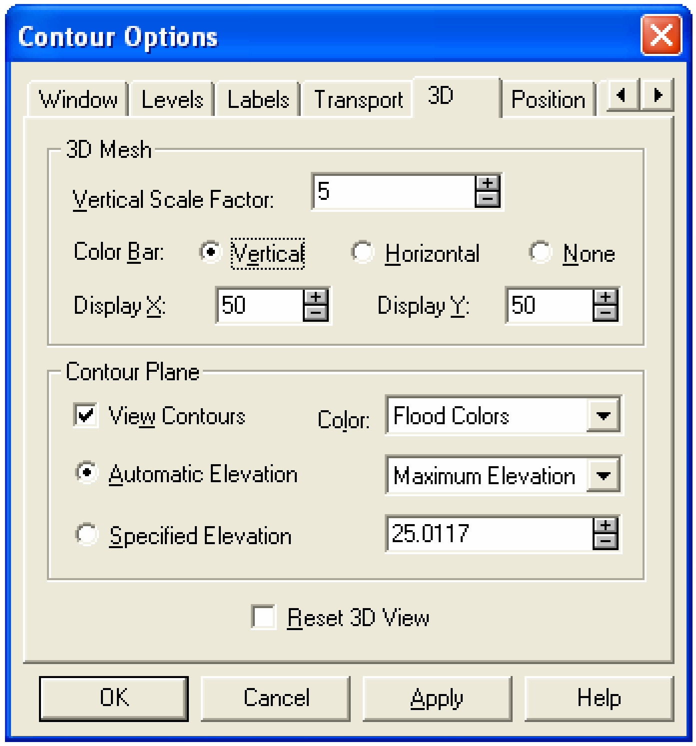
Min Log Cycle:	The minimum power of 10 used to when generating contours of concentration in a transport model
Per Cycle:	The number of contours displayed within each log cycle

Contour Label Information

Label Every N:	The number of non-labeled contour lines between labeled contour lines is one less than this value
Format:	The numeric format to use when displaying the label on labeled contour lines
Precision:	The number of digits to the right of the decimal point to use when displaying the label on labeled contour lines
Distance between:	The linear distance, in map units, between adjacent labels on a labeled contour line
Font	Defines the font, font style and size for contour label

Contour Label Line Information

Color:	The color to use when displaying labeled contour lines
Thickness:	The thickness, in points, of the line used when displaying labeled contour lines

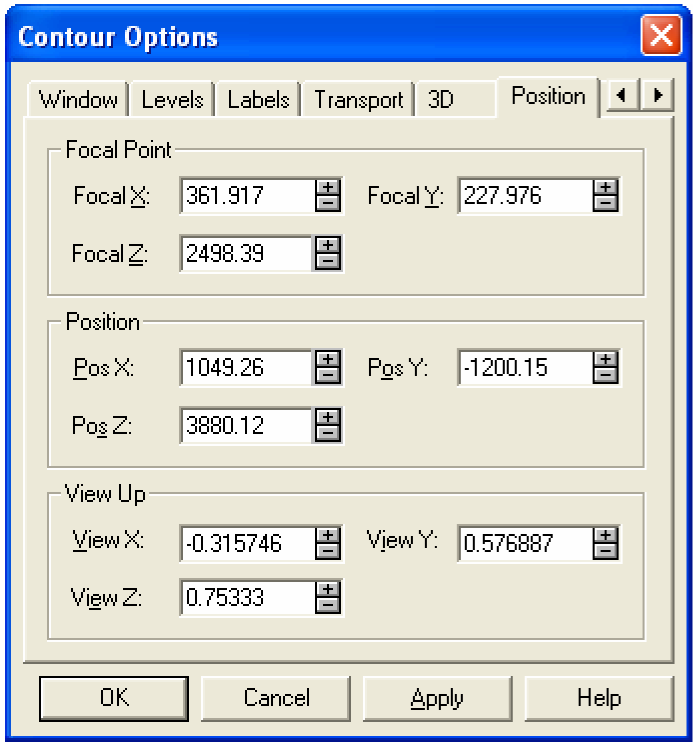


3D Mesh

Vertical Scale Factor:	A multiplier applied to head/drawdown to increase the vertical scale relative to the horizontal scale
Color Bar:	Controls the presence and location of the default scale bar
Display X:	The x-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar
Display Y:	The y-coordinate of the lower left corner (vertical) or upper right corner (horizontal) of the scale bar

Contour Plane

View Contours:	When checked, a contour map is displayed in the 3D perspective view
Color:	Controls the color of the contours which can be either Black or Flood Colors
Automatic Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked the contour map will automatically be relocated as the contour data changes
Specified Elevation:	Controls the elevation on the vertical axis which corresponds to the contour plane; when checked, the value entered in the adjacent edit field will be used to locate the contour plane
Reset 3D View	At times, the perspective drawing can leave the field of view. Checking this option will relocate it into view



Focal Point

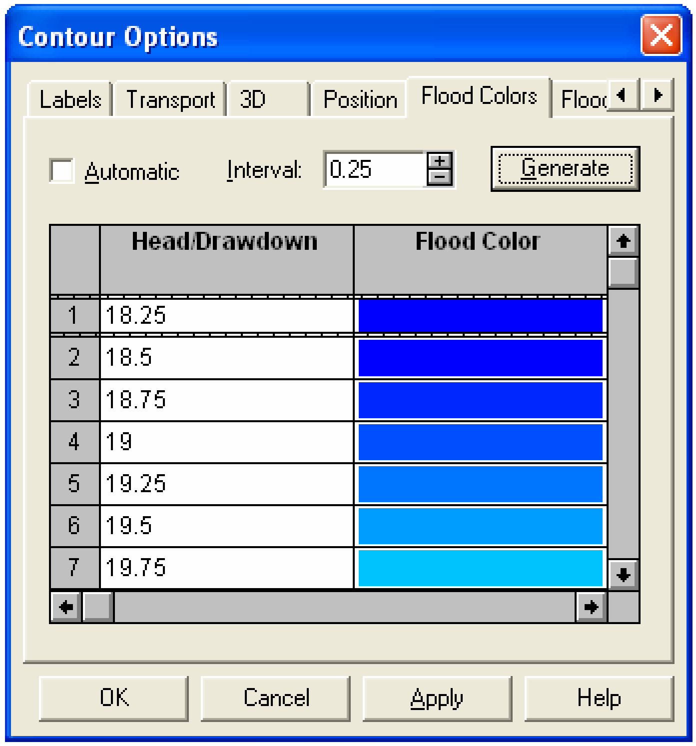
Focal X:	The x-coordinate of the focal point used to display the 3D perspective drawing
Focal Y:	The y-coordinate of the focal point used to display the 3D perspective drawing
Focal Z:	The z-coordinate of the focal point used to display the 3D perspective drawing

Position

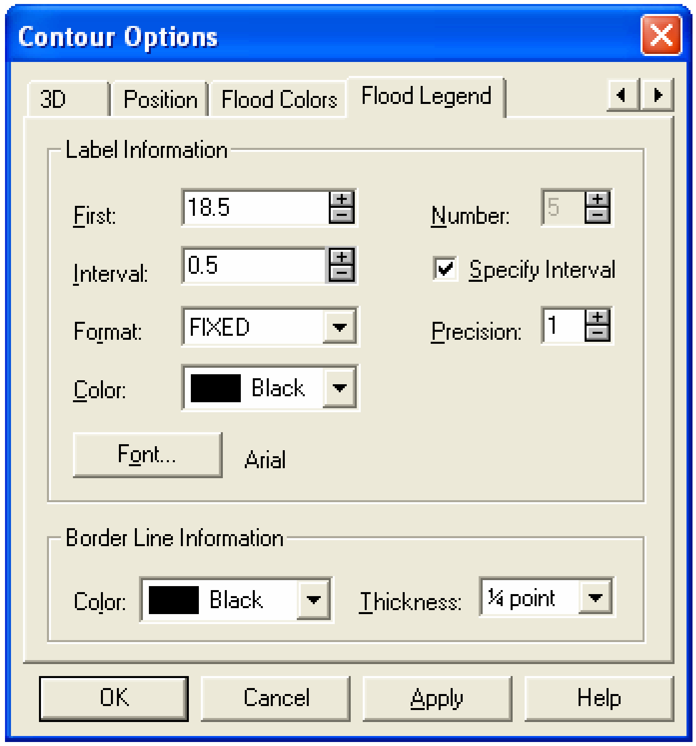
Position X:	The x-coordinate of the camera position used to display the 3D perspective drawing
Position Y:	The y-coordinate of the camera position used to display the 3D perspective drawing
Position Z:	The z-coordinate of the camera position used to display the 3D perspective drawing

View Up

View X:	The x-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Y:	The y-coordinate defining the view up direction for the camera used to display the 3D perspective drawing
View Z:	The z-coordinate defining the view up direction for the camera used to display the 3D perspective drawing



Automatic	When checked, the distribution of colors is automatically determined by the application
Interval:	The interval to use when generating flood colors for values of head/drawdown to be used when the **Generate** button is clicked.
Generate	When clicked, a new color distribution will be generated using the same algorithm as the **Automatic** setting; however, these can be edited by the user



Label Information

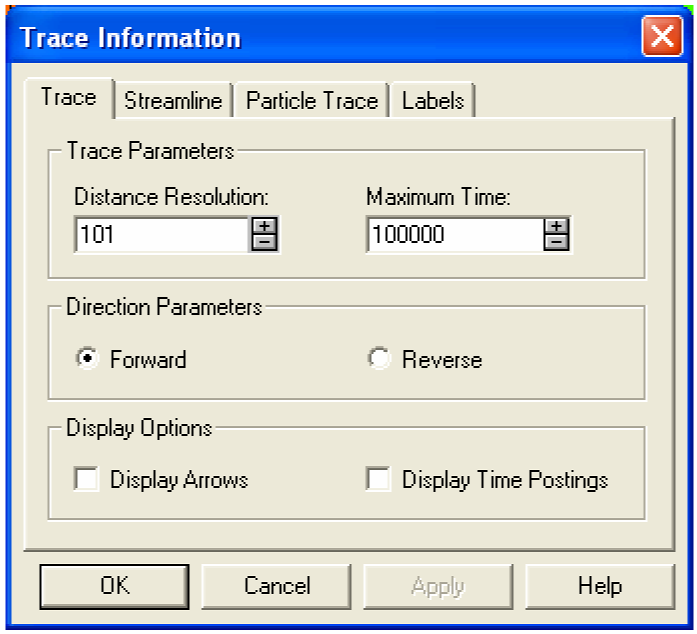
First	Sets the head/drawdown value for the first labeled value in color flood legends
Number:	The number of equally spaced labels to use when labeling the color flood legends
Interval:	The interval between adjacent labels; used when the Specify Interval option is activated
Specify Interval	When checked, the value for *Interval* is used when labeling color flood legends
Format:	The numeric format to use when displaying the labels on color flood legends
Precision:	The number of digits to the right of the decimal point to use when displaying the label on color flood legends
Color	The color to use when displaying labels on color flood legends
Font	Defines the font, font style and size for the label

Border Line Information

Color:	The color to use when displaying the border around the color flood legend
Thickness:	The thickness, in points, of the line used when displaying the border around the color flood legend

Trace

The **Trace** menu displays the *Trace Information* property sheet which is used to define the display and calculation characteristics of streamlines and particle traces within the model. There are two versions of the ***Trace*** tab, one for Steady State models and one for Transient models.



Trace Parameters

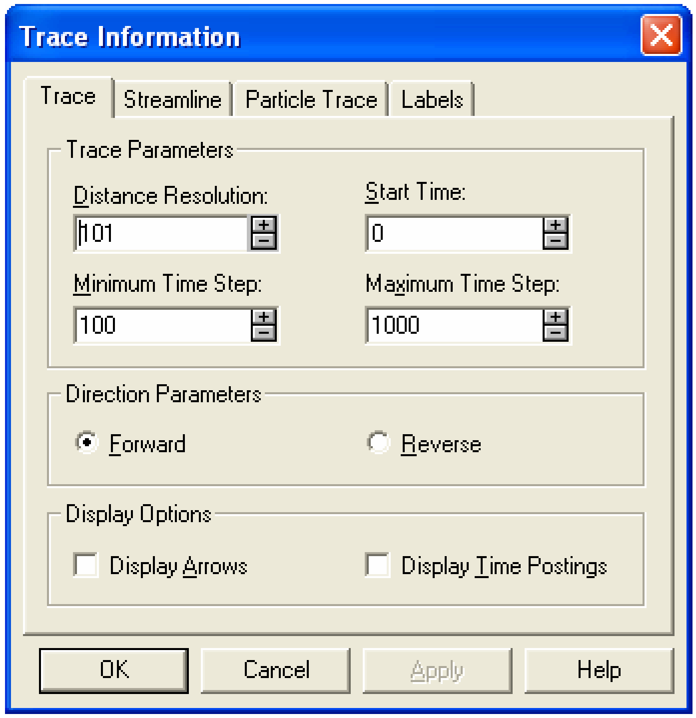
Distance Resolution:	The distance resolution determines the accuracy of the particle-tracking calculation, which is done numerically; a higher number will produce better results
Maximum Time:	The time to which particle traces are calculated; beyond this time particle traces will be truncated.

Direction Parameters

Forward	Particles are tracked from the location of the particle trace element in a down gradient direction
Reverse	Particles are tracked from the location of the particle trace element in an upgradient direction

Display Options

Display Arrows	When checked, arrows are displayed on streamlines according to the parameters on the ***Streamline*** tab or on particle traces according to the parameters on the ***Particle Trace*** tab
Display Time Postings	When checked, time postings are displayed on the particle traces according to the parameters on the ***Labels*** tab



Trace Parameters

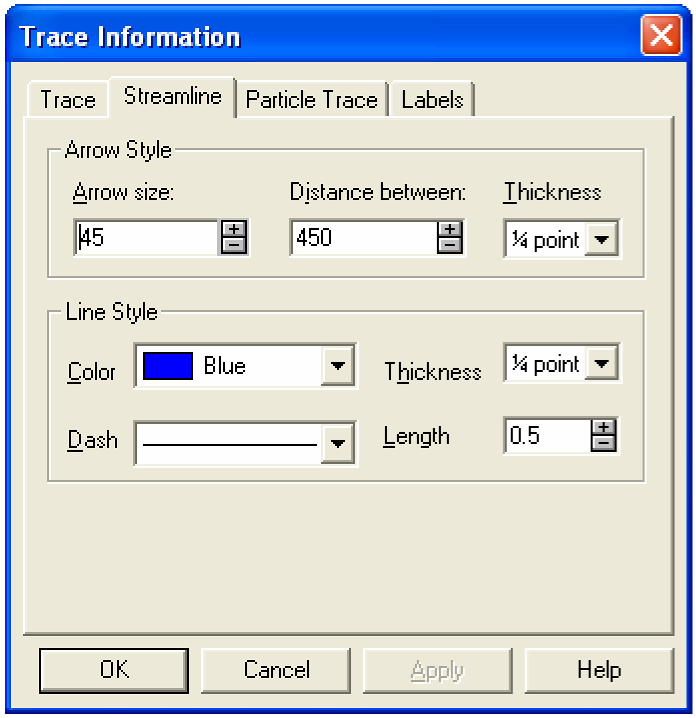
Distance Resolution:	The distance resolution determines the accuracy of the particle-tracking calculation, which is done numerically; a higher number will produce better results
Start Time:	The model time at which particle traces should begin to be calculated
Minimum Time Step:	Defines the minimum time period to use between adjacent particle movements
Maximum Time Step:	Defines the maximum time period to use between adjacent particle movements

Direction Parameters

Forward	Particles are tracked from the location of the particle trace element in a down gradient direction
Reverse	Particles are tracked from the location of the particle trace element in an upgradient direction

Display Options

Display Arrows	When checked, arrows are displayed on streamlines according to the parameters on the ***Streamline*** tab or on particle traces according to the parameters on the ***Particle Trace*** tab
Display Time Postings	When checked, time postings are displayed on the particle traces according to the parameters on the ***Labels*** tab

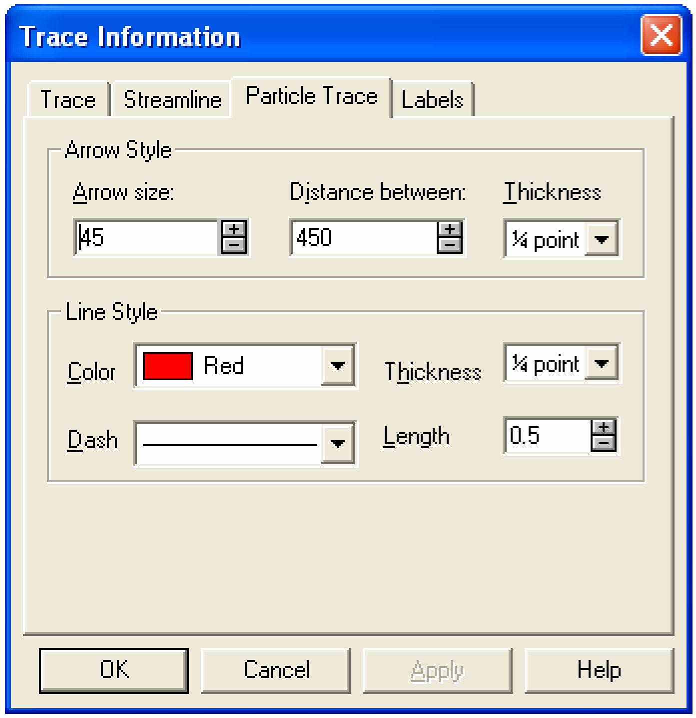


Arrow Style

Arrow size:	The size of the arrows displayed on streamlines defined in map units
Distance between:	The distance between adjacent arrows displayed on streamlines defined in map units
Thickness:	The thickness of the line used to draw arrows defined in points

Line Style

Color:	The color used to display the streamline
Thickness:	The thickness in points of the streamline
Dash:	The dash pattern to use for the streamline
Length:	The length in inches of the dash pattern for the streamline

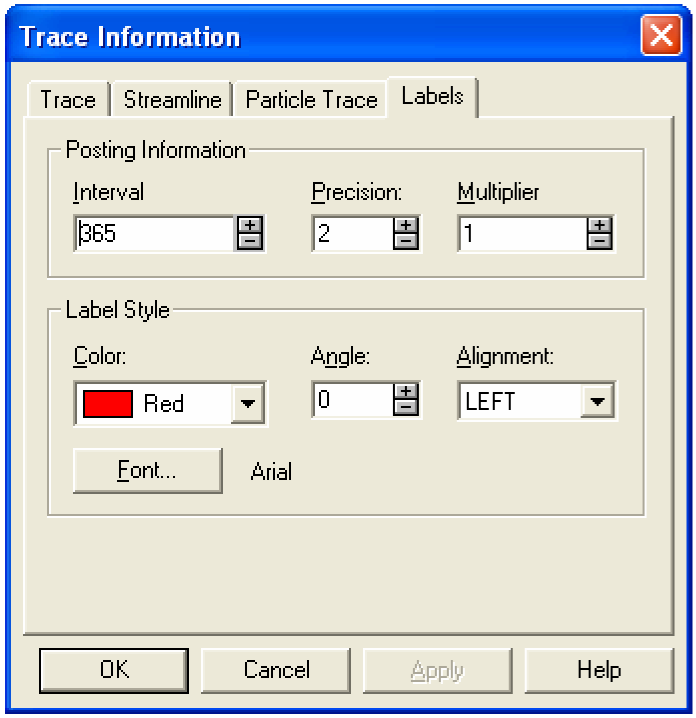


Arrow Style

Arrow size:	The size of the arrows displayed on particle traces defined in map units
Distance between:	The distance between adjacent arrows displayed on particle traces defined in map units
Thickness:	The thickness of the line used to draw arrows defined in points

Line Style

Color:	The color used to display the particle trace
Thickness:	The thickness in points of the particle trace
Dash:	The dash pattern to use for the particle trace
Length:	The length in inches of the dash pattern for the particle trace



Posting Information

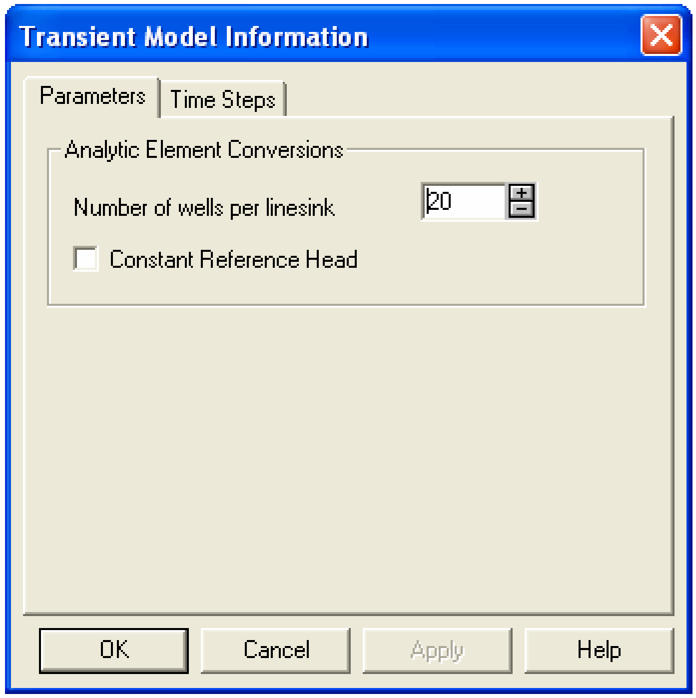
Interval	The time interval between the adjacent time labels
Precision	The number of decimal places after the decimal point to use when displaying time labels
Multiplier	The multiplier to apply to the time interval between adjacent time labels

Label Style

Color	The color to use when displaying the time label
Angle	The angle to rotate the time label around the location point
Alignment	The alignment of the time label relative to the location point
Font	Defines the font, font style, size, and effects for the time label

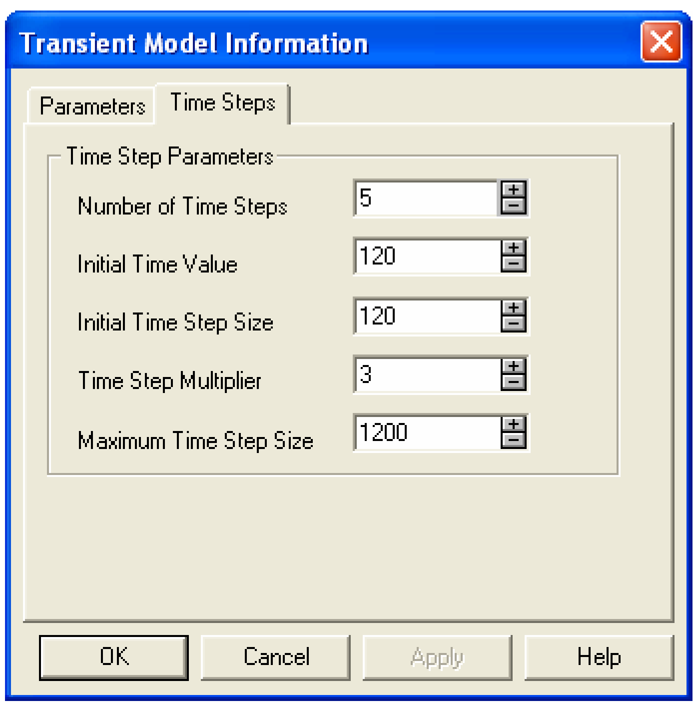
Transient

The **Transient** menu displays the *Transient Model Information* property sheet which is used to define the time steps to calculate for a transient model or a transport model.



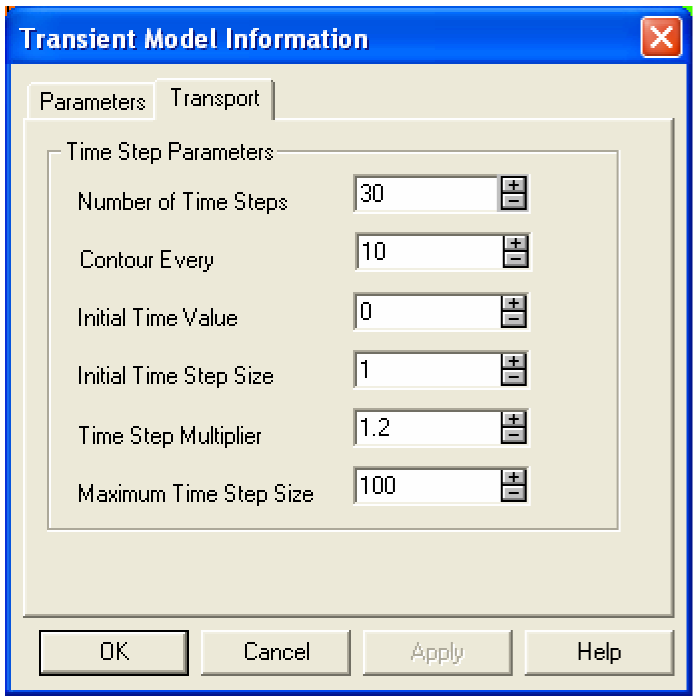
Analytic Element Conversions

Number of well per linesink	Specifies the number of equally spaced wells to use when representing a flux linesink in a transient model
Constant Reference Head	When checked, the reference head is held constant during a transient simulation



Time Step Parameters

Number of Time Steps	Specifies the number of time steps to calculate; each time step is represented by a separate tab in the tab view
Initial Time Value	Specifies the first time at which to calculate hydraulic head or drawdown
Initial Time Step Size	Specifies the time increment between the *Initial Time Value* and the second time step
Time Step Multiplier	After the first time step, the time step size is multiplied by this value to calculate a new time step size
Maximum Time Step Size	Specifies the maximum difference in time to use between adjacent time steps



Time Step Parameters

Number of Time Steps	Specifies the number of time steps to calculate
Contour Every	Specifies the number of time steps to skip between contour maps each of which are represented by a separate tab in the tab view
Initial Time Value	Specifies the first time at which to calculate concentrations
Initial Time Step Size	Specifies the time increment between the *Initial Time Value* and the second time step
Time Step Multiplier	After the first time step, the time step size is multiplied by this value to calculate a new time step size
Maximum Time Step Size	Specifies the maximum difference in time to use between adjacent time steps

Recalculate Traces

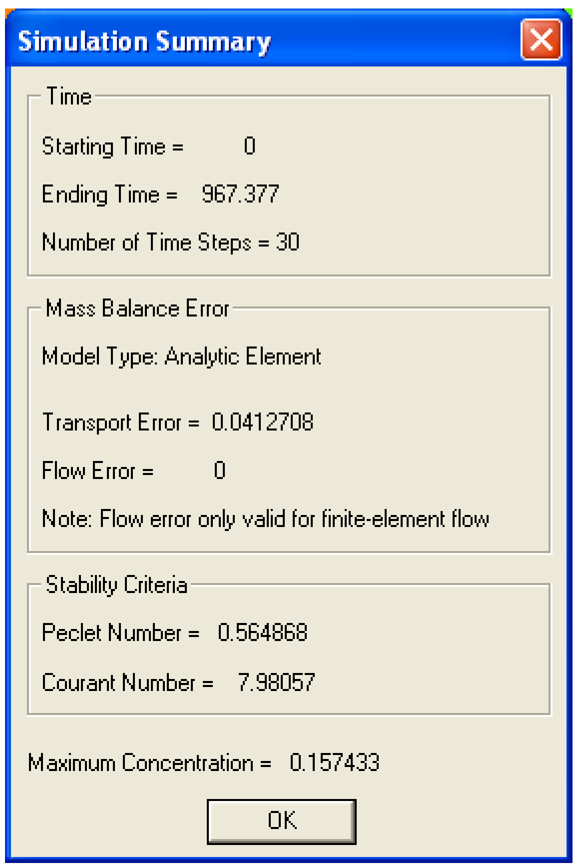
The **Recalculate Traces** menu controls whether streamlines and particle traces are automatically recalculated. If this menu is checked, newly added or edited streamlines and particle traces will be instantly calculated. When making numerous changes to a model, it is sometimes advisable to turn off automatic calculation.

Constant Reference Head

The **Constant Reference Heads** menu controls whether the reference head is held constant during a transient simulation. If this menu is checked and the currently active model supports this feature, the reference head will be held constant (as in a steady-state model). If not checked the reference head may change based upon drawdown computed at the reference head location. It is recommended that you do **NOT** keep the reference head constant when you are computing drawdown. Hold the reference head constant if you are comparing the transient model to results of a steady-state model.

Simulation Summary

The **Simulation Summary** menu displays a summary of the WinTran model.



Zero Concentrations

The **Zero Concentrations** menu resets initial concentration values to 0.

Restart

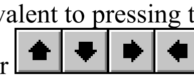
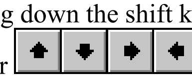
The **Restart** menu sets the initial concentration in the WinTran model to be the concentrations calculated at the last time step of the current model simulation.

Snap Wells to Contour Grid

The **Snap Wells to Contour Grid** menu controls whether the location of wells is automatically adjusted to coincide with the closest contour grid node. Remember that the finite-element mesh used for transport is identical to the contour matrix used to contour heads. If you enter a well at a location other than a node (intersection of contour grid lines), the velocities computed by WinTran around the well will not be accurate. The inaccurate velocities may produce undesirable mass balance errors. It is recommended that you use this option to ensure proper mass balance in the transport model.

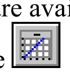
Calc Menu

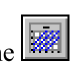
Coarse & Fine

The **Coarse** and **Fine** menus contain the four directions to move the data over the type curve (**Up**, **Down**, **Left**, **Right**). Coarse is equivalent to pressing the arrows on the keyboard or the large arrow buttons on the toolbar . Fine is equivalent to pressing the arrows on the keyboard while holding down the shift key. Fine is also equivalent to the small arrow buttons on the toolbar .

Linear Regression

NOTE: This menu is only active for step test analysis.

Two submenus are available on the **Linear Regression** menu. The first is the **Single Step** menu or the  button on the Step toolbar. The **Single Step** menu calculates a linear regression line based on the active step. Since this analysis requires that the slope of all regression lines for all steps be equal, the remaining steps are calculated by specifying the slope and then finding the intercept of the best-fit line. The step is chosen either by using the toolbar combobox in the Step toolbar, or using the **Edit->Toggle Step** menu.

The second is the **Multiple Steps** menu or the  button on the Step toolbar. The **Multiple Step** menu performs a linear regression on all steps combined to find the best slope across all steps and the individual intercepts for each step.

Optimize

The **Optimize** menu or the  button on either the Match toolbar or the Step toolbar uses the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to find the best statistical match for the data from the active data set. In a multiple well analysis, the active well is optimized.

Optimize Group

Select the **Optimize Group** menu uses the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to find the best statistical match using the data from all wells defined in the analysis. The parameters that are optimized across all wells are defined by the **Edit->Group Optimize Parameters** menu. If no parameters have be defined to optimize across all wells, the **Optimize Group** menu is equivalent to using the **Optimize** menu for each well individually.

Reset Data Offset

The **Reset Data Offset** menu will set the offset of the match data to 0 which synchronizes the origins of the data graph with the type curve graph it is superimposed upon. Occasionally, the field data can get shifted so as not to be visible on the graph; this condition can occur when the units of the data are changed without converting the data values or if the optimization procedure fails to converge.

Recalculate Type Curves

Select the **Recalculate Type Curves** menu to recalculate all type curves based on the prevailing parameters. At times the type curves become out of sync with the current parameters and must be recomputed. The rightmost field on the status bar indicates whether the displayed type curves are valid based on the prevailing parameters.

Match Early Data

Select the **Match Early Data** menu to set up the match data graph to display and/or adjust the match of early data in an unconfined analysis. In such an analysis, the early match and the late match are separate and changing between Match Early Data and Match Late Data may cause the data to shift relative to the type curves.

Match Late Data

Select the **Match Late Data** menu to set up the match data graph to display and/or adjust the match results of late data in an unconfined analysis. In such an analysis, the early match and the late match are separate and changing between Match Early Data and Match Late Data may cause the data to shift relative to the type curves.

Recalculate

The **Recalculate** menu or the  button of the Standard toolbar recalculates the predicted drawdown matrix in a Simulation Document. Whenever you change a parameter that invalidates the simulation, the word "Recalculate" appears in the status bar indicating that the contour map and/or predicted drawdown versus time graphs are not representative of the current simulation parameters.

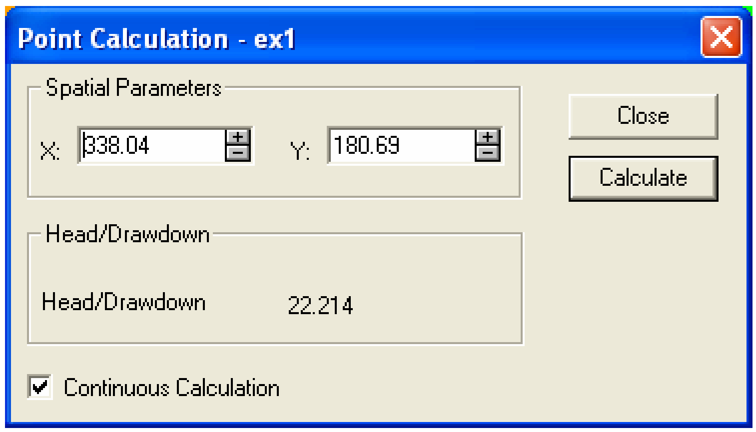
Recontour

The **Recontour** menu or the  button on the Standard toolbar recontours the predicted drawdown matrix in a Simulation Document. Whenever you change a parameter that changes the display characteristics of the contour lines the word

"Recalculate" appears in the status bar indicating that the contour map and/or predicted drawdown versus time graphs are not representative of the current simulation parameters. The **Recontour** menu will redisplay the contours without recalculating the predicted drawdown matrix.

Point Calc

Select the **Point Calc** menu to quickly compute values of drawdown at particular points based on map coordinates. Choosing this menu displays the *Point Calculation* dialog on the screen. You have the option to compute drawdown continuously as you move the mouse or noncontinuously by entering specific coordinates. In continuous mode, you move the mouse around the map and the drawdown is updated in the dialog. In noncontinuous mode, you enter the specific coordinates in the dialog, press the **Calculate** button, and the new drawdown is computed and displayed. The *Point Calculation* dialog is modeless and linked to a specific simulation document. As such, it remains visible until you close the document or press the **Close** button.



Spatial Parameters

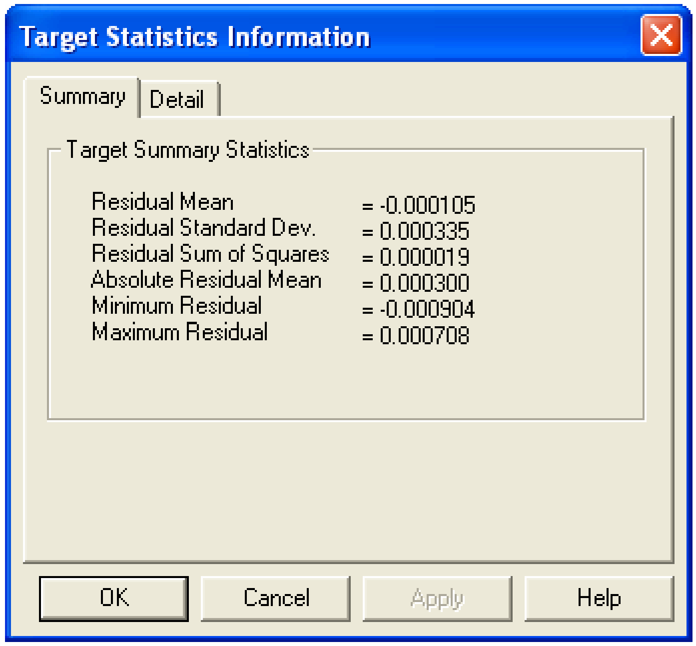
X:	The x-coordinate of the point to calculate
Y:	The y-coordinate of the point to calculate
Head/Drawdown	
Head/Drawdown	The calculated value of head/drawdown at the point
Continuous Calculation	If checked, the value will be calculated every time the mouse cursor moves over the map view
Close	Close the dialog
Calculate	Calculates the value of head/drawdown at the point specified in X and Y

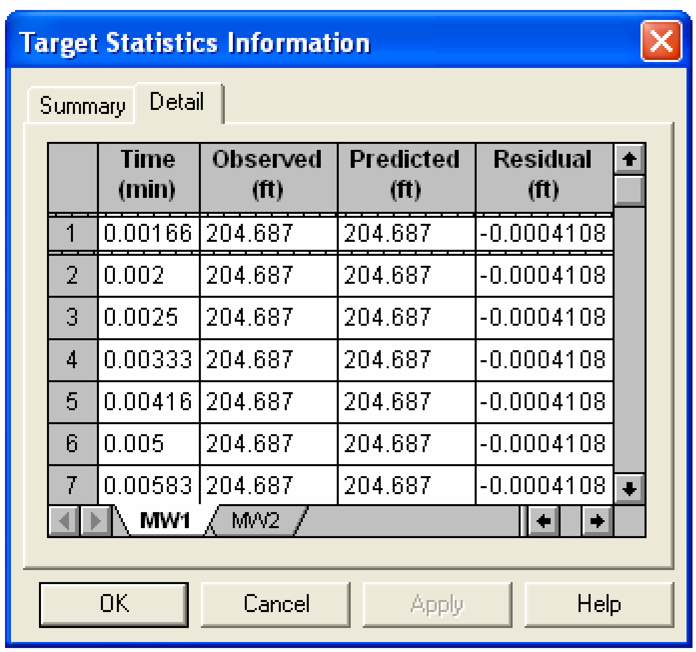
Match Data

Select the **Match Data** menu when the spreadsheet view is active to recalculate the data points for a multiple well analysis. This would typically be done after changes had been made to the time/drawdown data for individual wells in the spreadsheet view. If the changes are made on a property sheet, the calculation is done automatically.

Target Statistics

The **Target Statistics** menu displays the *Target Statistics Information* property sheet which is used to display the statistical differences between the observed target heads or drawdowns and those calculated by the model. The ***Summary*** tab presents the statistics across all target values. The ***Detail*** tab each individual target value and the corresponding calculated value.





Drawdown

The **Drawdown** menu toggles between the calculation of hydraulic head and drawdown. If the menu is checked, drawdown is calculated and the reference head is

ignored. If drawdown is being calculated, the target head values are assumed to represent drawdown as well.

Optimize Model

The **Optimize Model** menu uses the Marquardt (modified Gauss-Newton) nonlinear least-squares technique to find the best statistical match for the target head data defined in the model.

Derivative Analysis

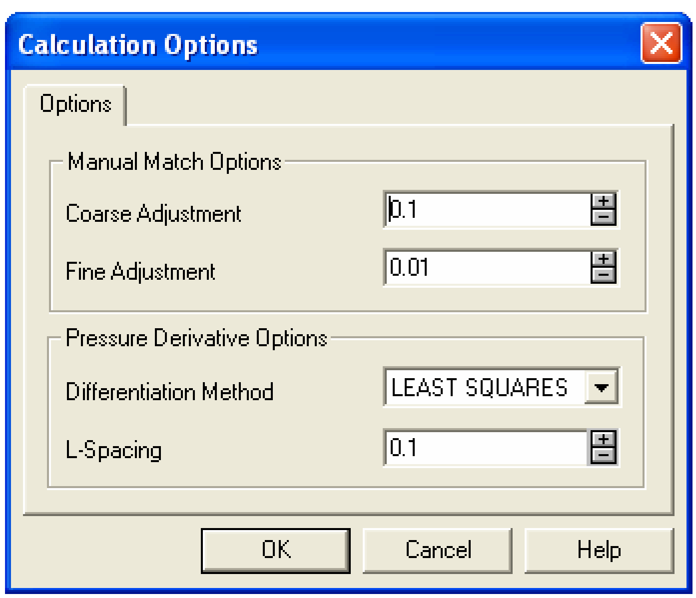
The **Derivative Analysis** menu toggles between the manual and automatic curve matching of first-order derivatives. If the menu is checked, derivative type curves are calculated and the derivative of the time/drawdown data is displayed. Likewise, the ***Predicted*** view tab displays the derivative of the observed time/drawdown data and the predicted derivative curve is calculated and displayed. Refer to the Derivative Analysis section of the Concepts Chapter for more details.

Derivative Optimization

The **Derivative Optimization** menu toggles between the optimizing of predicted drawdown and the optimizing of the first-order derivative of drawdown. If the menu is checked, the derivative of the time/drawdown data is optimized; otherwise, the drawdown itself is optimized. If the menu is grayed, the selected analysis does not support optimization of derivatives. Refer to the Derivative Analysis section of the Concepts Chapter for more details.

Options

The **Options** menu displays the *Calculation Options* property sheet used to control the differentiation method and l-spacing used during a derivative analysis. It is also used to specify the data offsets used in response to coarse and fine manual curve match adjustments.



Manual Match Options

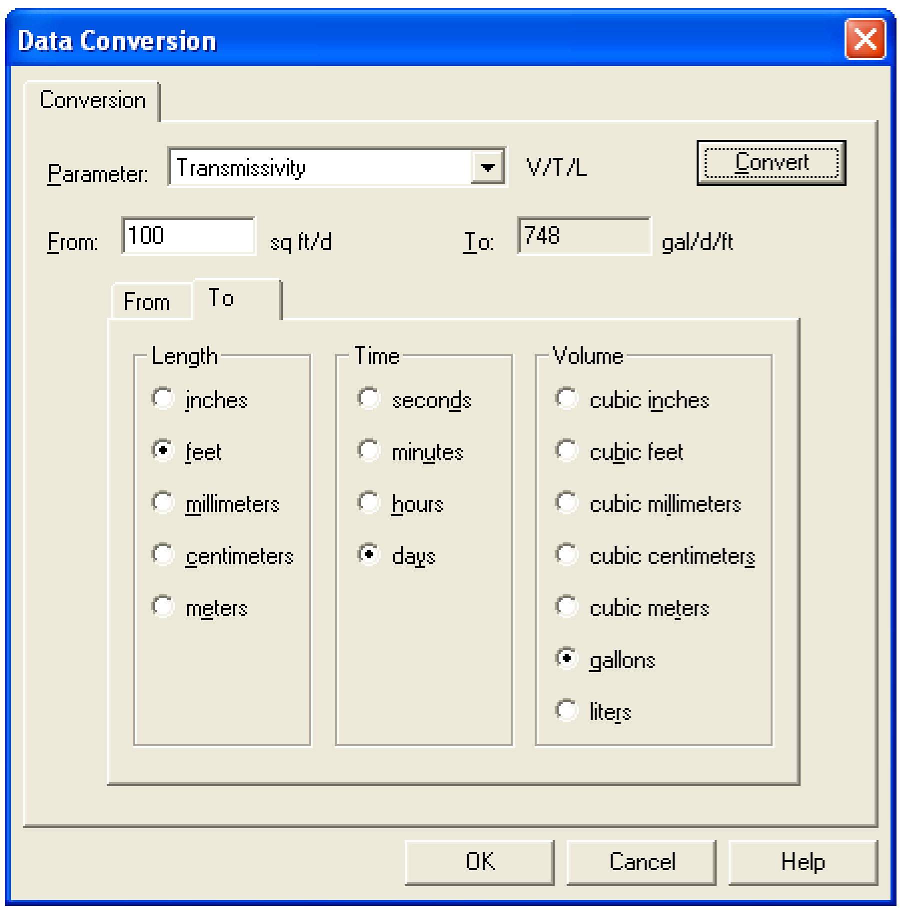
Coarse Adjustment	The distance in inches to move the data relative to the type curves when performing a coarse adjustment
Fine Adjustment	The distance in inches to move the data relative to the type curves when performing a fine adjustment

Pressure Derivative Options

Differentiation Method	Specifies the calculation method to use when calculating derivatives
LEAST SQUARES	See discussion above
FIXED ENDPOINT	See discussion above
L-Spacing	Specifies the interval over which data and type curve derivatives are calculated

Data Conversion

Select the **Data Conversion** menu to display the *Data Conversion* property sheet for converting data from one set of units to another. This does not effect any parameter or calculation in the program but is simply provided for your use in data conversions.



Parameter	Select the parameter you are converting values for
Convert	Activate the conversion
From	The value you want to convert in the adjacent units
To	The converted value in the adjacent units

To/From Tabs	
Length	Units of length corresponding to the active tab
Time	Units of time corresponding to the active tab
Volume	Units of volume corresponding to the active tab

Model Menu

The **Model** menu contains three submenus, **Steady-state, Transient** and **Transport**. A checkmark appears next to the currently selected menu. If the menu item is grayed, the currently selected model does not support this feature. You may change the current model type by clicking on either **Steady-State, Transient** or **Transport**. Only one checkmark can appear on this menu. For example, if the checkmark is next to the **Steady-state** menu and you click the **Transient** menu, the checkmark next to the **Steady-state** menu will disappear.

Steady-state models assume that all analytic elements (e.g. wells, linesinks, and ponds) function at the same rate forever. The resulting water-level contour map represents a long-term average condition, called steady-state. The transient model computes water levels over time based on the time steps defined in the *Transient Model Information* property sheet on the ***Time Steps*** tab. The transport model time steps are defined on the ***Transport*** tab.

Window Menu

Cascade

The **Cascade** menu arranges windows in an overlapped fashion. The windows are overlapped such that the corners of the windows align diagonally from the upper left corner of the frame window.

Tile Horizontally

The **Tile Horizontally** menu arranges windows in non-overlapped tiles of equal size. If possible the windows will be arranged horizontally.

Tile Vertically

The **Tile Vertically** menu arranges windows in non-overlapped tiles of equal size. If possible the windows will be arranged vertically.

Arrange Icons

The **Arrange Icons** menu arranges the icons of closed windows along the bottom of the frame window.

Split

The **Split** menu activates the separator between the panes of the active document window so it can be moved with the keyboard arrow keys.

Window 1, 2…

This menu represents a currently open document window. The menu will activate the window and bring it to the foreground of the frame window.

Help Menu

Help Topics

The **Help Topics** menu displays the online help for Aquifer^{Win32}. The online help contains this entire manual with links established between the application and pertinent chapters and/or subchapters.

Solution Help

The **Solution Help** menu displays the online help for the selected analytical solution or analytical model. If the menu is grayed, the solution does not have a help file associated with it. Typically the help file associated with particular analytical solution will contain details about the solution, its parameters and how to apply them.

What's This?

The **What's This?** menu allows identification of another menu, toolbar button, etc. After selecting this menu, navigate the menus to locate the item you would like help on and click on it.

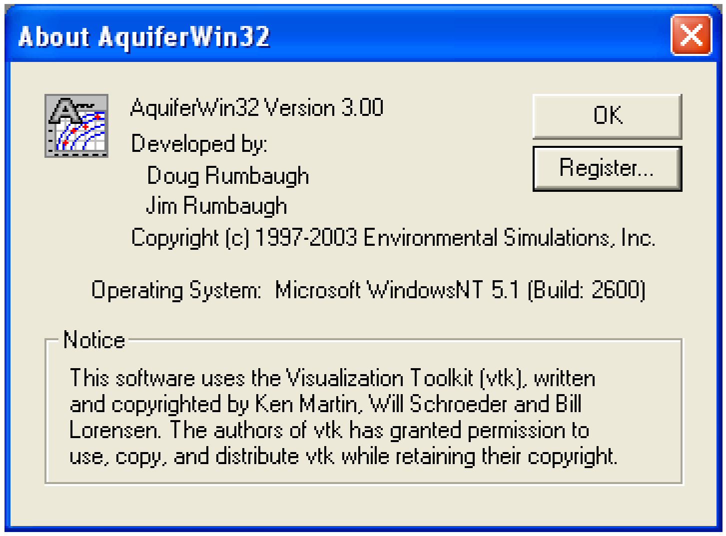
Tip of the Day

The **Tip of the Day** menu displays the *Tip of the Day* dialog used to display suggestions or help about functionality in Aquifer^{Win32}.

Show Tips on StartUp	If checked, this dialog is displayed when the program is started
Next Tip	Advances to the next suggestion or help item
Close	Closes the dialog

About AquiferWin32

The **About AquiferWin32** displays the *About AquiferWin32* dialog as below. This dialog presents the program version number, credits and identifies the operating system the application is running on.

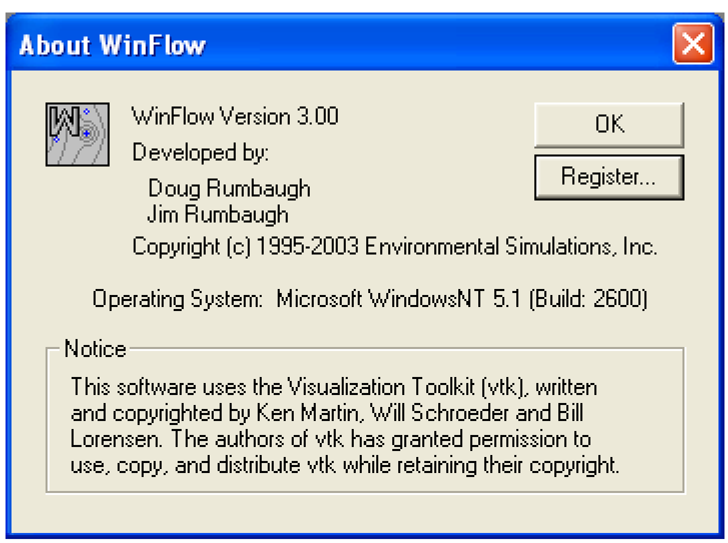


The **Register** button, when clicked, is used to register the software by displaying the *AquiferWin32 Registration* dialog as below. The dialog displays a *System Code* which can be copied to the clipboard (you do this by highlighting the system code and pressing Ctrl-C) and pasted into an email to be sent to aquifer@groundwatermodels.com or aquifer@esinternational.com. We will reply with a security code that you paste or enter into the *Security Code* field in the above dialog. Alternatively, you can get a security code by calling either ESI in the United States at (610) 670-3400 or ESL in the United Kingdom at +44 1743 248600; however, it is strongly recommended that the transaction be done via email since the codes are rather long.

Security Code:	This is an encrypted code containing information about your software license and the configuration of the computer used to validate the license
Authorization Level:	The level of functionality of the application granted by the license
System Code:	This is an encrypted code containing information about your hardware configuration used to generate a unique Security Code

About WinFlow

The **About WinFlow** displays the *About WinFlow* dialog as below. This dialog presents the program version number, credits and identifies the operating system the application is running on.



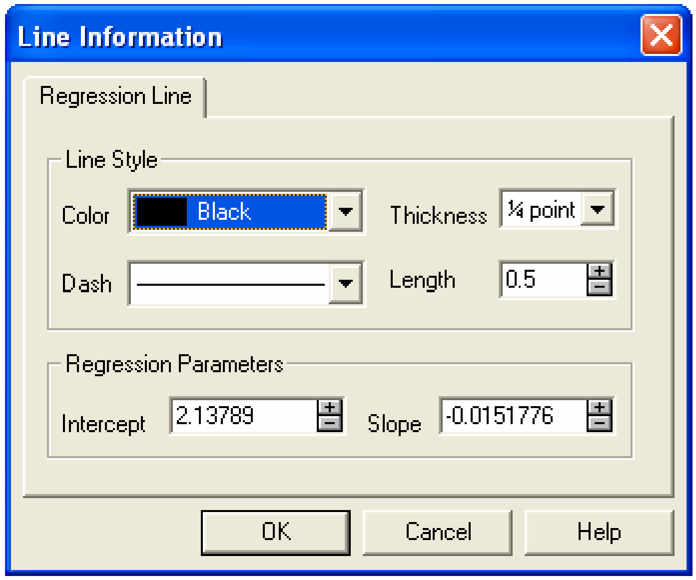
The **Register** button, when clicked, is used to register the software by displaying the *WinFlow Registration* dialog as below. The dialog displays a *System Code* which can be copied to the clipboard (you do this by highlighting the system code and pressing Ctrl-C) and pasted into an email to be sent to aquifer@groundwatermodels.com or aquifer@esinternational.com. We will reply with a security code that you paste or enter into the *Security Code* field in the above dialog. Alternatively, you can get a security code by calling either ESI in the United States at (610) 670-3400 or ESL in the United Kingdom at +44 1743 248600; however, it is strongly recommended that the transaction be done via email since the codes are rather long.

Security Code:	This is an encrypted code containing information about your software license and the configuration of the computer used to validate the license
Authorization Level:	The level of functionality of the application granted by the license
System Code:	This is an encrypted code containing information about your hardware configuration used to generate a unique Security Code

Miscellaneous Dialogs and Property Sheets

Regression Line Information

The *Regression Line Information* property sheet is used to manually set the intercept and slope of a linear regression line.



Line Style

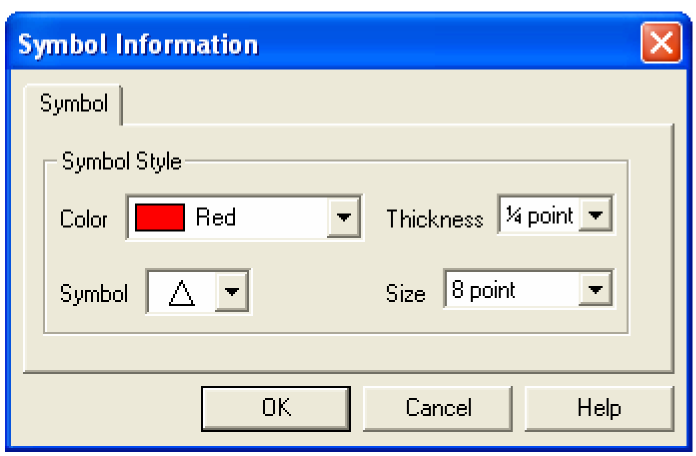
Color:	The color used to display the line
Thickness:	The thickness in points of the line
Dash:	The dash pattern to use for the line
Length:	The length in inches of the dash pattern for the line

Regression Parameters

Intercept	The y-intercept of the line
Slope	The slope of the line

Symbol Information

The *Symbol Information* property sheet is used to change the style of the symbol used to represent a specific data point.



Symbol Style

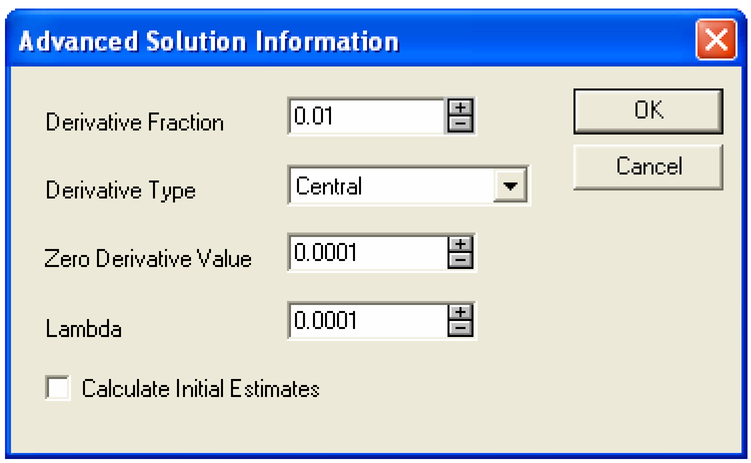
Color	The color to use when displaying the symbol
Thickness	The line thickness to use when displaying the symbol

Symbol	The symbol to display
Size	The size of the symbol in points

Advanced Solution

This *Advanced Solution Information* dialog is accessed from the **Advanced** button on the ***Solution*** tab of the *Solution Information* property sheet. If you have convergence problems, this dialog is the first place to go. Turn off **Calculate Initial Estimates** by clicking on the check-box to remove the check mark. Now, go back and try a manual match making sure to select the type curve you are matching to. Also, make sure you have fine-tuned the match after you have selected the curve. Merely selecting a type curve does not update the initial guess of the type parameter; this is only done when you perform a manual match.

If optimization fails again, you should increase the value of *Lambda* by several orders of magnitude. With higher values of Lambda the solution is more likely to converge but the precision of convergence is diminished. If you set *Lambda* to a value of 1 or 10 and you are still can't achieve convergence, go to the *Solution Information* property sheet and look at the ***Results*** tabs. One or more of the parameters are likely diverging and have become invalid. A likely candidate might be the parameter represented by the type curves. Go to the ***Exceptions*** and start enforcing minimum and maximum values. The more you constrain the solution, the more likely it will converge.



Derivative Fraction	Determines the fraction of the current parameter value to be add/subtracted when calculating derivatives during optimization
Derivative Type	Determines how derivatives are calculated

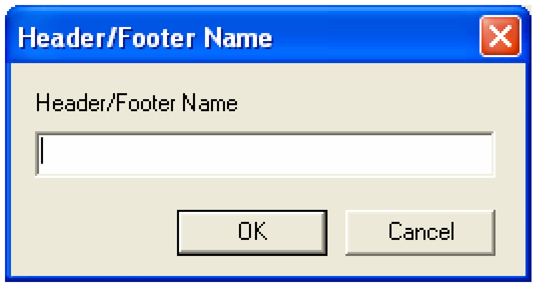
Central	Parameter Value plus or minus .5*(Derivative Fraction * Parameter Value)
Backward	Parameter Value minus (Derivative Fraction * Parameter Value)
Forward	Parameter Value plus (Derivative Fraction * Parameter Value)

Zero Derivative Value	Value to assign to (Derivative Fraction * Parameter Value) if it calculates to 0

Lambda	This parameter controls how sensitive the optimization solution is. The default value of .0001 can be increased by orders of magnitude to help solve convergence problems.
Calculate Initial Estimates	If checked, initial estimates for Transmissivity and Storage coefficient are calculated using a straight-line approximation.

Header/Footer Name

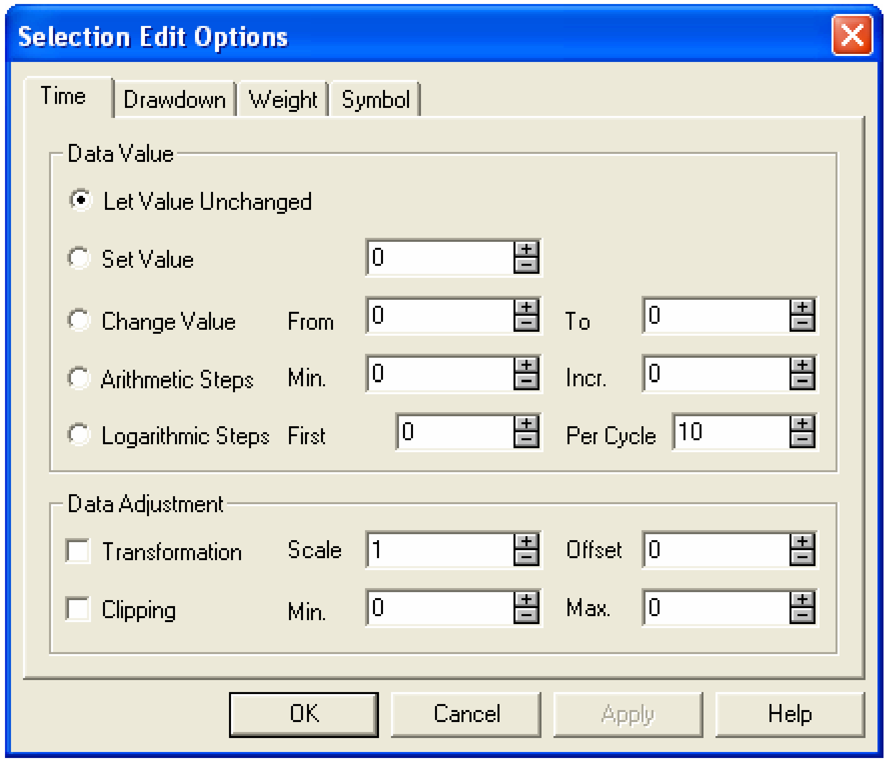
The *Header/Footer Name* dialog is accessed via the **New** and **Rename** buttons on the ***Header/Footer*** tab of *the Page Setup Options* property sheet.



Header/Footer Name	Descriptive name given to a header/footer element

Selection Edit Options

A dynamic *Selection Edit Options* property sheet is created to allow the currently selected data lines to be edited in a block. Each column visible in the spreadsheet is added as a separate tab in the property sheet with a name corresponding to the column it represents. There are two types of columns, numeric and symbol; an example of each is described below.



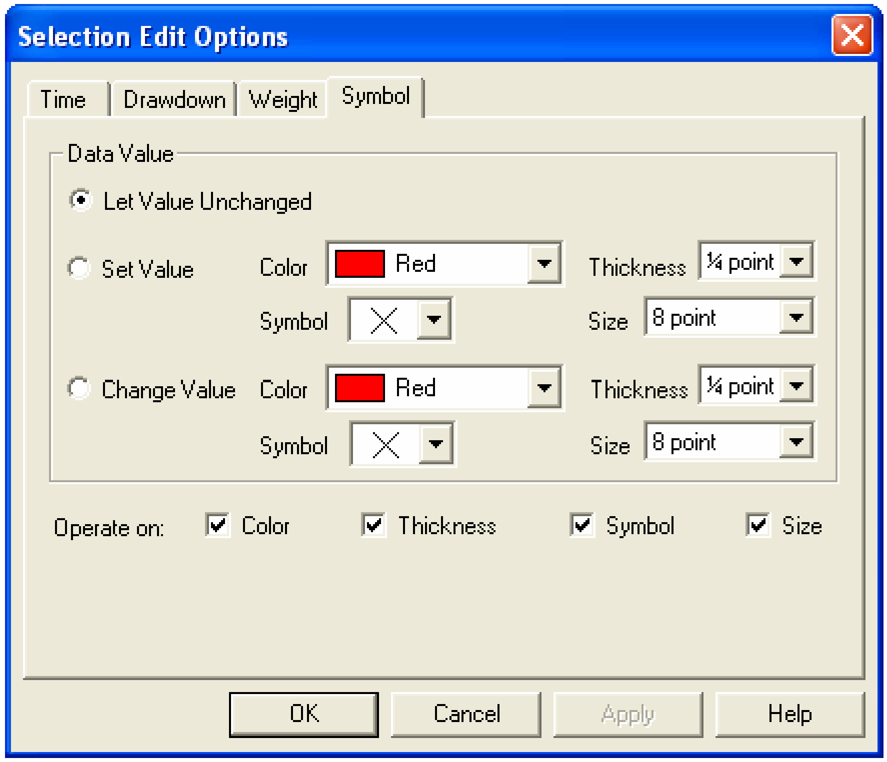
Data Value

Let Value Unchanged	When set, the data value in this column is not changed
Set Value	When set, the data value in this column is set to value in the adjacent edit field
Change Value	When set, if the data value in this column equals value in the adjacent *From* data field, the data value is changed to the value in the *To* data field
Arithmetic Steps	When set, the data value in this column for the first line of the selection is set to the value in the *Min.* data field and subsequent lines are incremented by the value in the *Incr.* data field
Logarithmic Steps	When set, the data value in this column for the first line of the selection is set to the value in the *First* data field and subsequent lines are incremented by the value calculated from the *Per Cycle* data field; the increment is calculated such that there would be *Per Cycle* evenly spaced log values per log cycle.

Time Adjustment

Transformation	When checked, the column data are transformed by multiplying it by the Scale and adding the Offset
Scale	Specifies the value by which to multiply the column value if transformation is active
Offset	Specifies the value to be added to the scaled column value if transformation is active

Clipping	When checked, the column data are clipped based on the specified Minimum and Maximum values
Minimum	Specifies the minimum acceptable value for the column data; data points with a column value less than this value are removed
Maximum	Specifies the maximum acceptable value for the column data; data points with a column value greater than this value are removed



Data Value

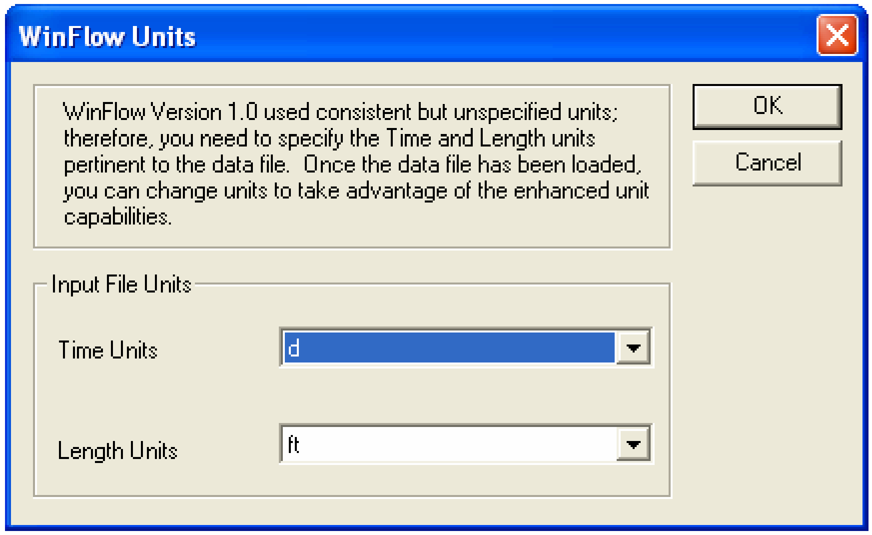
Let Value Unchanged	When set, the symbol in this column is not changed
Set Value	When set, the symbol in this column is set to the adjacent symbol styles
Change Value	When set, if the symbol in this column equals the adjacent symbol style, the symbol is changed to the symbol style defined for the **Set Value**

Operate on:

Color	When checked, the symbol definitions above include the symbol color
Thickness	When checked, the symbol definitions above include the symbol thickness
Symbol	When checked, the symbol definitions above include the symbol type
Size	When checked, the symbol definitions above include the symbol size

WinFlow Units

The *WinFlow Units* dialog sheet is used to set the units pertinent to the WinFlow Version 1.0 file being opened. WinFlow Version 1.0 used consistent units which were not specified.

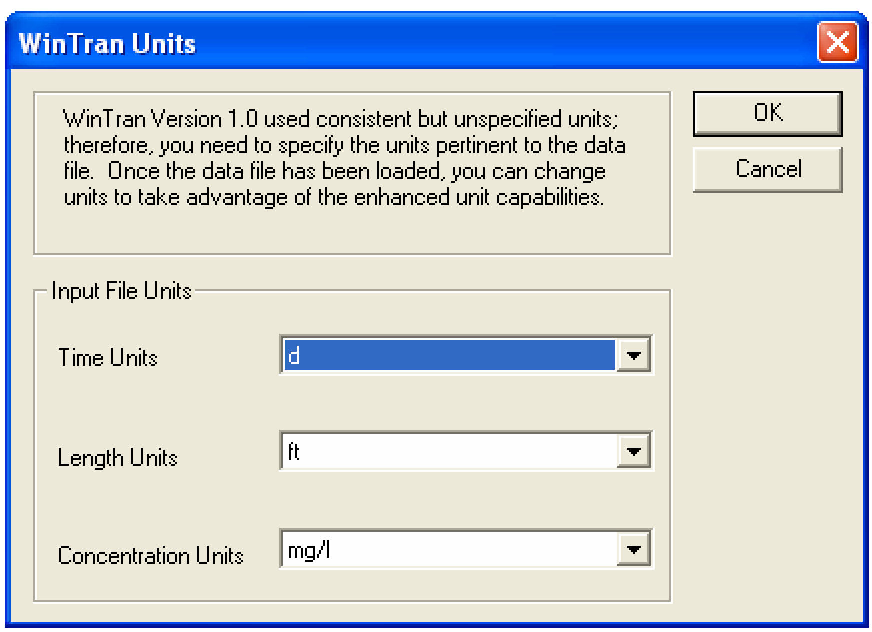


Input File Units

Time Units	The time units that apply to the WinFlow file
Length Units	The length units that apply to the WinFlow file

WinTran Units

The *WinTran Units* dialog is used to set the units pertinent to the WinTran Version 1.0 file being opened. WinTran Version 1.0 used consistent units which were not specified.

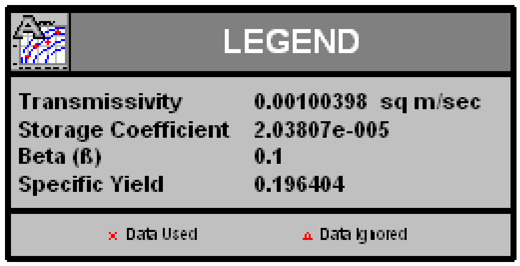


Input File Units

Time Units	The time units that apply to the WinTran file
Length Units	The length units that apply to the WinTran file
Concentration Units	The concentration units that apply to the WinTran file

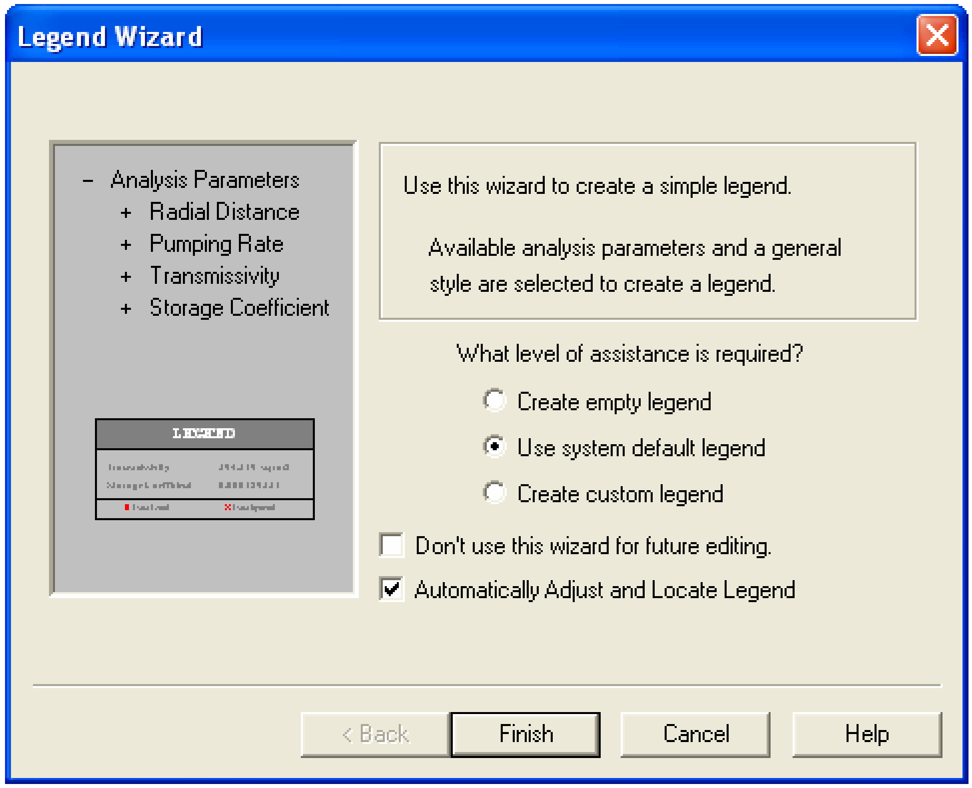
Legend Wizard

Lengends generated by the legend wizard are of the style below:

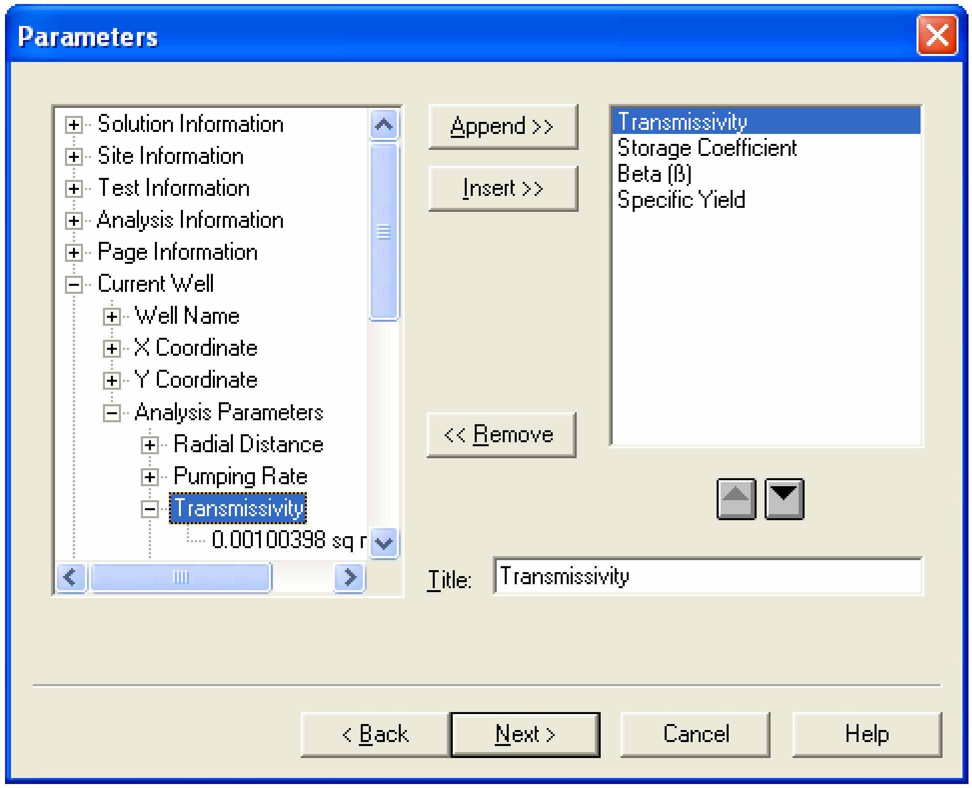


The legend has three sections, the Title sublegend, the Parameter sublegend and the Symbol sublegend. The Title and Symbol sublegends are optional as is the bitmap in the Title sublegend.

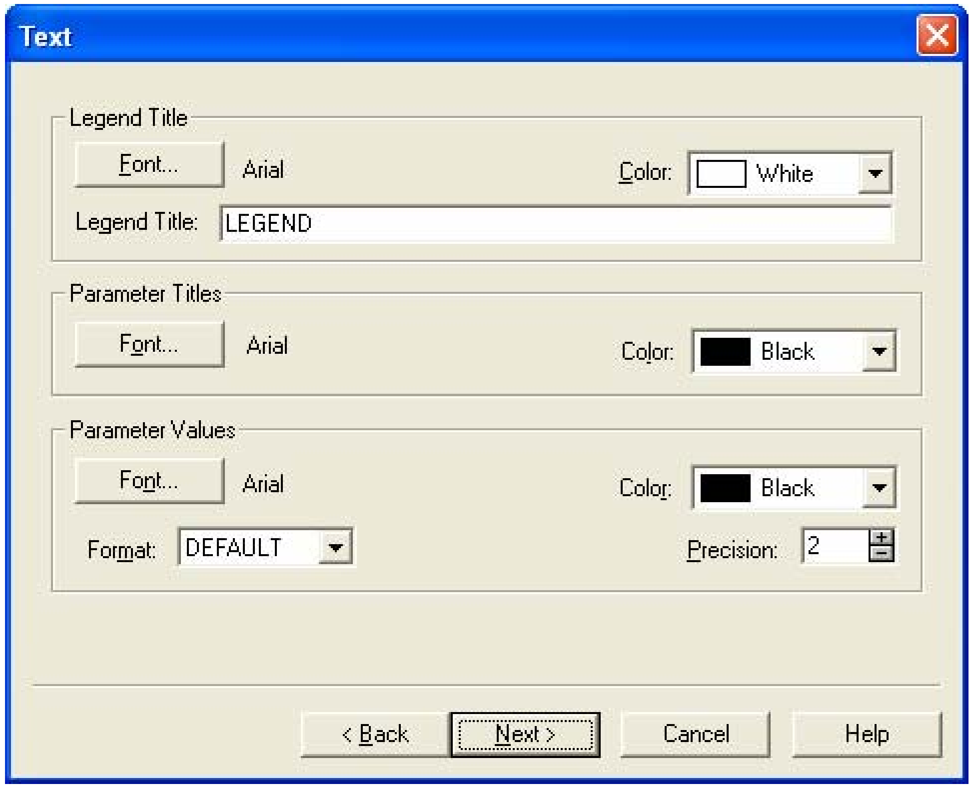
To customize the legend created with the wizard, click the **Create custom legend** radio button and the **Finish** button becomes a **Next** button to continue the customization. Two other options are available via check boxes. The **Don't use this wizard for future editing** option overrides the default behavior that, when you double click the legend to edit it, the wizard will not be used. The **Automatically Adjust and Locate Legend** check box controls whether the system will automatically maintain the location of the legend.



Create empty legend	An empty legend object is created as in previous versions so you can manually set it up.
Use system default legend	The system default legend is created for the selected analysis.
Create custom legend	The contents of the system default legend are loaded; however, subsequent pages of the wizard allow for customization of the legend.
Don’t use this wizard…	If this checkbox is checked, future editing of the legend will not use this wizard.
Automatically Adjust…	If this checkbox is checked, the contents and location of the legend will be automatically maintained.



Left tree control	The left tree control contains all parameters available for the selected analysis that could be added to the legend
Right list control	The right tree control contains the parameters that will be used to create the legend. They will appear, top to bottom, in the order they appear in this list.length
Append	The parameter selected in the left tree control will be added as the last item in the right list control
Insert	The parameter selected in the left tree control will be added above the item selected in the right list control
Remove	The parameter selected in the right list control will be removed from the list
Up Arrow	The parameter selected in the right list will be moved up in the list
Down Arrow	The parameter selected in the right list will be moved down in the list
Title	The title, to be displayed in the legend, of the item selected in the right list is displayed and editable in this field



Legend Title

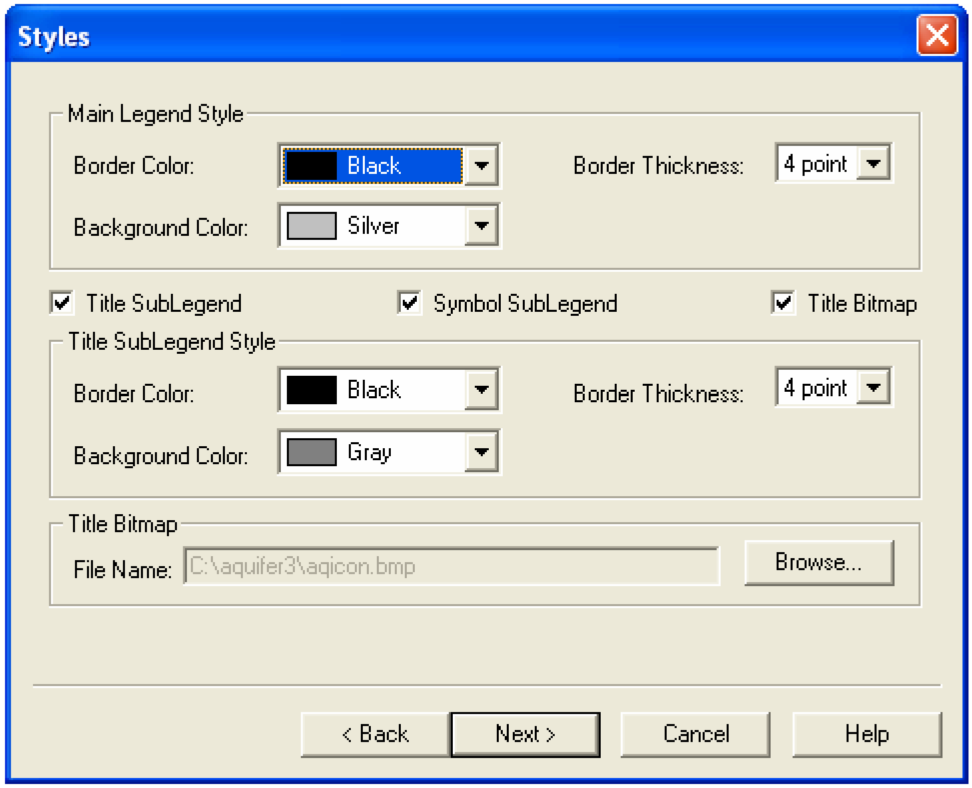
Font	Defines the font, font style and size for the legend title
Color	Defines the color used for the legend title text
Legend Title:	The title to use when creating the Title sublegend.

Parameter Titles

Font	Defines the font, font style and size for the parameter titles in the main part of the legend
Color	Defines the color used for the parameter titles

Parameter Values

Font	Defines the font, font style and size for the parameter values in the main part of the legend
Color	Defines the color used for the parameter values
Format:	The numeric format to use when displaying the parameter values
Precision:	The number of digits to the right of the decimal point to use when displaying the parameter values



Main Legend Style

Border Color:	The color to use for the border line around the main part of the legend
Border Thickness:	The line thckness of the main legend border in points
Background Color	The color to use for the background of the main part of the legend
Title SubLegend	When checked, the title sublegend is displayed in the legend
Symbol SubLegend	When checked, the symbol sublegend is displayed in the legend
Title Bitmap	When checked, a bitmap is displayed in the left hand portion of the title sublegend

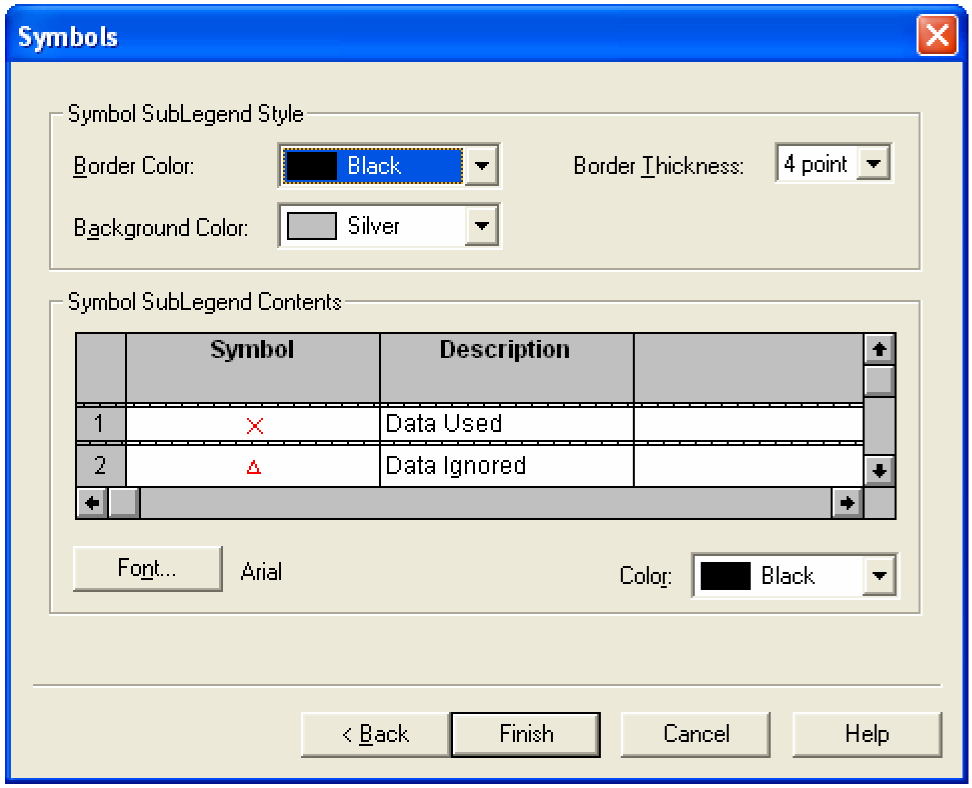
Title SubLegend Style

Border Color:	The color to use for the border line around the title sublegend part of the legend
Border Thickness:	The line thckness of the title sublegend border in points
Background Color	The color to use for the background of the title sublegend

Title Bitmap

File Name:	The name of the bitmap file that was loaded into the title sublegend; the file is stored internally so the file need not exist after it has been read

Browse	Click this button to display a File Open dialog to allow for the selection of an appropriate bitmap file



Symbol SubLegend Style

Border Color:	The color to use for the border line around the symbol sublegend part of the legend
Border Thickness:	The line thckness of the symbol sublegend border in points
Background Color	The color to use for the background of the symbol sublegend

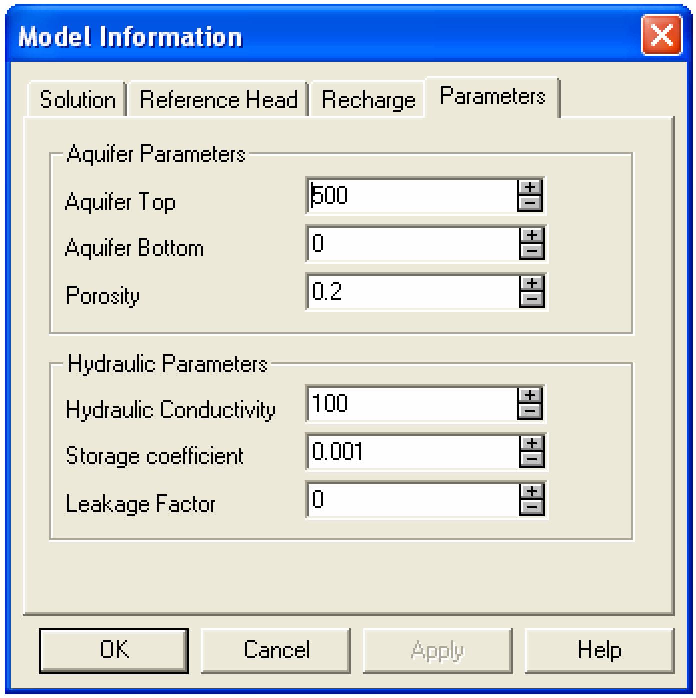
Symbol SubLegend Contents

The contents of the spreadsheet define the symbol/description combinations that will be displayed in the symbol sublegend

Font	Defines the font, font style and size for the symbol descriptions in the symbol sublegend
Color	Defines the color used for the descriptions in the symbol sublegend

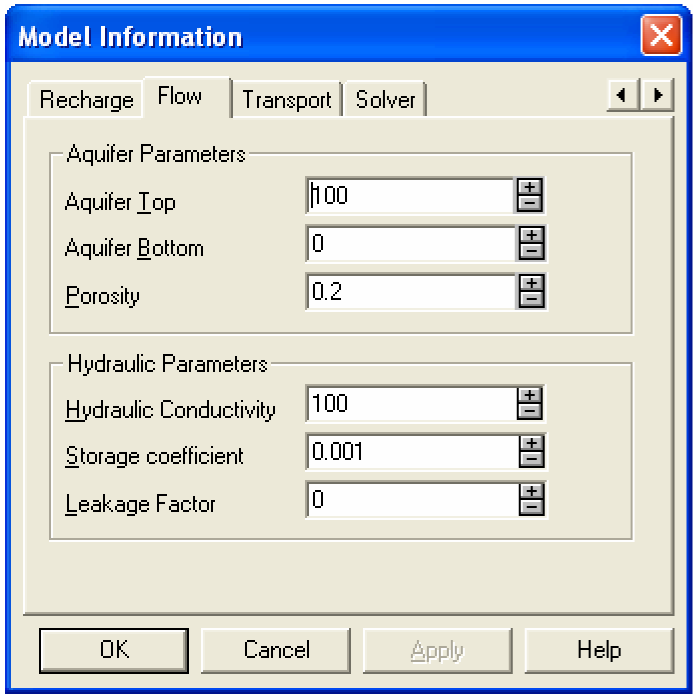
WinFlow and WinTran Parameter Tabs

Parameters



Aquifer Parameters	
Aquifer Top	Elevation of the top of the aquifer
Aquifer Bottom	Elevation of the bottom of the aquifer
Porosity	Porosity of the aquifer
Hydraulic Parameters	
Hydraulic Conductivity	Value to use for the hydraulic conductivity of the aquifer
Storage coefficient	Value to use for the storage coefficient of the aquifer
Leakage Factor	Value to use for the leakage factor of the aquifer

Flow



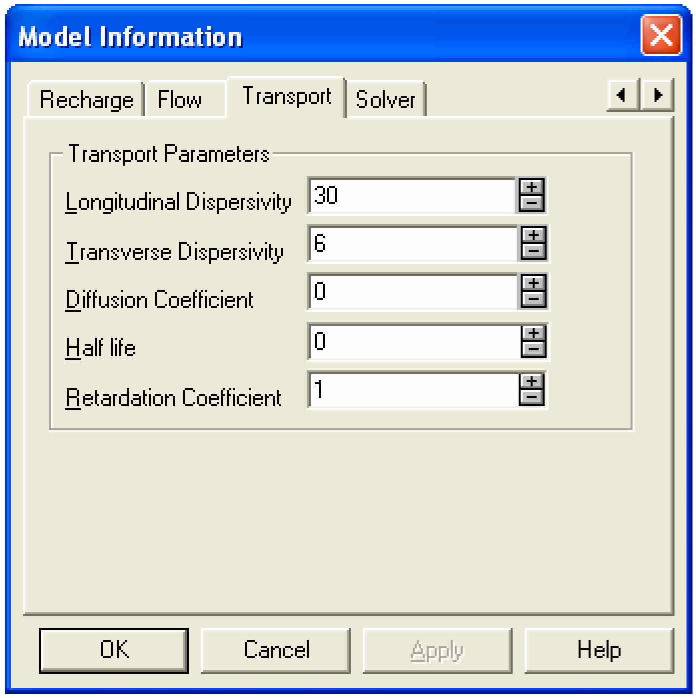
Aquifer Parameters

Aquifer Top	Elevation of the top of the aquifer
Aquifer Bottom	Elevation of the bottom of the aquifer
Porosity	Porosity of the aquifer

Hydraulic Parameters

Hydraulic Conductivity	Value to use for the hydraulic conductivity of the aquifer
Storage coefficient	Value to use for the storage coefficient of the aquifer
Leakage Factor	Value to use for the leakage factor of the aquifer

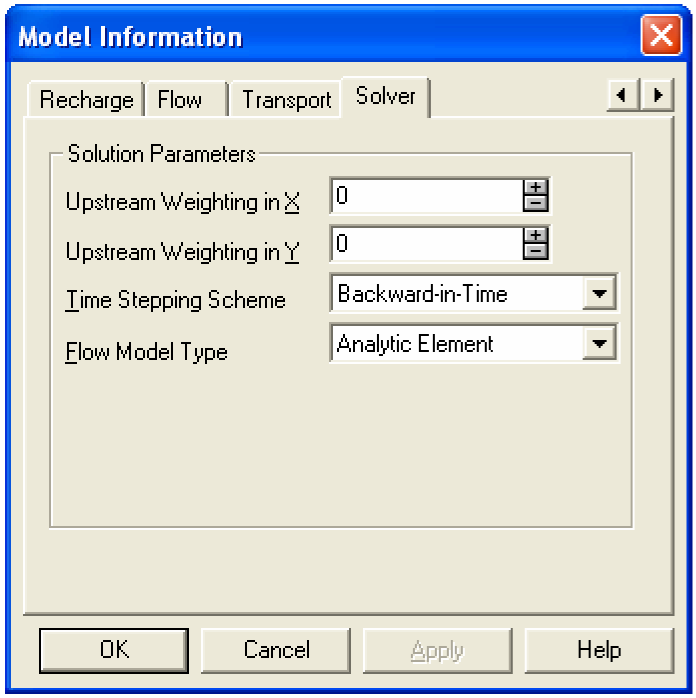
Transport



Transport Parameters

Longitudinal Dispersivity	Value to use for the longitudinal dispersivity of the aquifer
Transverse Dispersivity	Value to use for the transverse dispersivity of the aquifer
Diffusion Coefficient	Value to use for the diffusion coefficient of the aquifer
Half life	Value to use for the half life of the aquifer
Retardation Coefficient	Value to use for the retardation coefficient of the aquifer

Solver



Solution Parameters

Upstream Weighting in X	Value to use for the upstream weighting in x
Upstream Weighting in Y	Value to use for the upstream weighting in y within the WinTran solver
Time Stepping Scheme	Value to use for the time stepping scheme within the WinTran solver
Flow Model Type	Value to use for the flow model type within the WinTran solver

WinFlow/WinTran Mathematical Models

Steady-state Model

The equations used in the steady-state portion of WinFlow are derived by Otto D. L. Strack's Groundwater Mechanics (Strack, 1989). If you intend to use WinFlow routinely, you should get a copy of Groundwater Mechanics. It is well written and will give you valuable insight into the underlying assumptions of the analytical equations. You will also be introducted to more advanced techniques not included in WinFlow. The book contains sample FORTRAN source code for the analytical functions in WinFlow.

Dr. Strack is well known for his SLAEM software (Single Layer Analytic Element Model), which is much more advanced than WinFlow. He has also developed a three-dimensional version of SLAEM. If you like analytical modeling with WinFlow but need more power and flexibility, you may be interested in SLAEM. Dr. Strack can be reached in the Civil and Mineral Engineering Department of the University of Minnesota.

The analytic functions developed by Strack (1989) use the principle of superposition to compute the head at a point in the aquifer system. The total effect resulting from several analytic functions, such as a pumping wells, is equal to the sum of the individual effect caused by each analytic function acting separately. For example, if you wanted to compute the total drawdown at a point resulting from three pumping wells, you would compute the drawdown caused by each well at that point and then sum the drawdowns.

WinFlow allows you to select from a number of analytic functions to simulate two-dimensional horizontal ground-water flow, including

Uniform regional flow,

Wells,

Recharge (elliptical area),

Circular recharge areas (Ponds), and

Line sinks and sources.

The head at any point in the system may be computed by summing the effects of any number of the above functions. The equations used to compute the head are described below.

The analytic equations are expressed in terms of discharge potential to keep the equations linear for both confined and unconfined flow. The discharge vector points in the direction of ground-water flow and the magnitude of the discharge potential equals the volume of water flowing through a cross-section of unit width. In computing head at a point, the discharge potential is computed first and then converted to head using the following equations:

for confined flow:

$$\phi = \frac{\Phi + \frac{1}{2} K H^2}{KH}$$

and for unconfined flow:

$$\phi = \sqrt{\frac{2\Phi}{K}}$$

where:

Φ = discharge potential (L^3/T)

K = hydraulic conductivity (L/T)

H = aquifer thickness (L)

ϕ = head (L)

Using these formulae, WinFlow automatically accounts for the transition from confined to unconfined flow. The following equations are used to compute the discharge potential at any point (x,y) in the system. The equations above are then used to convert the discharge potential to head.

Uniform Flow

$$\Phi(x, y) = -Q_o(x \cos\alpha_u + y \sin\alpha_u) + C$$

Wells

$$\sum_{j=1}^{n} \frac{Q_j}{4\pi} \ln[r_j^2(x, y)] +$$

Recharge

$$-\tfrac{1}{2}\frac{N}{a^2+b^2}[(a^2\sin^2\alpha_r+b^2\cos^2\alpha_r)(x-x_r)^2-$$

$$2(a^2-b^2)(x-x_r)(y-y_r)\sin\alpha_r\cos\alpha_r$$

$$+(a^2\cos^2\alpha_r+b^2\sin^2\alpha_r)(y-y_r)^2-a^2b^2]$$

Ponds

Inside Pond:

$$-\sum_{j=1}^{n}\tfrac{1}{4}[(x-x_{p_j})^2+(y-y_{p_j})^2-R^2{}_{p_j}]\,N_{p_j}$$

Outside Pond:

$$-\sum_{j=1}^{n}\tfrac{1}{4}[R^2{}_{p_j}\ln[\frac{(x-x_{p_j})^2+(y-y_{p_j})^2}{R^2{}_{p_j}}]]\,N_{p_j}$$

Linesinks of Known Flux

$$+\sum_{j=1}^{n}\frac{\sigma_j L_j}{4\pi}\Re\{(Z_j+1)\ln(Z_j+1)-(Z_j-1)\ln(Z_j-1)+2\ln[\frac{1}{2}(\frac{2}{z_j}-\frac{1}{z_j}]-2\}$$

where:

x	= x coordinate of calculation point
y	= y coordinate of calculation point
Q_o	= uniform flow $[L^2/T]$
α_u	= angle between uniform flow and x-axis
Q_j	= discharge of well j $[L^3/T]$
r_j	= distance from well j to calculation point [L]
N	= recharge rate [L/T]
a	= length of a-axis of recharge ellipse [L]
b	= length of b-axis of recharge ellipse [L]
x_r	= x coordinate of center of recharge ellipse [L]
y_r	= y coordinate of center of recharge ellipse [L]
α_r	= angle between a-axis and x-axis

$x_{p\,j}$	= x coordinate of center of pond j [L]
$y_{p\,j}$	= y coordinate of center of pond j [L]
$R_{p\,j}$	= radius of pond j [L]
$N_{p\,j}$	= infiltration rate of pond j
σ_j	= flow per unit length for linesink j [L^2/T]
L_j	= length of linesink j [L]
z^1	= starting coordinates of linesink j
z^2	= ending coordinates of linesink j
C	= constant
z	= x + iy

$$Z_j = Z_j(z, \frac{1}{z_j}, \frac{2}{z_j}) = \frac{z - \frac{1}{2}(\frac{1}{z_j} + \frac{2}{z_j})}{\frac{1}{2}(\frac{2}{z_j} - \frac{1}{z_j})}$$

The uniform flow component above does not contain a gradient term explicitly, even though you enter the gradient in WinFlow to define uniform regional flow. The Q_o term represents the flow per unit width of aquifer and is computed as $Q_o = KBi$, where: i = the gradient, K = hydraulic conductivity, and B = saturated thickness. WinFlow computes the Q_o term at the reference point; therefore, you do not need to enter Q_o.

There are two equations for ponds depending upon whether the point (x,y) is located inside the pond or outside of the pond. Thus, either pond equation is used, but not both.

The term that computes the contribution to the discharge potential for line sinks is expressed in terms of complex numbers. The expression $\Re\{\}$ signifies that the real portion of the complex number computed by the complex expression {} is used in the equation.

The expression for the discharge potential contains one unknown constant C. The constant C is evaluated by requiring that the potential be known at some point (x_o,y_o) in the system. Once this potential is known, the equation is solved for the constant C. An important ramification of this approach is that the head always equals the reference head at the reference point. This approach is equivalent to setting a constant head cell in a numerical model. It is very important to keep this reference head as far as possible from the area of interest.

WinFlow allows you to specify linesinks of unknown flux by defining a head at the center of the linesink. For *n* linesinks of specified head, there are *n+1* unknowns (the flux for each linesink and the constant C). In this case, the equations are solved numerically to compute the constant C and the flux for each linesink of specified head.

Transient Model

Basic Models

The transient model in WinFlow uses the analytical solutions of Theis (1935) and Hantush and Jacob (1955) to compute drawdown from a pumping well. Drawdowns from multiple individual wells are then added using the principle of superposition (Reilly et al., 1987) to compute the effective drawdown at a point. Finally, the cumulative drawdown is subtracted from a planar potentiometric surface. The surface may be horizontal or inclined at some angle, given by the uniform gradient vector.

The procedure for calculating the head at any point and time (x,y,t) is given below:

$$\phi(x, y, t) = C - G(x\cos\alpha + y\sin\alpha) - \sum_{j=1}^{n} s_j$$

where:

ϕ	= head
G	= regional gradient [L/L]
α	= angle between regional gradient and x-axis
(x,y)	= coordinates of calculation point
t	= time to compute drawdown
s_j	= drawdown computed for well j
C	= constant

The constant C is computed using the reference head, as in the steady-state model. The main difference between the steady-state model and the transient model is that the reference head is maintained at a constant value in the steady-state model. However, the reference head is simply a starting point for calculations in the transient model. That is, drawdowns computed at the reference location are subtracted from the reference head. The constant C is evaluated as follows:

$$C = \phi_r + G(x_o\cos\alpha + y_o\sin\alpha)$$

where:

ϕ_r	= reference head
x_o	= x coordinate of reference head
y_o	= y coordinate of reference head

There is an option, however, to keep the reference head at a constant value. This option was added so that the results would be consistent with the steady-state model. If you elect to keep the reference head constant in the transient model, drawdown is computed at the reference head location and then added to all heads in the contour matrix. The result is that the potentiometric surface is raised by a constant value.

Although the absolute values of head will be different between the two approaches, the flow directions and travel times (using particle-tracking) will be identical. The reference head should not be held constant if a drawdown model is being calculated because there would be zero drawdown at the reference head location.

The drawdown (s_j) is computed from one of two equations. If the leakage factor (L) is zero, the Theis equation is used. If leakage is nonzero, the Hantush and Jacob leaky aquifer solution is computed.

The Theis (1935) equation for unsteady flow to a well in a confined aquifer makes the following simplifying assumptions:

aquifer has infinite areal extent;

aquifer is homogeneous, isotropic, and of uniform thickness;

aquifer potentiometric surface is initially horizontal;

pumping rate is constant;

well fully penetrates the aquifer;

horizontal ground-water flow;

aquifer is confined;

flow is unsteady;

water is released instantaneously from storage with decline of hydraulic head;

diameter of pumping well is very small so that storage in the well can be neglected;

Drawdown is calculated as described below.

$$s = \frac{Q}{4\pi T} w(u)$$

where:

w(u) = Theis well function = $\int_u^\infty \frac{e^{-y}}{y} dy$

u = r^2 S / (4 T t)

r = distance from well to point (x,y)

T = aquifer transmissivity [L^2/T]

S = storage coefficient [dimensionless]

t = time

Q = pumping rate [L^3/T]

The Theis well function, also known as the exponential integral, is computed in WinFlow using a numerical approximation given by Abramowitz and Stegun (1965). This approach is verified in the next chapter.

The Hantush and Jacob (1955) equation for unsteady flow to a well in a semi-confined aquifer with no storage in aquitards makes the following simplifying assumptions:

- aquifer has infinite areal extent;
- aquifer is homogeneous, isotropic, and of uniform thickness;
- aquifer potentiometric surface is initially horizontal;
- pumping rate is constant;
- well fully penetrates the aquifer;
- horizontal ground-water flow;
- aquifer is semi-confined;
- flow is unsteady;
- water is released instantaneously from storage with decline of hydraulic head;
- diameter of pumping well is very small so that storage in the well can be neglected;
- confining bed(s) has infinite areal extent, uniform vertical hydraulic conductivity, and uniform thickness;
- confining bed(s) is overlain or underlain by an infinite constant-head plane source; and
- flow in the aquitard is vertical.

Drawdown is calculated as described below.

$$s = \frac{Q}{4\pi T} w(u, \frac{r}{B})$$

where:

w(u,r/B) = Hantush well function = $\int_u^\infty \frac{1}{y} e^{[-y-\frac{r^2}{4B^2y}]} dy$

u = r^2 S / (4 T t)

B = $\sqrt{\frac{Tb'}{K'}}$

b' = thickness of aquitard [L]

K' = vertical hydraulic conductivity of aquitard [L/T]

T = aquifer transmissivity [L^2/T]

The Hantush and Jacob well function is evaluated numerically using a method described by Case et al. (1979). The next chapter verifies that the Hantush and Jacob (1955) well function calculations are accurate.

Implementing Ponds and Linesinks

Ponds and linesinks are available for the transient model as well as the steady-state model. The pond element is implemented using the Hantush (1967) analytical solution for computing the water-table rise beneath a circular recharging area. Linesinks (flux only) are implemented approximately using a series of wells evenly spaced along the linesink. You may determine the number of wells used to approximate each linesink. It will be more accurate as the number of wells increases. Both pond and linesink transient elements are described below.

Ponds are computed in the transient model using the Hantush (1967) method for circular recharge areas. WinFlow uses the approximate version of the Hantush mound equation, given as follows:

for r < R:

$$h^2 - h_i^2 = \frac{V}{2\pi K}[W(u_0) - (\frac{r}{R})^2 e^{-u_0} + \frac{1}{u_0}(1 - e^{-u_0})]$$

and for r > R:

$$h^2 - h_i^2 = \frac{V}{2\pi K}[W(u) + 0.5\, u_0\, e^{-u}]$$

where:

h	= the water-table elevation (above the datum plane)
h_i	= the initial water-table elevation without the pond
K	= hydraulic conductivity (L/T) of the aquifer
W(u)	= Theis well function
u_0	= $R^2/4vt$
u	= $r^2/4vt$
t	= time after start of infiltration
v	= Kb/S
V	= $w\pi R^2$
w	= constant percolation rate (L/T)
S	= storativity

b	$= 0.5[h_i + h]$
R	= radius of the pond (L)
r	= radius of calculation point from center of pond (L)

The Hantush (1967) mound solution was developed with the following simplifying assumptions:

the water-table rise is less than 2 percent of the saturated thickness

$t \geq 0.5\ r^2/v$ ($u \leq 0.5$) for $r < R$

$t \geq 0.5\ R^2/v$ for $r > R$

otherwise, it uses the same assumptions as the Theis solution.

Linesinks are simulated in the transient model using an approximate method. The linesink is discretized into *n* evenly spaced wells with one well located at either end of the linesink. Each well in the interior of the linesink pumps at a rate of Q/(n-1) and the wells at the endpoints of the linesink pump at a rate of 0.5 Q/(n-1). This approximation becomes more accurate as the number of wells increases. You control the number of wells used to approximate linesinks in WinFlow.

Solute Transport Model

Introduction

Closed form analytical solutions to the governing equations of ground- water flow have wide application in subsurface remediation projects. Complex flow problems can be solved using these analytical techniques. The analytic element method developed by Strack (1989), as discussed in the previous section, is especially useful in modeling complex two- dimensional ground- water flow systerns. The analytic elements include wells, line- sinks, and recharge areas, among others, that can be used to simulate a variety of subsurface remedial alternatives. While these analytic techniques cannot treat the range of complexity provided by numerical techniques, the analytical models have advantages over numerical models in ease of use and speed of application.

Analytical solutions to the solute transport equations, on the other hand, are not as directly applicable to remediation projects. One of the primary problems with transport analytical solutions is the inability to treat changes in the flow field caused by wells, drains, and recharge. Transport solutions are normally limited to a uniform groundwater flow field. In order to obtain useful solutions to transport problems, therefore, the modeler must resort to more powerful numerical techniques, which require more time and effort to simulate.

A hybrid technique has been developed for use in WinTran that combines an analytical flow model with a numerical transport model. This technique combines the ease of use of an analytical model with the flexibility of a numerical model. The flow model utilizes the analytical element techniques of Strack (1989). The transport model is based upon the finite- element method using rectangular elements and linear basis functions. The two models are both contained within WinTran.

The hybrid model first solves for the flow field using the analytic element method. Boundary conditions for the finite- element model are then automatically taken from the analytical flow model. The finite- element mesh is coincident with the head

matrix used to contour results obtained from the flow simulation. Thus, you do not need to explicitly design a numerical grid or mesh system of nodes. You simply specify the location of the mesh and the number of rows and columns in the mesh. Because you are somewhat insulated from the mesh design, significant error-checking facilities are provided to warn of large mass balance errors and other potential problems such as violating specified Peclet and Courant criteria.

The Hybrid Approach

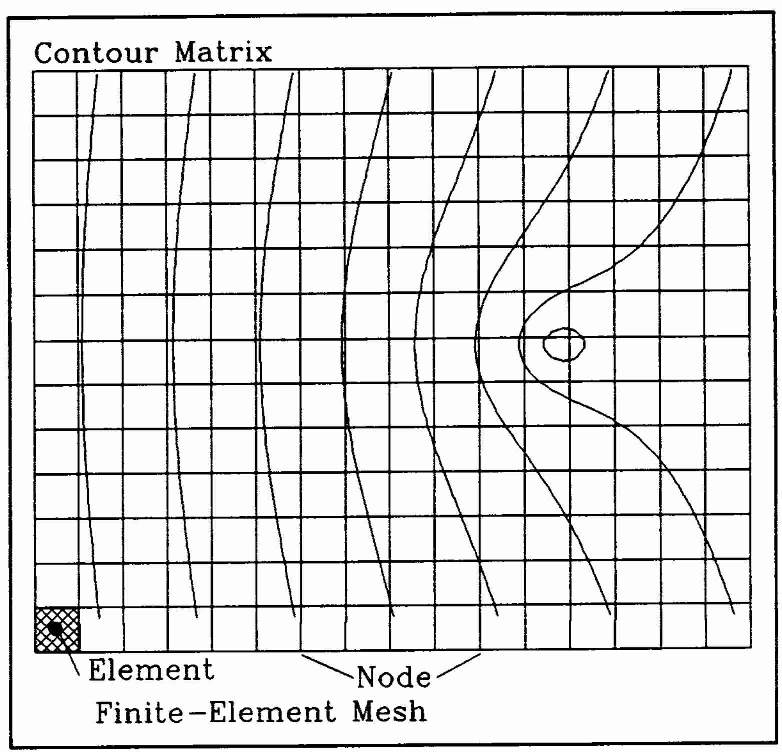
The hybrid analytical flow/numerical transport model combines the analytic element method developed by Strack (1989) with a finite- element transport technique developed by Huyakorn and others (1983). The model is constructed in six stages, most of which are transparent to the user. The six stages indude the following:

(1) The modeler designs the analytical flow model by specifying uniform aquifer properties, a regional hydraulic gradient, and analytic elements (e.g. wells, line sinks, circular recharge areas, and uniform recharge). The flow model was derived from the WinFlow model (ESI, 1995).

(2) The analytical flow model is infinite in extent; however, the user must specify a rectangular region of interest where head is computed and contoured.

(3) Head is computed at discrete points over the rectangular area of interest and a contour map is produced. These points are arranged in a regular mesh of n rows by m columns called the contour matrix. The spacings between rows and between columns are constant.

(4) Ground- water velocities are computed analytically at the centroid of each rectangular cell in the contour matrix (See the Figure below). These velocities are provided directly to the transport model and the contour matrix defines the finite-element mesh.



(5) Specify initial concentrations over the contour matrix and the nature and extent of contaminant sources.

(6) The finite- element transport model is solved for the specified simulation time(s) and results are contoured.

These six stages require relatively little user- intervention. For example, the finite- element mesh data are generated automatically. In addition, ground- water velocities are recomputed each time a change is made to the flow model. The element velocities are passed automatically to the transport model.

The Finite Element Transport Model

The solute transport model solves the partial differential equation describing the advection and dispersion of the dissolved species, as shown below:

$$\frac{\partial}{\partial x}(D_{xx}\frac{\partial c}{\partial x})+\frac{\partial}{\partial y}(D_{yy}\frac{\partial c}{\partial y})+\frac{\partial}{\partial x}(D_{xy}\frac{\partial c}{\partial y})+\frac{\partial}{\partial y}(D_{yx}\frac{\partial c}{\partial x})$$

$$-V_x\frac{\partial c}{\partial x}-V_y\frac{\partial c}{\partial y}=\phi R\frac{\partial c}{\partial t}+\lambda\phi Rc+q(c-c^*)$$

where c is the solute concentration (M/L^3); c' is the solute concentration in the injected water (M/L^3); D_{xx}, D_{yy}, D_{xy}, and D_{yx} are the components of the hydrodynamic dispersion tensor (L^2/T); ϕ is porosity (dimensionless); R is the retardation factor (dimensionless); q is the injection rate per volume of aquifer material ($L^3/T/L^3$); and λ is the first- order decay coefficient (T^{-1}). The Darcy velocity components are computed by the analytical flow model at the element centroids, as described above.

The dispersion coefficients are computed as described below:

$$D_{xx}=\frac{(\alpha_L-\alpha_T)V_x^{\,2}}{|V|}+\alpha_T|V|+D_{xx}^{\,*}$$

$$D_{yy}=\frac{(\alpha_L-\alpha_T)V_y^{\,2}}{|V|}+\alpha_T|V|+D_{yy}^{\,*}$$

$$D_{xy}=D_{yx}=\frac{(\alpha_L-\alpha_T)V_yV_x}{|V|}$$

where α_L is the longitudinal dispersivity, α_T is the transverse dispersivity, D* is the molecular diffusion coefficient (L^2/T), and |V| is the magnitude of the Darcy velocity (L/T).

The retardation factor is computed from the aquifer bulk density (ρ_s in M/L^3) and the distribution coefficient (k_d in L^3/M) as described below:

$$R=\frac{\rho_s k_d}{\phi}+1$$

Boundary conditions for the transport model include prescribed concentration and mixed- type boundaries. The latter are used around the edges of the finite- element mesh, where solute is removed if flow is exiting the model domain.

The solute transport equation is solved at each node in the finite element mesh using the Galerkin finite element method. A simplification has been adopted for

rectangular elements with linear basis functions. The technique is called the influence coefficient method and is described by Huyakorn and others (1983). The finite element formulation results in a system of linear algebraic equations with an asymmetric banded coefficient matrix. The matrix is solved using a direct solver based on the LU decomposition of a banded matrix. The finite element equations are not presented but can be found in Huyakorn and others (1983) or in Huyakorn and Pinder (1983).

WinFlow/WinTran Verification

Introduction

Verification is the process of demonstrating that the computer program performs as documented. In the case of a model, such as WinFlow, verification tests for proper implementation of the applicable equations. These equations are documented in Chapter 5 and are tested in this chapter.

The steady-state and transient models are tested separately, as described below. In each case, the model is first tested using a simple example that can be solved with a calculator. Next, WinFlow computations are compared against either another code solving the same problem or against published answers. The steady-state model is further tested by comparing WinFlow results against those of a popular numerical model, MODFLOW (McDonald and Harbaugh, 1988).

Steady-state Model

Three sets of verification problems are presented for the steady-state analytical functions used in WinFlow. In the first problem, a simple uniform flow field with a single pumping well is solved using WinFlow and a calculator. This is one of the more common uses for WinFlow and illustrates that the basic code functions are programmed accurately. In the second case, a series of problems are benchmarked against the program SLWL (Strack, 1989). Finally, a simple test case of a single well in a uniform unconfined flow field is a benchmark against the numerical model, MODFLOW.

Case 1: Uniform Flow with a Single Well

The steady-state analytic function for a single well in a uniform flow field is given by Strack (1989) as follows:

$$\Phi = -Q_o(x\cos\alpha + y\sin\alpha) + \frac{Q}{4\pi}\ln[r^2(x,y)] + C$$

where

Φ	= discharge potential [L^3/T],
Q_o	= uniform ground-water flow [L^2/T],
x,y	= coordinates of the calculation point,
α	= angle between uniform flow and x-axis,
r(x,y)	= distance from the well to the calculation point (x,y),
Q	= well discharge [L^3/T],
C	= constant.

In a confined aquifer system, the discharge potential, Φ, is converted to head (ϕ) by the following equation.

$$\phi = \frac{\Phi + \frac{1}{2} KH^2}{KH}$$

where

ϕ	= head [L],
K	= hydraulic conductivity [L/T],
H	= aquifer thickness [L].

The constant, C, is evaluated by specifying a reference head at a certain location within the flow system. The reference head remains constant during all subsequent calculations. The constant, C, is computed as follows:

$$C = \Phi_o + Q_o(x_o \cos\alpha + y_o \sin\alpha) - \frac{Q}{4\pi} \ln[r^2(x_o, y_o)]$$

where

Φ_o	= reference discharge potential,
(x_o,y_o)	= coordinates of reference head.

In the first verification problem, the aquifer is confined with a uniform regional gradient parallel to the x-axis. The problem assumptions and parameters are listed below.

K = 100 ft/d

H = 100 ft

Gradient (i) = 0.01 ft/ft

Q_o = KiH = 100 ft^2/d

reference head, ϕ_o = 200 ft at (x_o=0,y_o=0)

$\Phi_o = KH\phi_o - ½KH^2$ = 1500000 ft^3/d

Q = 100,000 ft^3/d at (x=1000,y=1000)

Using these parameters and equation (3), the constant C equals 1,384,541. Table 1 lists the results of hand calculations and WinFlow results (using the Point Calculation option) for a series of coordinates. The two results are identical to five significant figures; the calculator results were rounded to five figures. Thus, WinFlow computes the correct answer for this test case.

Table 1 Comparison between WinFlow and calculator results for test case 1.

X	Y	Φ	φ	φ (WinFlow)
0	1000	1,494,480	199.45	199.448
250	1000	1,464,902	196.49	196.491
500	1000	1,433,449	193.34	193.345
750	1000	1,397,417	189.74	189.742
1000	1000	1,284,441	178.44	178.444
1250	1000	1,347,417	184.74	184.742
1500	1000	1,333,449	183.34	183.345
1750	1000	1,314,902	181.49	181.491
2000	1000	1,294,481	179.45	179.448

Case 2: Benchmark with SLWL

The SLWL program is provided with the book, Groundwater Mechanics, (Strack, 1989). SLWL performs the same calculations as WinFlow. The primary difference between the two codes is that SLWL is written in FORTRAN, while WinFlow is written in the C programming language. SLWL has additional capabilities to those of WinFlow but is not as user-friendly nor does SLWL have good output capabilities.

A series of twelve test cases are developed to test each of the major components in WinFlow, including wells, ponds, linesinks, and recharge. Each feature added to the simulation is designed to produce a significant impact on the flow field, so that significant errors would be easily detected. Both confined and unconfined conditions are tested. These verification data sets are included on the WinFlow disk. The data file names are VER1.WFL, VER2.WFL, ..., and VER12.WFL.

SLWL was modified to export a SURFER contour matrix (grid file) in the same manner as WinFlow. The SURFER grid files were then subtracted from one another to create a matrix of differences. A simple program was created to compute the mean and maximum difference. The results are summarized in Table 2. The features tested in each simulation are summarized in Table 2, along with the mean and maximum differences between the two codes. The specific details of each test may be examined by retrieving the verification data files from within WinFlow.

The maximum difference for each simulation was a uniform value of 0.000198 feet. The maximum error was constant, probably due to a consistent difference in the computational algorithms used in the C and FORTRAN compilers used for the two codes (Microsoft FORTRAN and Microsoft Visual C++). The mean error for each run varied from a low of 0.00000186 (VER6.WFL) to a high of 0.0000139

(VER7.WFL). In all cases, the differences between the two codes are on the order of 1.0×10^{-6} percent.

Table 2 Mean and maximum differences between WinFlow and SLWL in 12 test cases.

Data File	Uniform	Wells	Ponds	Line-sinks (head)	Line-sinks (flux)	Recharge	Aquifer Type (C/U)	Max. Error	Mean Error
ver1.wfl	✓	✓					C	0.000198	0.0000037
ver2.wfl	✓	✓					U	0.000198	0.0000019
ver3.wfl	✓	✓	✓				C	0.000198	0.0000038
ver4.wfl	✓	✓	✓				U	0.000198	0.0000020
ver5.wfl	✓	✓		✓			C	0.000198	0.0000051
ver6.wfl	✓	✓		✓			U	0.000198	0.0000019
ver7.wfl	✓	✓			✓		C	0.000198	0.0000014
ver8.wfl	✓	✓			✓		U	0.000198	0.0000066
ver9.wfl	✓	✓	✓	✓	✓		C	0.000198	0.0000048
ver10.wfl	✓	✓	✓	✓	✓		U	0.000198	0.0000030
ver11.wfl	✓	✓	✓	✓	✓	✓	C	0.000198	0.0000048
ver12.wfl	✓	✓	✓	✓	✓	✓	U	0.000198	0.0000030

Case 3: Benchmark with Numerical Model

A final test of the steady-state analytic functions in WinFlow is a comparison with a numerical model. The model chosen for comparison is MODFLOW (McDonald and Harbaugh, 1988), which is a three-dimensional, finite-difference ground-water flow model developed by the United States Geological Survey. MODFLOW is one of the most widely used numerical ground-water flow models.

A simple problem involving a single pumping well in a uniform flow field is chosen as the test case. The aquifer is unconfined with homogeneous properties. The model parameters are summarized below for the WinFlow data set.

K = 100 ft/d;

Aquifer bottom elevation = 0.0 ft;

Gradient (i) = 0.001 ft/ft at an angle of 0^{o} to the x-axis;

Q_o = KiH = 10 ft^2/d;

ϕ_o = 100 ft at (x_o=0, y_o=0).

A single well located at coordinates (x=5000, y=5000) pumps 100,000 ft^3/d. The WinFlow input data file for this problem is provided on the distribution disk. The file name is "modf1.wfl".

Additional information is required to simulate the same system with a numerical model, such as MODFLOW. A finite-difference grid was constructed measuring 10,000 feet in both the x- and y-directions. There are 125 rows and 125 columns in the grid, with a cell spacing of 80 ft. A constant head of 100 ft was placed along the first column and a constant head of 89.532 was placed along the last column. The odd number was used to maintain a constant regional flow of 10 ft^3/d/ft across the finite-difference grid under nonpumping conditions. The MODFLOW data set for this problem are contained on the WinFlow disk. Several files are required for input to the MODFLOW code. The files have a common root file name of "wflow" and a three-letter extension designating the MODFLOW package name. The MODFLOW files for this problem are as follows:

WFLOW.BAS	Basic Package Input
WFLOW.BCF	Block-Centered-Flow Package Input
WFLOW.SIP	Strongly Implicit Package Input
WFLOW.WEL	Well Package Input
WFLOW.OC	Output Control Input

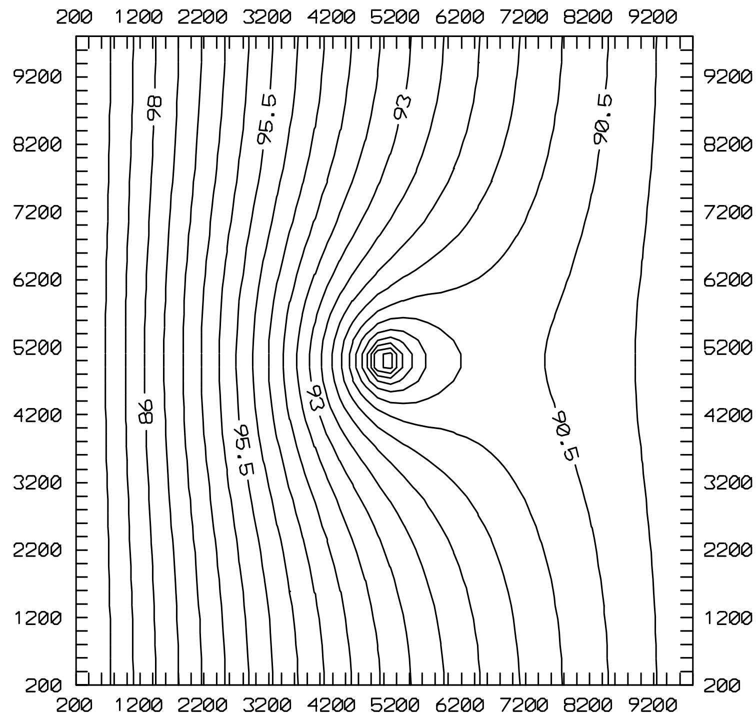
The WinFlow and MODFLOW calculations were compared by producing a SURFER grid file with 50 rows and 50 columns. The grid corners are located at (x=200, y=200) and (x=9800, y=9800). The two grid files were subtracted from each other to obtain a head difference file. A simple program was written to compute the maximum and mean differences. Contour maps produced for the WinFlow and MODFLOW results are also shown in Figure 1.

In the initial test case, MODFLOW and WinFlow compare favorably, with a maximum error of 0.84 feet and a mean error of 0.25 feet. The change in head across the model is 10.468 feet. Thus, there is a maximum difference of about 8 percent between the two codes. The contour maps shown in Figure 1 for the two codes are very similar. The primary difference is the behavior of the contours at the upper and lower (north and south) edge of the model. Contours from the MODFLOW run are perpendicular to the boundary, while WinFlow generated contours hit the boundary at an angle. This happens because MODFLOW treats the edge of the model as a no-flow or impermeable boundary forcing the contours to hit the boundary at right angles. WinFlow, on the other hand, assumes that the aquifer is infinite without any no-flow or impermeable boundaries.

A second test case was simulated by both WinFlow and MODFLOW in which no-flow boundaries were simulated with WinFlow. The northern and southern no-flow boundaries were reproduced in WinFlow using image wells. Two image wells were placed at coordinates (x=5000, y=15000) and (x=5000, y=-5000). Each image well pumped 100,000 ft^3/d. Contour maps for the second test case are shown in Figure 2. Now the WinFlow contours also strike the boundary at close to right angles. The maximum difference between WinFlow and MODFLOW for the second case is 0.39 feet, with a mean difference of 0.11 feet. This represents a significant improvement over the first test case. The maximum difference is 3.7 percent in this case.

The two test cases presented for the benchmark between WinFlow and MODFLOW show that both codes calculate similar head fields for the same problem. Even though the method of solution is different (analytical vs. numerical), each software package gives similar results. These comparisons provide the user with confidence that WinFlow is solving the ground-water flow equations properly.

MODFLOW



WinFlow

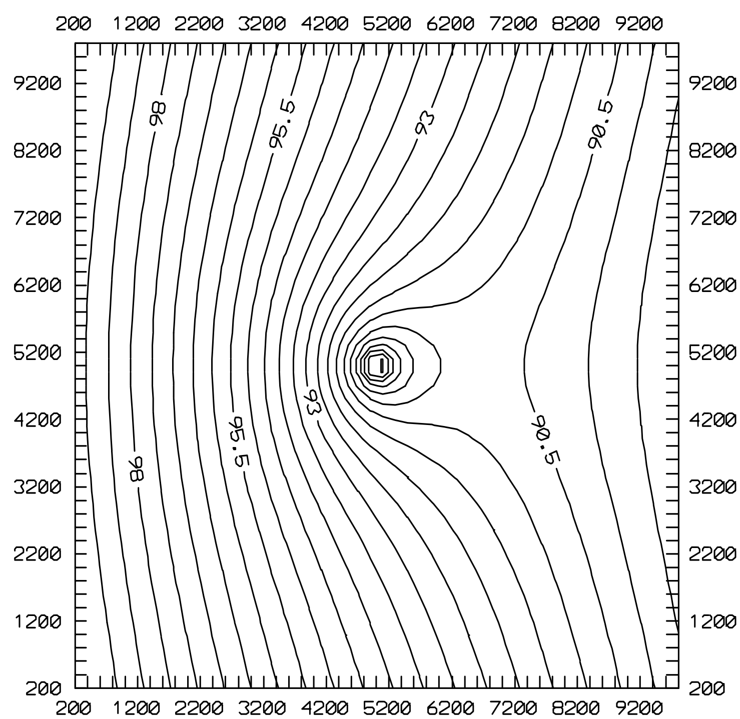
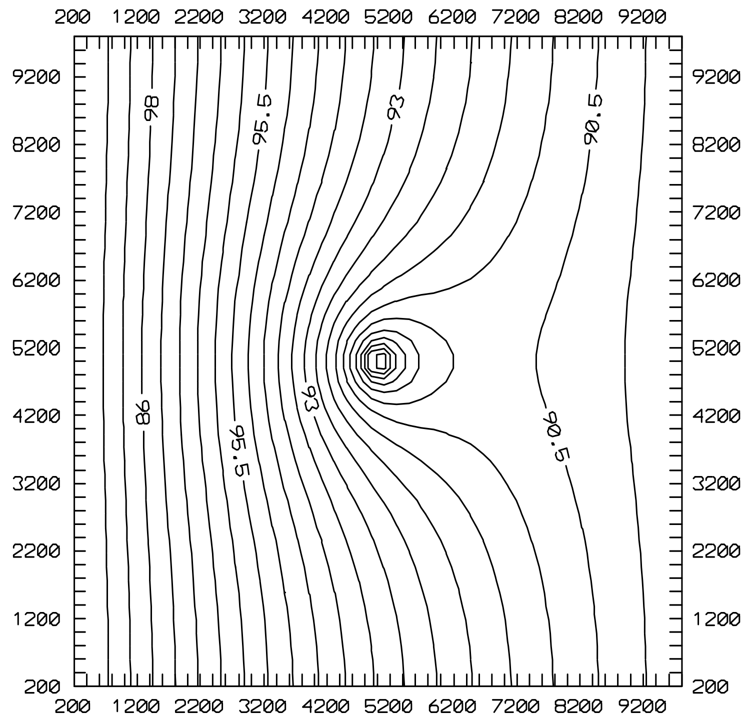


Figure 1. Comparison between WinFlow and MODFLOW for Test Case 1.

MODFLOW



WinFlow with Image Wells

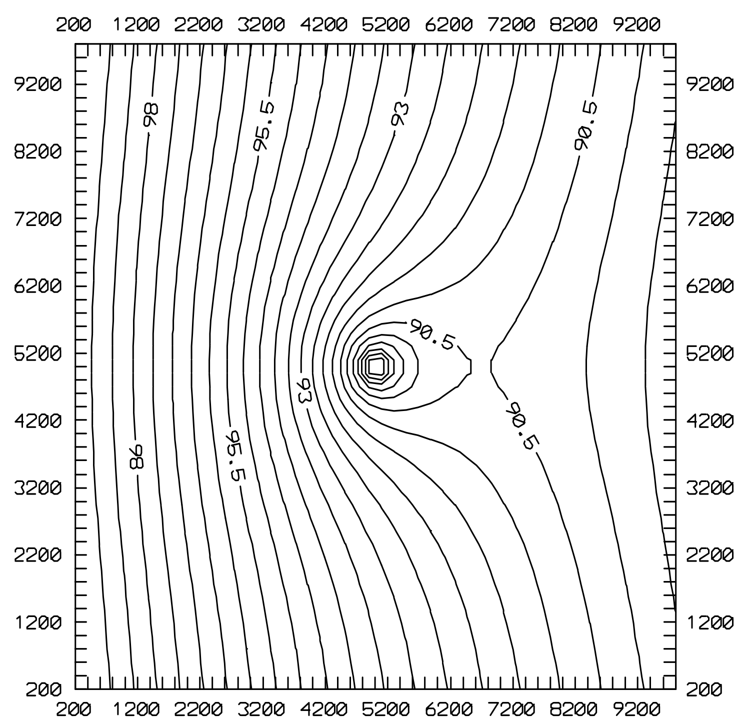


Figure 2. Comparison between WinFlow and MODFLOW for Test Case 2.

Transient Model

Three sets of verification problems are presented for the transient analytical functions used in WinFlow. In the first problem, drawdown is computed for a single well. In the second case, a uniform regional gradient is added to the problem. In each of the first two test cases, WinFlow calculations are compared to those performed with a calculator. The final test presents tables of the Theis (1935) and Hantush and Jacob (1955) well functions for comparison with published tables.

Case 1: Drawdown from a Single Well

The drawdown due to a single pumping well may be computed for any point in an aquifer using the following equation (Theis 1935):

$$s = \frac{Q}{4\pi T} W(u)$$

where

s	= drawdown [L],
Q	= well pumping rate [L^3/T],
T	= transmissivity [L^2/T],
u	= (r^2 S)/(4 T t),
r	= distance between well and calculation point,
S	= storage coefficient [dimensionless],
t	= time after start of pumping [T],
W(u)	= Theis well function.

In this example problem, we will choose the values of the parameters so that calculation is straightforward on a hand calculator and published tables of the Theis well function. The following parameters are used for Case 1:

T = 2500 ft^2/d

S = 0.01

t = 1.0 d

Q = 10,000 ft^3/d

WinFlow computed the same values of drawdown (s) as those computed using a calculator to four significant figures. The results of Case 1 are presented in Table 3.

Table 3 Comparison between WinFlow and calculator results for transient case 1.

Radius (ft)	u	W(u)	s (ft)	s (WinFlow)
1.0	10^{-6}	13.24	4.214	4.214
10.0	10^{-4}	8.633	2.748	2.748
20.0	4×10^{-4}	7.247	2.307	2.307
30.0	9×10^{-4}	6.437	2.049	2.049
40.0	1.6×10^{-3}	5.862	1.866	1.866
50.0	2.5×10^{-3}	5.417	1.724	1.724
60.0	3.6×10^{-3}	5.053	1.608	1.608
70.0	4.9×10^{-3}	4.746	1.511	1.511
80.0	6.4×10^{-3}	4.481	1.426	1.426
90.0	8.1×10^{-3}	4.247	1.352	1.352
100.0	0.01	4.038	1.285	1.285

Case 2: Drawdown from a Single Well in a Uniform Flow Field

The same parameters used in Case 1 above will be used in Case 2 and a uniform regional gradient will be added. Assume that the gradient is 0.001 ft/ft, with a reference head of 100 ft at the well. Because the transient model does not assume that the reference head is constant, the reference head may be specified anywhere (even at the well). We will also assume that the origin of the coordinate system (x=0, y=0) is at the well center.

The equation for a single well in a uniform flow field under transient conditions was given in the last chapter as

$$\phi(x,y,t) = C - G(x\cos\alpha + y\sin\alpha) - s$$

where

ϕ	= head [L],
G	= regional gradient [L/L],
α	= angle between regional gradient and x-axis,
(x,y)	= coordinates of calculation point,
t	= time since start of pumping,
s	= drawdown from well,
C	= constant.

The constant, C, is equal to the reference head in this case.

The heads computed by WinFlow and using a hand calculator are presented in Table 4. Again, WinFlow results and the calculator results are identical to six significant figures.

Table 4 Comparison between WinFlow and hand calculations for transient case 2.

X	Y	ϕ	ϕ (WinFlow)
1.0	0.0	95.786	95.786
10.0	0.0	97.152	97.152
20.0	0.0	97.493	97.493
30.0	0.0	97.651	97.651
40.0	0.0	97.734	97.734
50.0	0.0	97.776	97.776
60.0	0.0	97.792	97.792
70.0	0.0	97.789	97.789
80.0	0.0	97.774	97.774
90.0	0.0	97.748	97.748
100.0	0.0	97.715	97.715

Case 3: Calculation of Well Function Tables

The first two transient test cases tested the ability of WinFlow to compute drawdown with and without a regional gradient. These tests illustrated that WinFlow internal drawdown calculations are properly implemented. A further test of the software is calculation of well function tables, which tests WinFlow's ability to accurately compute drawdown over a wide range of conditions.

WinFlow uses two transient analytical functions: (1) the Theis (1935) equation for confined aquifers, and (2) the Hantush and Jacob (1955) equation for semi-confined (or leaky) aquifers. Values of the Theis well function, W(u), were computed using the numerical routines in WinFlow for a wide range of values of u. These calculations are shown in Table 5. These values can be compared to any published values, although the format of the table is identical to that published by Kruseman and deRidder (1990) in Annex 3.1, page 294. Table 5 and Annex 3.1 (Kruseman and deRidder 1990) are identical, illustrating that WinFlow can calculate the Theis well function accurately over a wide range in u.

Similarly, the Hantush and Jacob (1955) well function, W(u,r/L), was computed using the routines in WinFlow for a range of u and r/L values. These are shown in Tables 6, 7, and 8. Kruseman and deRidder (1990) have published similar tables in Annex 4.2 (pages 298 and 299). The Kruseman and deRidder (1990) tables and Tables 6, 7, and 8 are identical, confirming that WinFlow accurately computes values for the Hantush and Jacob leaky well function.

Table 5 Theis well function, W(u), computed using routines in WinFlow.

u	W(u)	$W(u\ 10^{-1})$	$W(u\ 10^{-2})$	$W(u\ 10^{-3})$	$W(u\ 10^{-4})$	$W(u\ 10^{-5})$	$W(u\ 10^{-6})$	$W(u\ 10^{-7})$	$W(u\ 10^{-8})$	$W(u\ 10^{-9})$	$W(u\ 10^{-10})$
1.0	2.194e- 01	1.823e+00	4.038e+00	6.332e+00	8.633e+00	1.094e+01	1.324e+01	1.554e+01	1.784e+01	2.015e+01	2.245e+01
1.2	1.584e- 01	1.660e+00	3.858e+00	6.149e+00	8.451e+00	1.075e+01	1.306e+01	1.536e+01	1.766e+01	1.996e+01	2.227e+01
1.5	1.000e- 01	1.464e+00	3.637e+00	5.927e+00	8.228e+00	1.053e+01	1.283e+01	1.514e+01	1.744e+01	1.974e+01	2.204e+01
2.0	4.890e- 02	1.223e+00	3.355e+00	5.639e+00	7.940e+00	1.024e+01	1.255e+01	1.485e+01	1.715e+01	1.945e+01	2.176e+01
2.5	2.491e- 02	1.044e+00	3.137e+00	5.417e+00	7.717e+00	1.002e+01	1.232e+01	1.462e+01	1.693e+01	1.923e+01	2.153e+01
3.0	1.305e- 02	9.057e- 01	2.959e+00	5.235e+00	7.535e+00	9.837e+00	1.214e+01	1.444e+01	1.674e+01	1.905e+01	2.135e+01
3.5	6.970e- 03	7.942e- 01	2.810e+00	5.081e+00	7.381e+00	9.683e+00	1.199e+01	1.429e+01	1.659e+01	1.889e+01	2.120e+01
4.0	3.779e- 03	7.024e- 01	2.681e+00	4.948e+00	7.247e+00	9.549e+00	1.185e+01	1.415e+01	1.646e+01	1.876e+01	2.106e+01
4.5	2.073e- 03	6.253e- 01	2.568e+00	4.831e+00	7.129e+00	9.432e+00	1.173e+01	1.404e+01	1.634e+01	1.864e+01	2.094e+01
5.0	1.148e- 03	5.598e- 01	2.468e+00	4.726e+00	7.024e+00	9.326e+00	1.163e+01	1.393e+01	1.623e+01	1.854e+01	2.084e+01
6.0	3.601e- 04	4.544e- 01	2.295e+00	4.545e+00	6.842e+00	9.144e+00	1.145e+01	1.375e+01	1.605e+01	1.835e+01	2.066e+01
7.0	1.155e- 04	3.738e- 01	2.151e+00	4.392e+00	6.688e+00	8.990e+00	1.129e+01	1.359e+01	1.590e+01	1.820e+01	2.050e+01
8.0	3.767e- 05	3.106e- 01	2.027e+00	4.259e+00	6.554e+00	8.856e+00	1.116e+01	1.346e+01	1.576e+01	1.807e+01	2.037e+01
9.0	1.245e- 05	2.602e- 01	1.919e+00	4.142e+00	6.437e+00	8.739e+00	1.104e+01	1.334e+01	1.565e+01	1.795e+01	2.025e+01

Table 6 Hantush well function, W(u,r/L), computed using routines in WinFlow.

u	r/L = 0	0.005	0.01	0.02	0.03	0.04	0.05	0.06	0.07	0.08	0.09
1.0e- 06	1.32e+01	1.08e+01	9.44e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
2.0e- 06	1.25e+01	1.08e+01	9.44e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
4.0e- 06	1.19e+01	1.07e+01	9.44e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
6.0e- 06	1.14e+01	1.06e+01	9.44e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
8.0e- 06	1.12e+01	1.05e+01	9.43e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
1.0e- 05	1.09e+01	1.04e+01	9.42e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
2.0e- 05	1.02e+01	9.95e+00	9.30e+00	8.06e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
4.0e- 05	9.55e+00	9.40e+00	9.01e+00	8.03e+00	7.25e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
6.0e- 05	9.14e+00	9.04e+00	8.77e+00	7.98e+00	7.24e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
8.0e- 05	8.86e+00	8.78e+00	8.57e+00	7.91e+00	7.23e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
1.0e- 04	8.63e+00	8.57e+00	8.40e+00	7.84e+00	7.21e+00	6.67e+00	6.23e+00	5.87e+00	5.56e+00	5.29e+00	5.06e+00
2.0e- 04	7.94e+00	7.91e+00	7.82e+00	7.50e+00	7.07e+00	6.62e+00	6.22e+00	5.86e+00	5.56e+00	5.29e+00	5.06e+00
4.0e- 04	7.25e+00	7.23e+00	7.19e+00	7.01e+00	6.76e+00	6.45e+00	6.14e+00	5.83e+00	5.55e+00	5.29e+00	5.06e+00
6.0e- 04	6.84e+00	6.83e+00	6.80e+00	6.68e+00	6.50e+00	6.27e+00	6.02e+00	5.77e+00	5.51e+00	5.27e+00	5.05e+00
8.0e- 04	6.55e+00	6.55e+00	6.52e+00	6.43e+00	6.29e+00	6.11e+00	5.91e+00	5.69e+00	5.46e+00	5.25e+00	5.04e+00
1.0e- 03	6.33e+00	6.33e+00	6.31e+00	6.23e+00	6.12e+00	5.97e+00	5.80e+00	5.61e+00	5.41e+00	5.21e+00	5.01e+00
2.0e- 03	5.64e+00	5.64e+00	5.63e+00	5.59e+00	5.53e+00	5.45e+00	5.35e+00	5.24e+00	5.12e+00	4.98e+00	4.85e+00
4.0e- 03	4.95e+00	4.95e+00	4.94e+00	4.92e+00	4.89e+00	4.85e+00	4.80e+00	4.74e+00	4.67e+00	4.59e+00	4.51e+00
6.0e- 03	4.54e+00	4.54e+00	4.54e+00	4.53e+00	4.51e+00	4.48e+00	4.45e+00	4.41e+00	4.36e+00	4.30e+00	4.24e+00
8.0e- 03	4.26e+00	4.26e+00	4.26e+00	4.25e+00	4.23e+00	4.21e+00	4.19e+00	4.15e+00	4.12e+00	4.08e+00	4.03e+00
1.0e- 02	4.04e+00	4.04e+00	4.04e+00	4.03e+00	4.02e+00	4.00e+00	3.98e+00	3.95e+00	3.93e+00	3.89e+00	3.86e+00
2.0e- 02	3.35e+00	3.35e+00	3.35e+00	3.35e+00	3.34e+00	3.34e+00	3.33e+00	3.31e+00	3.30e+00	3.28e+00	3.26e+00
4.0e- 02	2.68e+00	2.68e+00	2.68e+00	2.68e+00	2.68e+00	2.67e+00	2.67e+00	2.66e+00	2.66e+00	2.65e+00	2.64e+00
6.0e- 02	2.30e+00	2.30e+00	2.29e+00	2.29e+00	2.29e+00	2.29e+00	2.29e+00	2.28e+00	2.28e+00	2.27e+00	2.27e+00
8.0e- 02	2.03e+00	2.03e+00	2.03e+00	2.03e+00	2.02e+00	2.02e+00	2.02e+00	2.02e+00	2.02e+00	2.01e+00	2.01e+00
1.0e- 01	1.82e+00	1.82e+00	1.82e+00	1.82e+00	1.82e+00	1.82e+00	1.82e+00	1.82e+00	1.81e+00	1.81e+00	1.81e+00
2.0e- 01	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00	1.22e+00
4.0e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.02e- 01	7.01e- 01	7.01e- 01	7.00e- 01
6.0e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.54e- 01	4.53e- 01
8.0e- 01	3.11e- 01	3.11e- 01	3.11e- 01	3.11e- 01	3.11e- 01	3.10e- 01	3.10e- 01	3.10e- 01	3.10e- 01	3.10e- 01	3.10e- 01

Table 7 Hantush well function, W(u,r/L), computed using routines in WinFlow.

u	r/L = 0	0.1	0.2	0.3	0.4	0.6	0.8
1.0e- 04	8.63e+00	4.85e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
2.0e- 04	7.94e+00	4.85e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
4.0e- 04	7.25e+00	4.85e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
6.0e- 04	6.84e+00	4.85e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
8.0e- 04	6.55e+00	4.84e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
1.0e- 03	6.33e+00	4.83e+00	3.51e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
2.0e- 03	5.64e+00	4.71e+00	3.50e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
4.0e- 03	4.95e+00	4.42e+00	3.48e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
6.0e- 03	4.54e+00	4.18e+00	3.43e+00	2.74e+00	2.23e+00	1.56e+00	1.13e+00
8.0e- 03	4.26e+00	3.98e+00	3.36e+00	2.73e+00	2.23e+00	1.56e+00	1.13e+00
1.0e- 02	4.04e+00	3.82e+00	3.29e+00	2.71e+00	2.23e+00	1.56e+00	1.13e+00
2.0e- 02	3.35e+00	3.24e+00	2.95e+00	2.57e+00	2.18e+00	1.55e+00	1.13e+00
4.0e- 02	2.68e+00	2.63e+00	2.48e+00	2.27e+00	2.02e+00	1.52e+00	1.13e+00
6.0e- 02	2.30e+00	2.26e+00	2.17e+00	2.02e+00	1.85e+00	1.46e+00	1.11e+00
8.0e- 02	2.03e+00	2.00e+00	1.94e+00	1.83e+00	1.69e+00	1.39e+00	1.08e+00
1.0e- 01	1.82e+00	1.80e+00	1.75e+00	1.67e+00	1.56e+00	1.31e+00	1.05e+00
2.0e- 01	1.22e+00	1.22e+00	1.19e+00	1.16e+00	1.11e+00	9.96e- 01	8.58e- 01
4.0e- 01	7.02e- 01	7.00e- 01	6.93e- 01	6.81e- 01	6.65e- 01	6.21e- 01	5.65e- 01
6.0e- 01	4.54e- 01	4.53e- 01	4.50e- 01	4.44e- 01	4.36e- 01	4.15e- 01	3.87e- 01
8.0e- 01	3.11e- 01	3.10e- 01	3.08e- 01	3.05e- 01	3.01e- 01	2.89e- 01	2.73e- 01
1.0e+00	2.19e- 01	2.19e- 01	2.18e- 01	2.16e- 01	2.14e- 01	2.06e- 01	1.97e- 01
2.0e+00	4.89e- 02	4.89e- 02	4.87e- 02	4.85e- 02	4.82e- 02	4.72e- 02	4.60e- 02

Table 8 Hantush well function, W(u,r/L), computed using routines in WinFlow.

u	r/L = 0	1.0	2.0	3.0	4.0	5.0	6.0
1.0e- 02	4.04e+00	8.42e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
2.0e- 02	3.35e+00	8.42e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
4.0e- 02	2.68e+00	8.42e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
6.0e- 02	2.30e+00	8.39e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
8.0e- 02	2.03e+00	8.32e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
1.0e- 01	1.82e+00	8.19e- 01	2.28e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
2.0e- 01	1.22e+00	7.15e- 01	2.27e- 01	6.95e- 02	2.23e- 02	7.38e- 03	2.49e- 03
4.0e- 01	7.02e- 01	5.02e- 01	2.10e- 01	6.91e- 02	2.23e- 02	7.38e- 03	2.49e- 03
6.0e- 01	4.54e- 01	3.54e- 01	1.77e- 01	6.64e- 02	2.22e- 02	7.38e- 03	2.49e- 03
8.0e- 01	3.11e- 01	2.54e- 01	1.44e- 01	6.07e- 02	2.17e- 02	7.36e- 03	2.49e- 03
1.0e+00	2.19e- 01	1.85e- 01	1.14e- 01	5.34e- 02	2.07e- 02	7.27e- 03	2.49e- 03
2.0e+00	4.89e- 02	4.44e- 02	3.34e- 02	2.10e- 02	1.12e- 02	5.13e- 03	2.10e- 03
4.0e+00	3.78e- 03	3.58e- 03	3.06e- 03	2.35e- 03	1.63e- 03	1.03e- 03	5.86e- 04

Transport Model

Introduction

The finite-element transport model in WinTran is verified through comparison with an analytical solution from Wexler (1992) and with another finite-element transport model called SEFTRAN (Huyakorn et al., 1984). The Wexler analytical solution models transport of a dissolved contaminant from a point source in a two-dimensional uniform flow field. Six test cases were investigated with SEFTRAN for the three different source configurations (injection well, pond, and linesink) in both uniform flow and in non-uniform flow fields.

Comparison to an Analytical Solution

Wexler (1992) presents a series of analytical solutions to the partial differential equations of dissolved contaminant transport in porous media. WinTran was compared to the solution for a continuous point source in an aquifer of infinite extent (see page 26 of Wexler, 1992). The analytical solution was implemented by Wexler in a FORTRAN program called POINT2.

The data for the test problem are presented in Table 1. Concentration is plotted versus time at two locations downgradient of the source for both WinTran and SEFTRAN (see Figure 1). These curves show that WinTran results are virtually identical to those of the analytical solution. Contours for both WinTran results and POINT2 results are shown in Figure 2. Again, these contours are almost identical for the two solutions. The largest difference is at the source, where WinTran slightly underpredicts the source concentration. This is probably caused by dilution of the source concentration in the finite-element cell. The majority of the plume, however, matches quite well between WinTran and POINT2.

Comparison of WinTran to an analytical solution confirms that the basic transport model has been coded properly. The analytical solution, however, assumes that the flow field is uniform and the source is a single point and continuous over time. The next section presents a series of tests that illustrate that WinTran performs properly for more complex scenarios.

Table 1. Model Parameters for the Analytical Solution Comparison

Parameter	Value
Hydraulic conductivity	100 ft/d
Top Elevation	-75 ft
Bottom Elevation	-100 ft
Porosity	0.2
Hydraulic Gradient	0.01 to the East
Groundwater Velocity	5 ft/d
Longitudinal Dispersivity	30 ft
Transverse Dispersivity	3 ft
Retardation Coefficient	1
X coordinate of source	212.32 ft
Y coordinate of source	230.87 ft
Source fluid flow rate	-1 ft^3/d
Source concentration	100
Number of X nodes	70
Number of Y nodes	70
Minimum X coordinate	50.0 ft
Minimum Y coordinate	50.0 ft
Nodal Spacing in X	8.116 ft
Nodal Spacing in Y	5.652 ft
Number of time steps	50
Minimum time step size	0.5 day
Maximum time step size	10 days
Time step multiplier	1.1
Final time value	280.569 days

Figure 1. Time-series comparison between WinTran and an analytical solution at two downgradient nodes

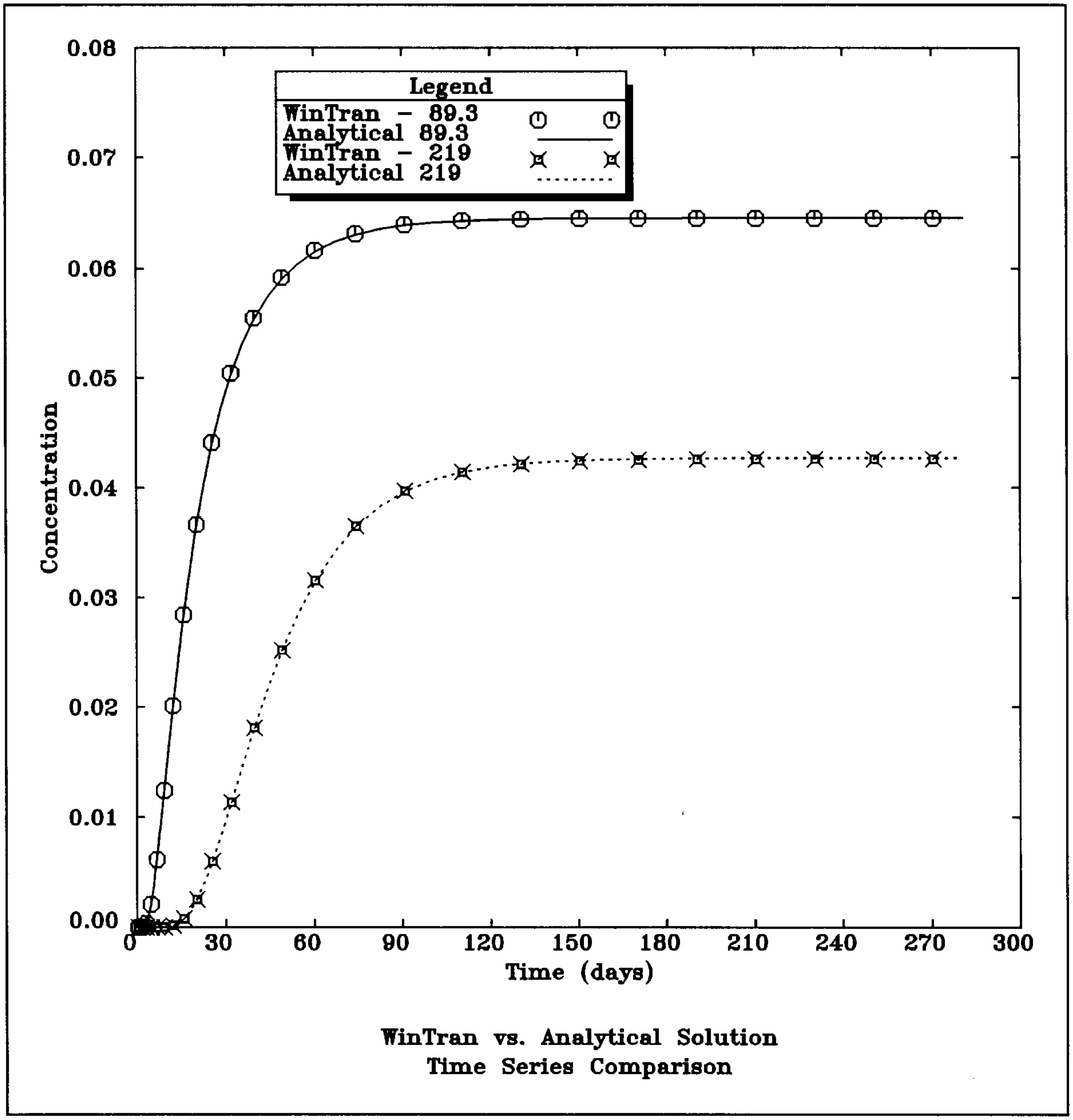
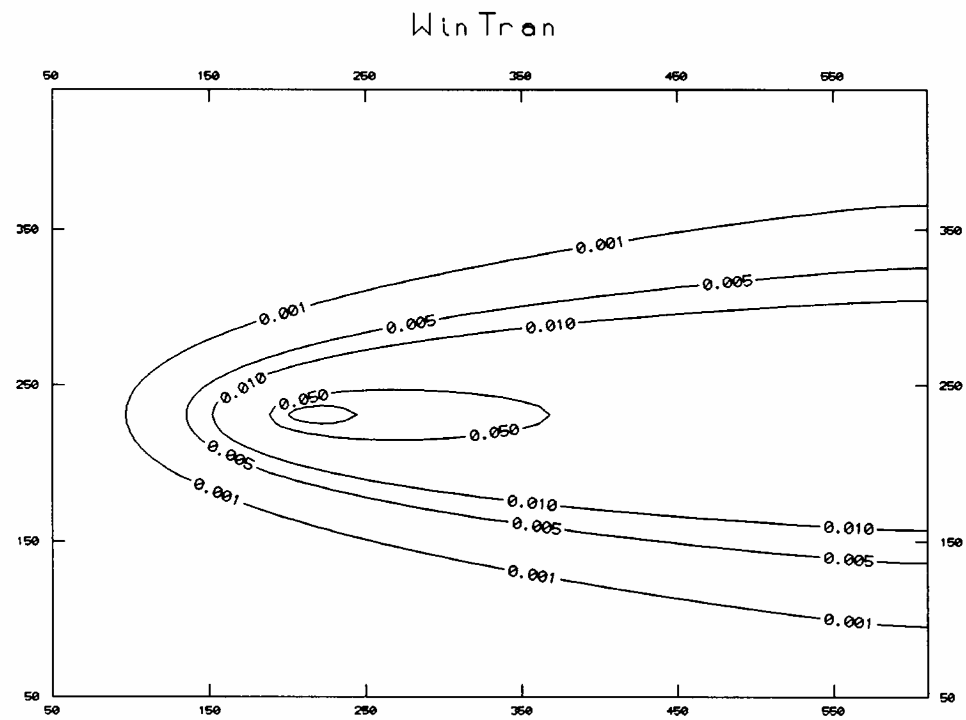
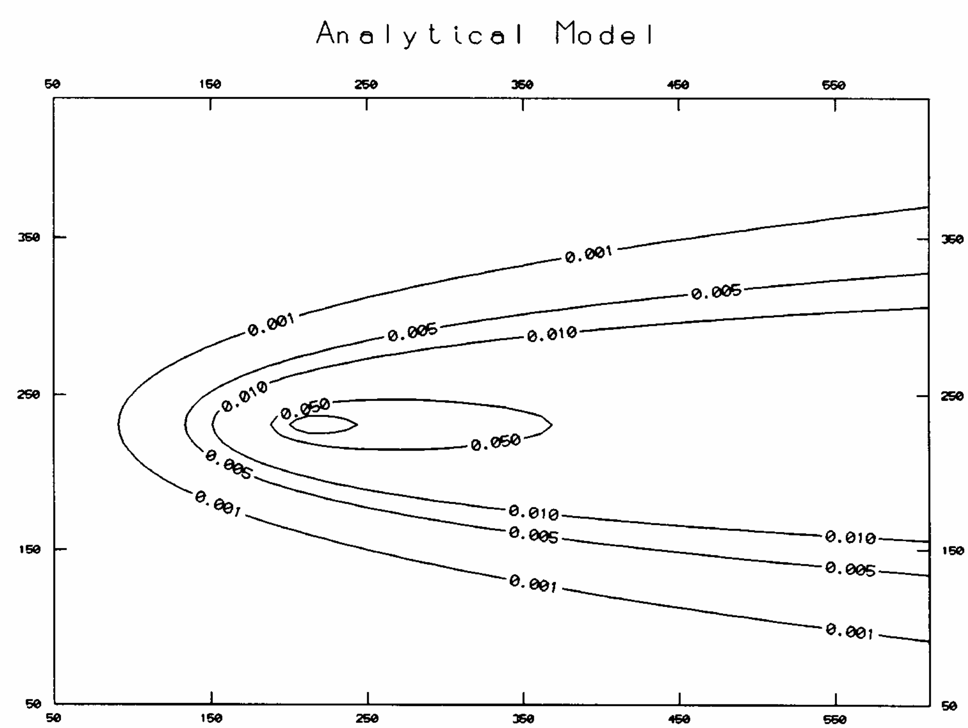


Figure 2. Concentration contours for WinTran and the analytical solution at time=260.569 days.



WinTran
50
150
250
350
450
550
350
250
150
50
0.001
0.005
0.010
0.050



Analytical Model
50
150
250
350
450
550
350
250
150
50
0.001
0.005
0.010
0.050

Benchmarking with SEFTRAN

SEFTRAN (Huyakorn et al., 1984) was chosen for the majority of testing because it uses the same finite-element techniques that are employed by WinTran. SEFTRAN also makes a good choice for benchmark testing because it has undergone a significant amount of testing at the International Ground Water Modeling Center (Huyakorn et al., 1984).

To facilitate this testing, a special option has been added to the WinTran Export menu allowing WinTran to create SEFTRAN data input files. Three files are created, (1) a SEFTRAN flow data set (always called FLOW.IN), (2) a SEFTRAN transport data set (you specify the name in the dialog), and (3) a velocity file with analytically-computed velocities (always called FLOW.VEL).

A series of six simulations were performed to test the three different source configurations (point source using an injection well, pond infiltration, and linesink injection). Each of the three source terms was tested in both a uniform flow field and a non-uniform flow field. The non-uniform flow field was produced by adding a pumping well downgradient from the source. The results for the six simulations are summarized in Table 2 and Table 2b. Data for the simulations are shown in Table 3.

The benchmark simulations are evaluated by presenting the following in Table 2: (1) maximum source concentration computed by WinTran and SEFTRAN, (2) the mean and maximum differences (errors) when SEFTRAN uses WinTran-computed velocities, (3) the mean and maximum differences when SEFTRAN uses SEFTRAN-computed velocities, and (4) mass balance errors for the two models. The source concentrations were scaled to a value of 1.0 in WinTran. The mass balance errors are in percent.

The mean and maximum differences between the two codes are very low for the case when each code uses velocities computed by WinTran. This tests the WinTran transport model because both codes are using the same velocity field. The tests illustrate that the transport model in WinTran is functioning properly for all cases. The mass balance error for each code is comparable for all cases and the source concentrations are accurate to the fourth decimal place.

The second set of errors (differences) presented in Table 2 are for SEFTRAN results computed using velocities computed by the SEFTRAN flow model. In the first set of differences described in the previous paragraph, the SEFTRAN transport model read velocity data computed by WinTran. The second set of comparisons, therefore, are used to evaluate the hybrid modeling approach. The results show that for uniform flow conditions, WinTran and SEFTRAN velocities produce virtually the same results. In a non-uniform flow field, however, the differences are larger. This indicates that the analytically-computed velocities are slightly in error.

Table 2b presents the differences between SEFTRAN and WinTran when velocities in WinTran are computed using finite elements (rather than the analytical model). In this case, the differences are very minor. Thus, for complex flow fields, you may want to consider using the finite-element flow model to compute velocities. You may select this option using the **Model->Flow Model Type** menu.

Figures 3 through 8 present concentration contour maps created by WinTran and SEFTRAN. These figures further substantiate that the two models are producing the same results.

Table 2. Comparison Between WinTran and SEFTRAN for Six Simulations.

Description	Maximum Conc. WinTran	Maximum Conc. Seftran	WinTran Velocities Mean Error	WinTran Velocities Maximum Error	Seftran Velocities Mean Error	Seftran Velocities Maximum Error	Mass Balance Error WinTran	Mass Balance Error Seftran
Test 1 Point Source Uniform Flow	1.0	1.000052	-1.1e-05	7.5e-05	3.8e-05	7.0e-05	0.0129	0.00082
Test 2 Pond Source Uniform Flow	1.0	1.00024	-4.2e-05	2.4e-04	4.9e-05	1.99e-04	0.00758	0.0069
Test 3 Line Source Uniform Flow	1.0	0.99992	1.66e-05	2.04e-04	1.47e-04	2.4e-03	0.00438	0.018
Test 4 Point Source Nonuniform Flow	1.0	1.00005	-9.8e-06	7.3e-05	7.5e-06	5.8e-03	0.2057	0.195
Test 5 Pond Source Nonuniform Flow	1.0	0.99996	7.5e-06	7.23e-05	2.0e-05	0.045	0.147	0.136
Test 6 Line Source Nonuniform Flow	1.0	0.99991	1.06e-05	1.4e-04	4.2e-05	0.025	0.056	0.046

Table 2b. Comparison Between WinTran (Using the Finite Element Flow Model) and SEFTRAN for the Nonuniform Flow Test Cases.

Description	Mean Error	Maximum Error	WinTran Mass Balance Error
Test 4	-6.33e-06	6.78e-05	0.145
Test 5	1.3e-06	1.4e-04	0.161
Test 6	2.6e-05	2.7e-04	0.20

Table 3. Model Parameters for the SEFTRAN Benckmarking

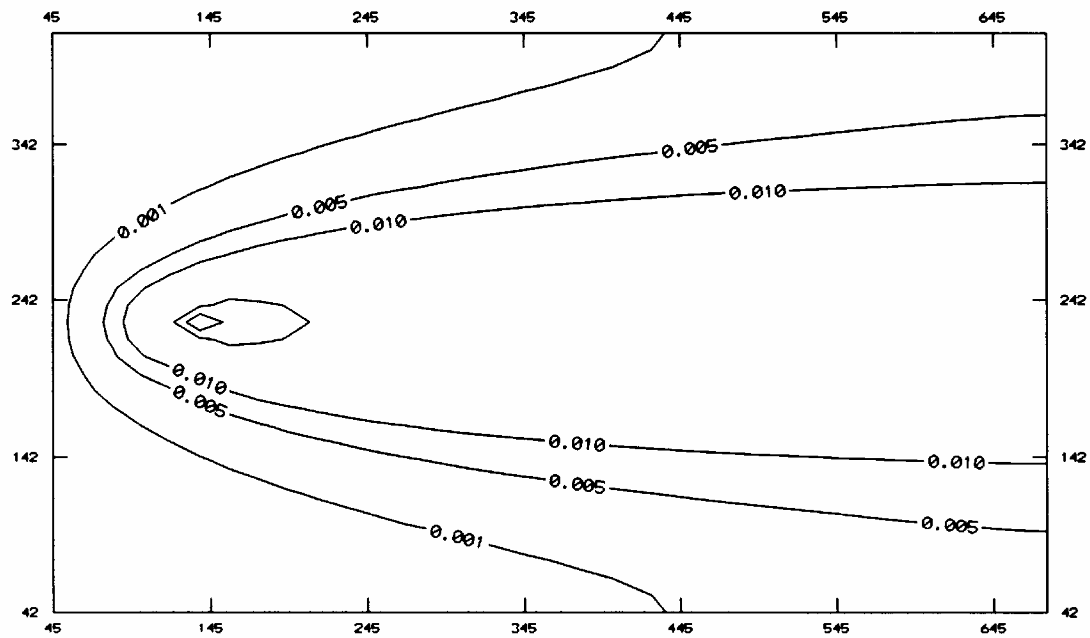
Parameter	Value
Hydraulic conductivity	100 ft/d
Top Elevation	100 ft
Bottom Elevation	0 ft
Reference Head	25 ft at (75,65)
Porosity	0.2
Hydraulic Gradient	0.01 to the East
Longitudinal Dispersivity	30 ft
Transverse Dispersivity	6 ft
Retardation Coefficient	1
Number of X nodes	35
Number of Y nodes	35
Minimum X coordinate	45.03 ft
Minimum Y coordinate	42.29 ft
Maximum X coordinate	678.81 ft
Maximum Y coordinate	413.66 ft
Number of time steps	30
Minimum time step size	1 day
Maximum time step size	100 days
Time step multiplier	1.2
Point Source Information (Simulation 1 and 4)	
Fluid Injection Rate	-1.0 ft^3/d
Concentration in fluid	100
Coordinates of Well (x,y)	(138.23,227.98)
Pumping Well Information (Simulations 4 through 6)	
Pumping Rate	10,000 ft^3/d
Coordinates of Well (x,y)	(604.25,315.36)

Table 3 (continued). Model Parameters for the SEFTRAN Benckmarking

<u>Linesink Source Information (Simulations 3 and 6)</u>	
Linesink Injection Rate	-1 ft^2/d
Concentration in fluid	100
Beginning Coordinates of line (x,y)	(145.27,275.11)
Ending Coordinates of line (x,y)	(143.65,167.59)
<u>Pond Source Information (Simulations 2 and 5)</u>	
Pond Infiltration Rate	0.0015 ft/d
Concentration in fluid	100
Pond Radius	24.68 ft
Coordinates of pond center (x,y)	(137.99,227.41)

Figure 3. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 1.

WinTran Results



SEFTRAN Results

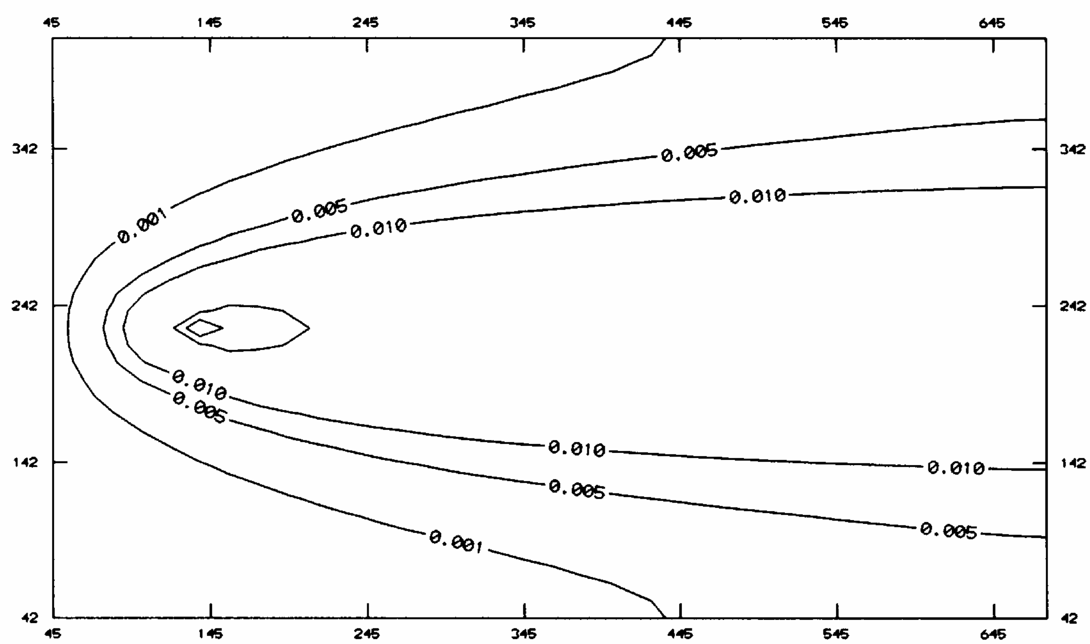
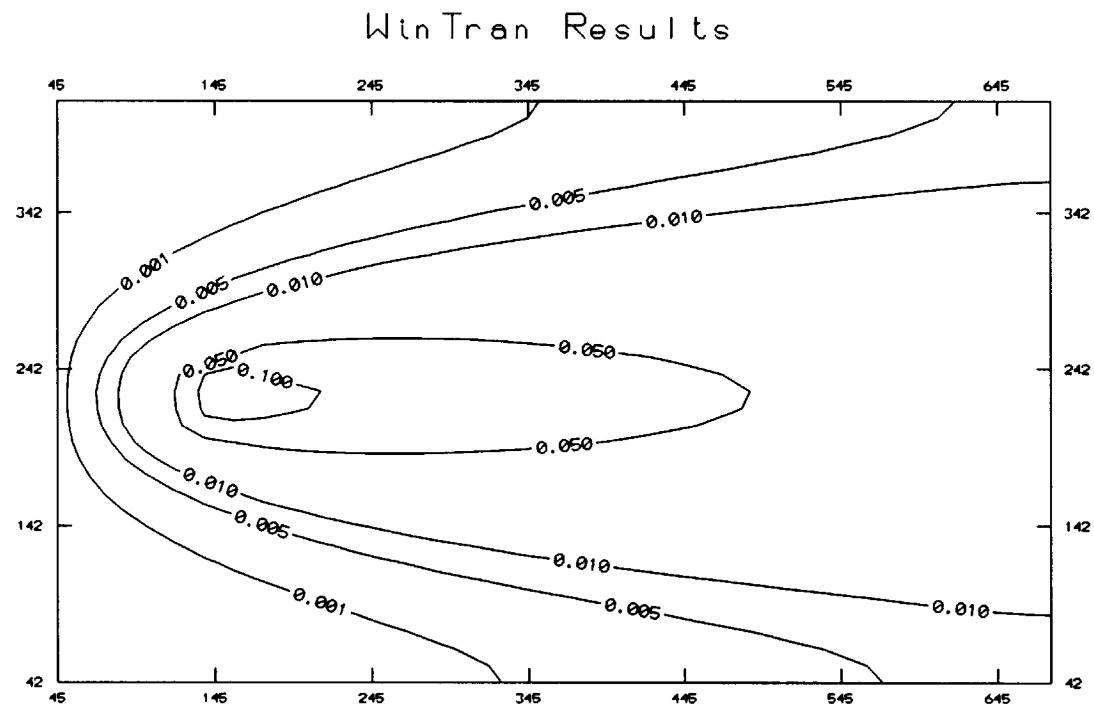


Figure 4. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 2.


WinTran Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010
0.050
0.100

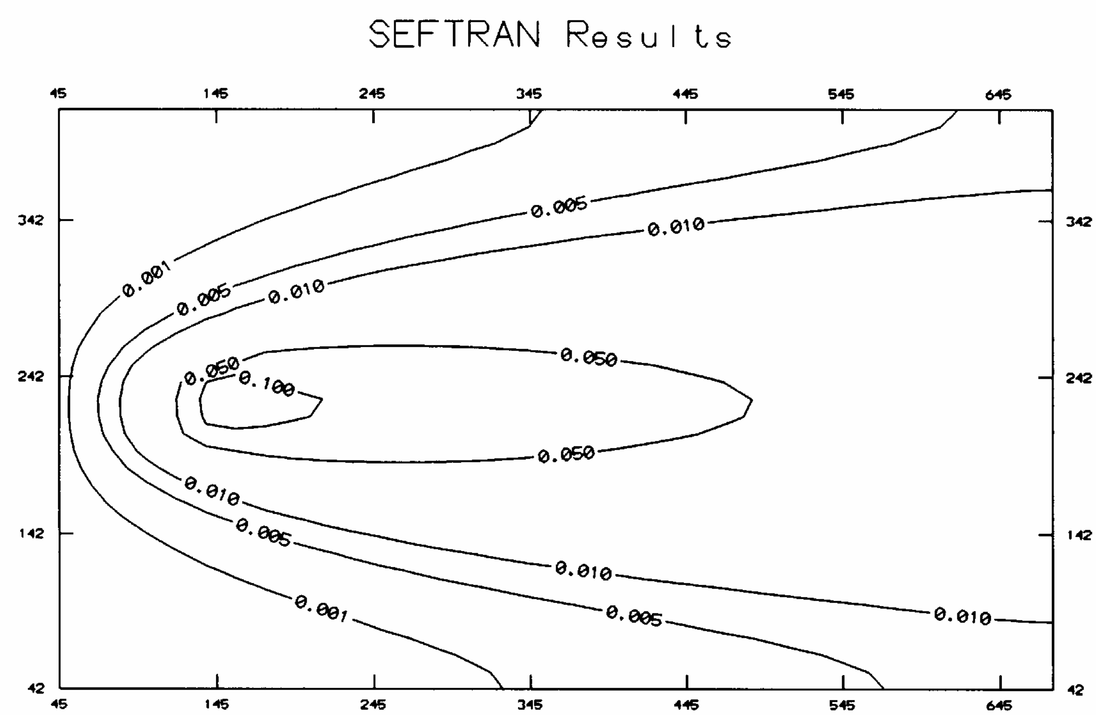
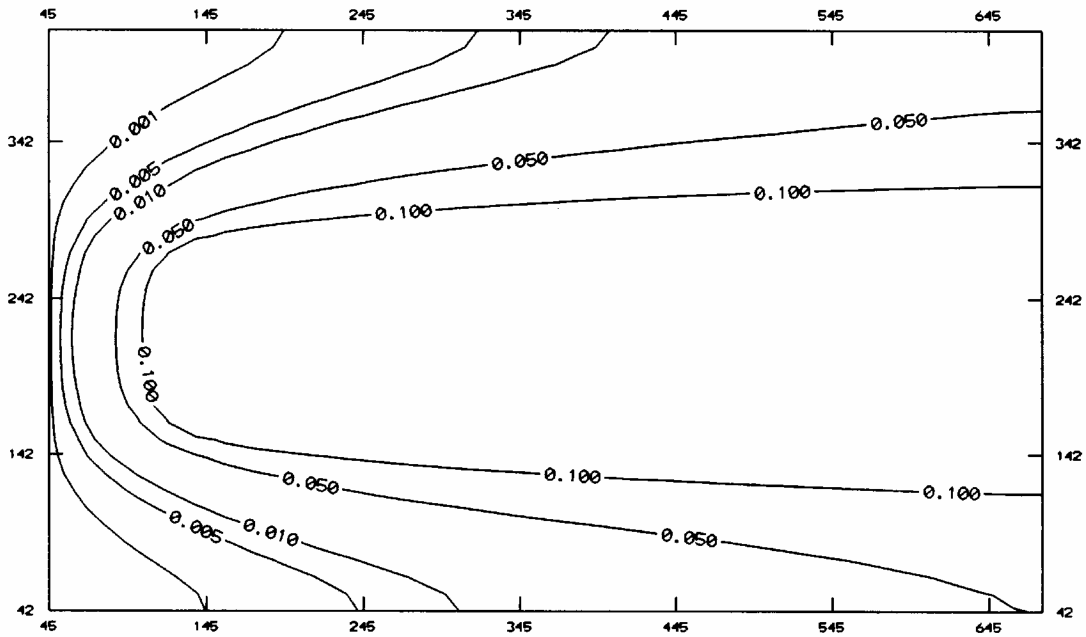

SEFTRAN Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010
0.050
0.100

Figure 5. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 3.

WinTran Results



SEFTRAN Results

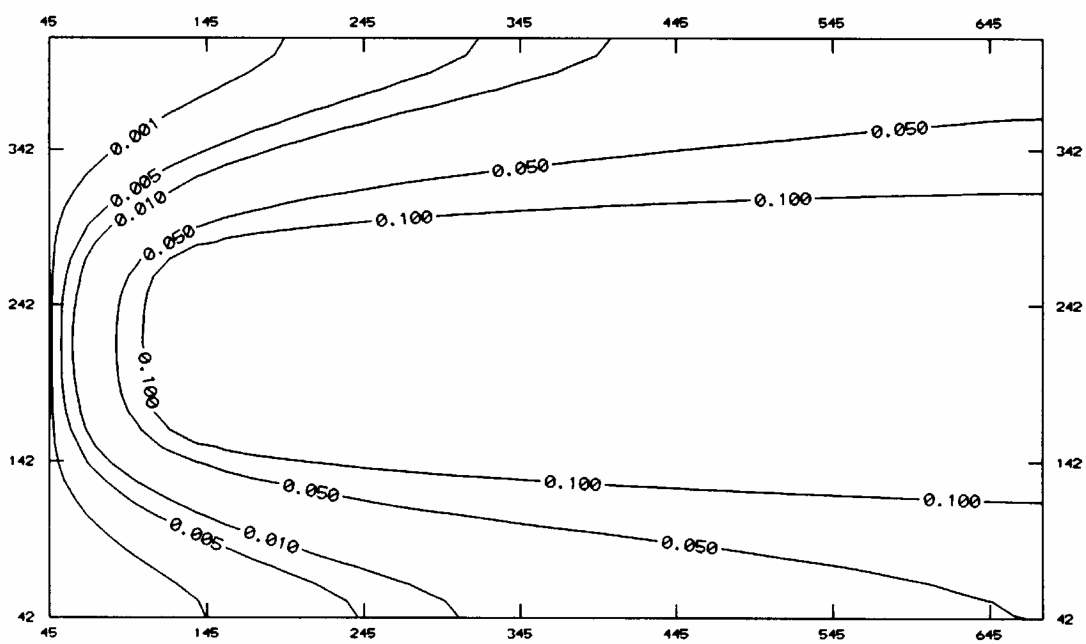
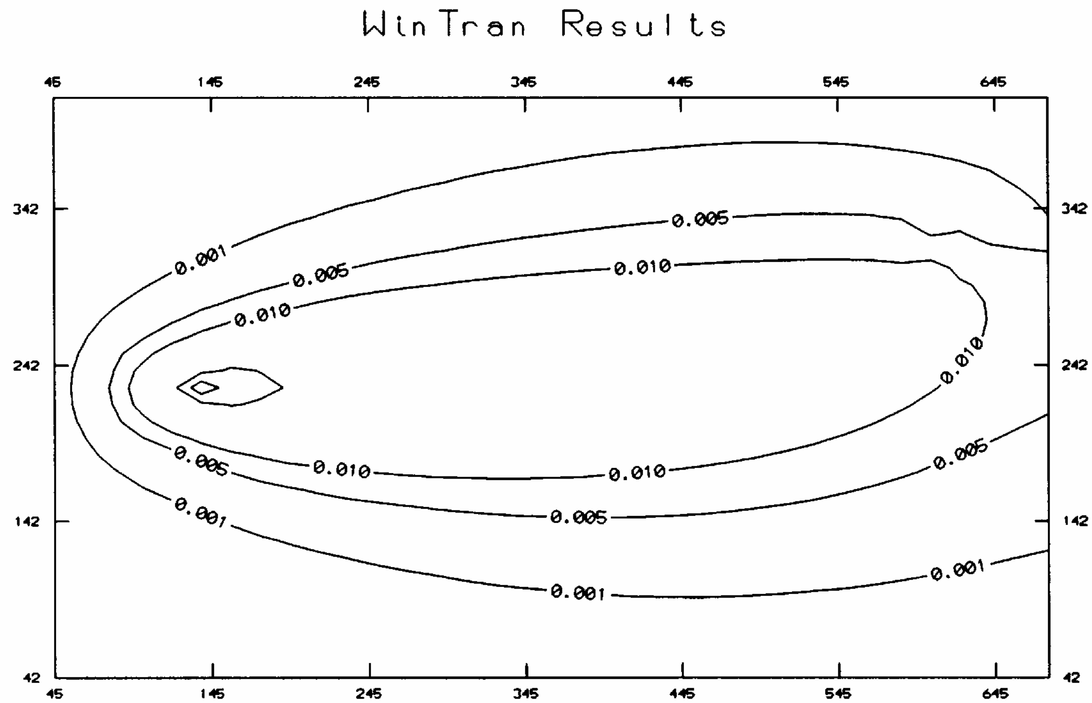


Figure 6. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 4.


WinTran Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010

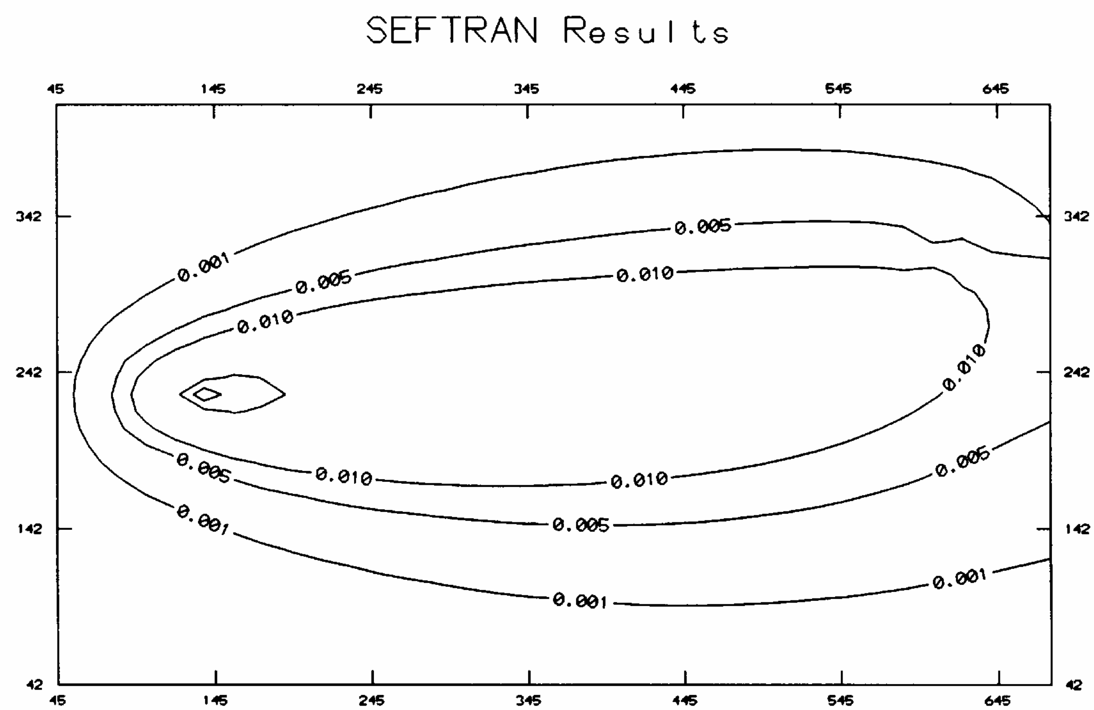
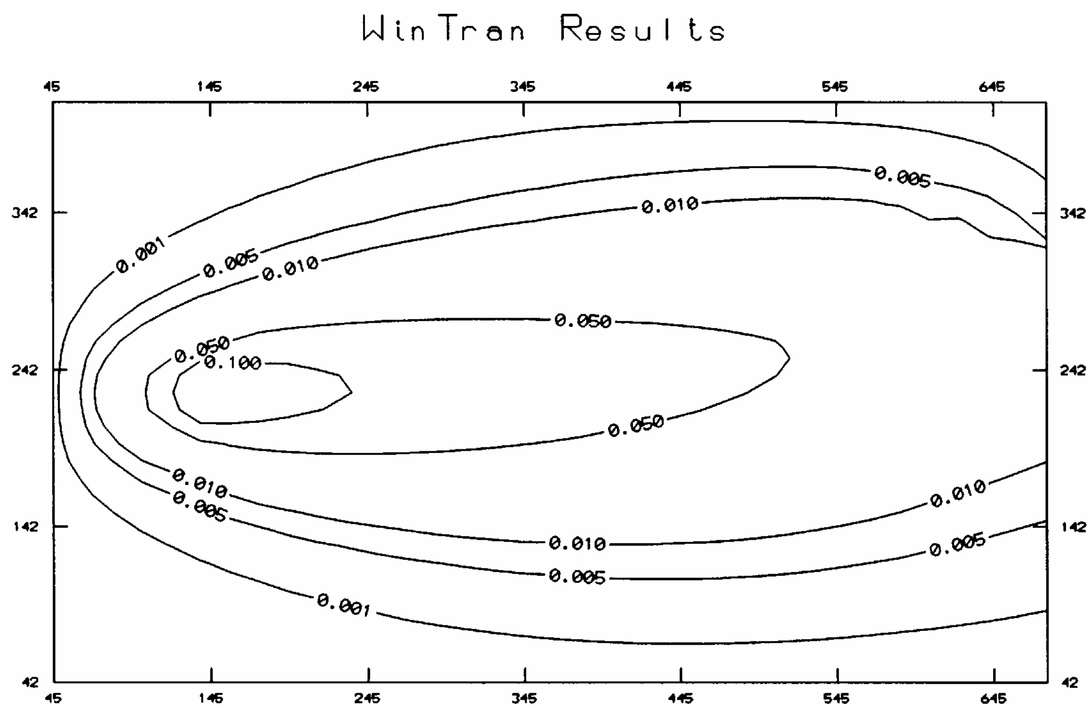

SEFTRAN Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010

Figure 7. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 5.


WinTran Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010
0.050
0.100

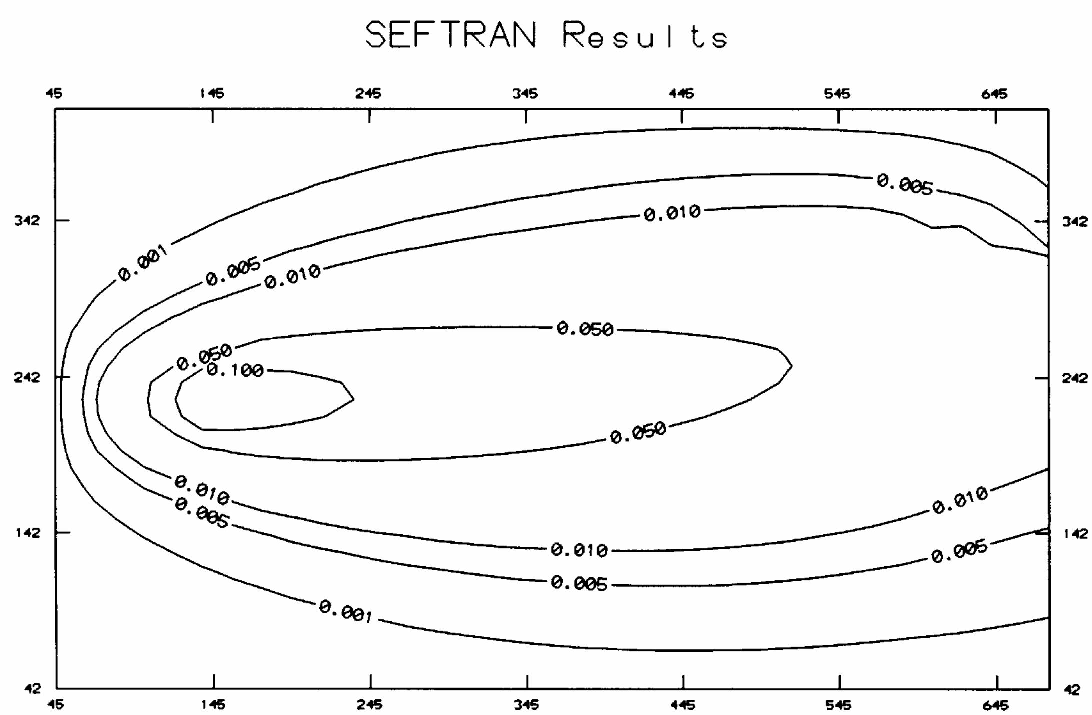
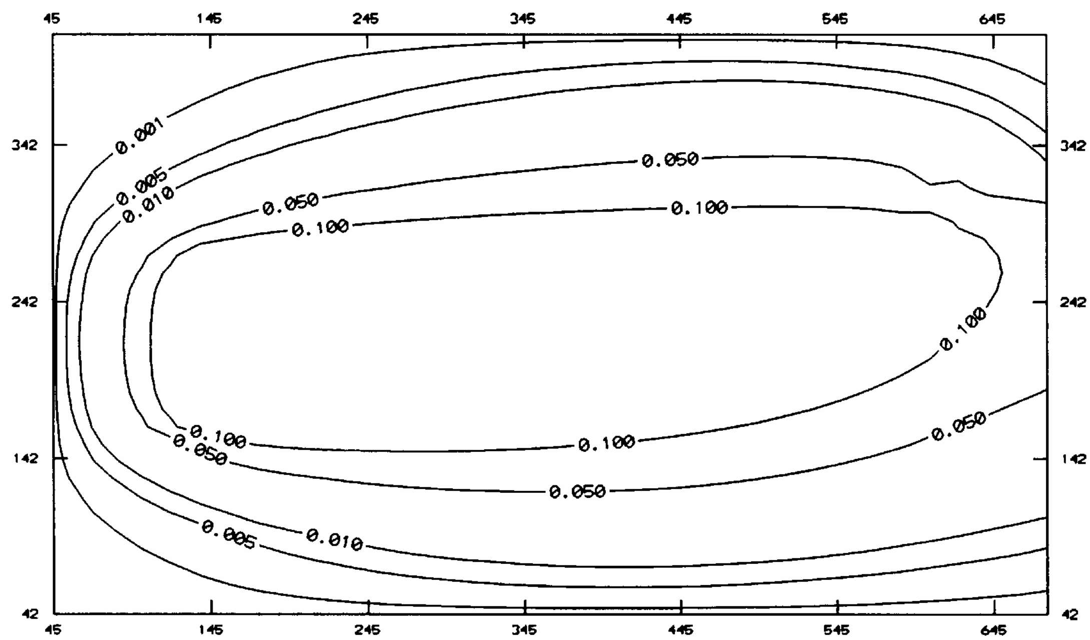
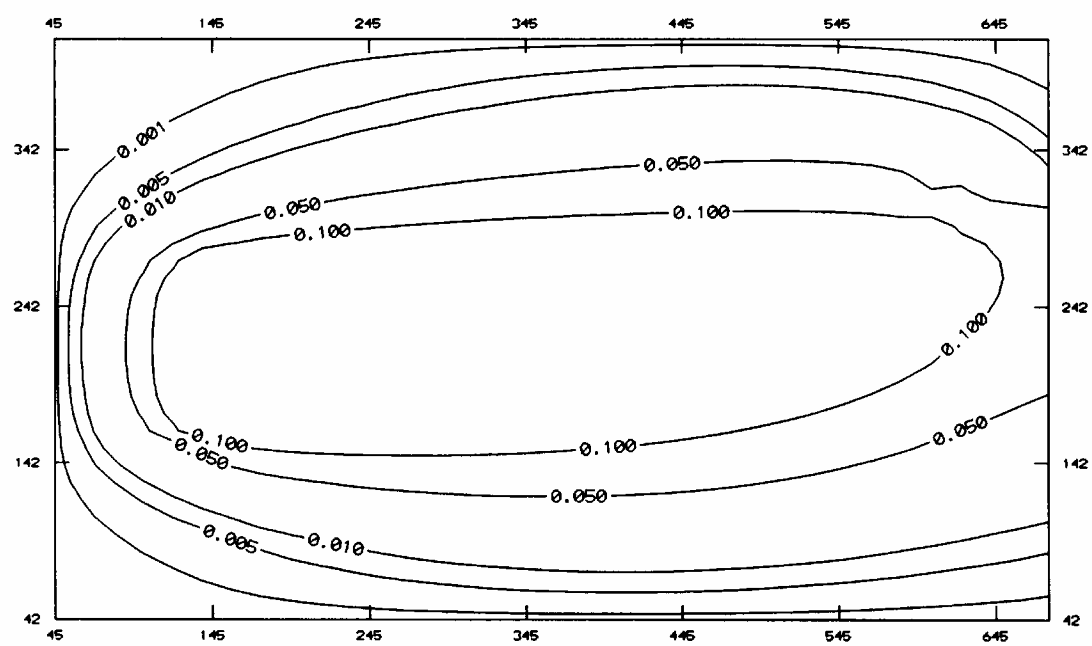

SEFTRAN Results
45
145
245
345
445
545
645
342
242
142
42
0.001
0.005
0.010
0.050
0.100

Figure 8. Concentration contours for WinTran and SEFTRAN at the final time step for Test Case 6.

WinTran Results



SEFTRAN Results



WinFlow Application Guide

WinFlow Assumptions

It is important to understand the many simplifying assumptions inherent in an analytical model before the model can be applied to a real-world problem. Chapter 5 described the equations that are solved in WinFlow. Chapter 6 verified that these equations are properly implemented in the WinFlow software. This chapter presents potential applications of WinFlow to the solution of ground-water problems. First, however, some important assumptions are discussed as they apply to the practical application of WinFlow. For easy identification, the primary assumptions are underlined.

WinFlow is designed to solve two-dimensional ground-water flow problems in a horizontal plane. It is not designed for two-dimensional cross-sections (2D vertical plane). <u>The two primary assumptions are that ground-water flow is horizontal and occurs in an infinite aquifer</u>. WinFlow should not be applied to aquifers exhibiting strong vertical gradients unless the scale of the problem is such that horizontal flow can still be considered dominant. WinFlow can be used even in cases where there are significant vertical gradients if the horizontal scale of the model is much larger than the vertical scale, such as in regional studies.

Another assumption is that the <u>aquifer hydraulic conductivity is assumed to be isotropic and homogeneous.</u> The base of the aquifer is horizontal and fixed at a given elevation. In the steady-state and transient models, the top of the aquifer is also horizontal and fixed at a given elevation. In the steady-state model, however, unconfined conditions are simulated when the hydraulic head is below the top of the aquifer. In the transient model, the aquifer is always confined, even when the head falls below the top of the aquifer.

<u>The reference head in the steady-state model is constant throughout all calculations</u>. The reference head is analogous to a constant head boundary condition in a numerical model. It is therefore very important to keep the reference head far from the area of interest so that model predictions are not impacted.

The reference head in the transient model is only used in combination with the uniform gradient to compute an initial planar potentiometric surface. Drawdowns computed by either the Theis (1935) or the Hantush and Jacob (1955) methods are then subtracted from the planar potentiometric surface to obtain the resulting flow field. Drawdowns are also subtracted from the reference head in the transient model; however, there is an option that allows the user to keep the reference head constant in the transient model. This option should only be used when trying to compare the transient model to the steady-state model.

All pumping rates, linesink fluxes, pond recharge, and elliptical recharge rates are constant through time. In the transient model, all wells start pumping or injecting water at time zero.

All wells are assumed to fully penetrate the aquifer. Wells are assumed to be perfectly efficient and linesinks are in perfect hydraulic communication with the aquifer. Both assumptions are rarely encountered in practice. There is often head loss around the well screen or stream bottom caused by clogging of the pore-space by fine-grained material (clay). There are two important consequences of imperfect hydraulic communication.

(1) Pumping rates predicted by WinFlow to achieve a desired response may not be attainable because more drawdown will be encountered in the actual well. The increased drawdown encountered in the field is caused by inefficiency around the well screen. The same effect will happen using linesinks to simulate trenches or drains.

(2) The amount of water produced or injected by a linesink to maintain a specified head in the linesink will be overestimated if the actual drain has less than 100 percent efficiency.

Particle traces and streamlines are two-dimensional. In cases where the aquifer receives recharge, the capture zone of a pumping well will be large enough to capture the amount of recharge equaling the pumping rate of the well (Larson et al., 1987). In two-dimensional analyses, such as in WinFlow, the capture zone extends upgradient until encountering a ground-water divide or infinity. This is an important consideration in designing a containment system.

Analysis of Remedial Actions

WinFlow can provide valuable guidance in designing a ground-water remediation system. The most obvious remedial action that WinFlow can simulate is "pump & treat" where the goal is to contain a volume of contaminated aquifer. WinFlow can simulate the effects of both pumping and injection wells. To illustrate the capture zone of a well, use reverse particle-tracking and start the particles in a circle around the well.

WinFlow can simulate trenches and drains using linesinks. There are two options in simulating drains: (1) specify a head to be maintained in the drain and WinFlow will compute the discharge rate necessary to achieve the given head; or (2) specify the discharge rate and compute the resulting head in the drain. To illustrate the capture zone of the drain, use reverse particle-tracking and start the particles along two lines on either side of the linesink.

WinFlow can simulate a lagoon closure by using ponds. To do this, set up the initial analytical model with ponds that simulate the lagoon. Adjust the pond recharge rate to match field-measured heads. Finally, remove the pond (or set the pond recharge equal to zero) to simulate the effects of closure.

The effects of capping can be simulated with a combination of elliptical recharge and circular ponds. Set up the initial analytical model using recharge to match field-measured heads. A circular cap can then be simulated with a pond that has a recharge rate equivalent to the regional recharge rate but opposite in sign (e.g. negative).

Pumping Test Analysis and Design

WinFlow's transient model can simulate the effects of a pumping test to facilitate interpreting test results or designing a future test. Pumping test results can be interpreted by contouring drawdown at a specified time after the start of the test. To contour drawdown, set the reference head equal to zero and the gradient equal to zero. Make sure that the top of the aquifer is less than zero if the steady-state model is used.

Drawdowns computed by WinFlow can be compared to drawdown contours from the pumping test. Hydraulic conductivity and storage can be adjusted until a reasonable match between observed and computed drawdown is achieved. Image wells can be added to the model to simulate boundary effects. Use calibration targets to provide a quantitative match between the results of your aquifer test and the model calculations.

When designing an aquifer test, WinFlow estimates the drawdown likely to occur at selected times and at various distances from the pumping well. Time and drawdown estimates can help select appropriate wells to monitor and determine the length of the test.

Regional Modeling

Strack (1989) advocates the use of "analytic element models" (his term for the superposition of analytical functions) in regional flow system modeling. At a regional scale, most aquifers are very thin compared to the distance across the aquifer in the horizontal plane. Thus, the z-axis (vertical dimension) becomes quite small and vertical gradients are negligible compared to horizontal gradients. In this case, the problem becomes two-dimensional and can be easily simulated with analytical functions.

The regional model is constructed using linesinks to simulate rivers and streams. Recharge from precipitation is applied in a large ellipse covering the area of interest. Circular recharge areas (ponds) simulate lakes. Obviously, wells represent areas of ground-water extraction, such as wellfields.

Strack (1989) has developed many complex analytical functions or analytic elements to facilitate regional modeling. The Single-Layer Analytic Element Model (SLAEM) developed by Strack contains these advanced functions not available in WinFlow. SLAEM is available from Dr. Strack.

Well Head Protection

Many states are requiring water companies to model the ground-water flow system around all public supply wells to determine the "zone of contribution" for each well. Small water companies will find it difficult to pay for expensive numerical modeling studies. WinFlow is ideally suited for these small wellfields, because a simple regional model can be constructed to comply with wellhead protection regulations at little cost. WinFlow can also be useful for preliminary studies at larger wellfields prior to numerical modeling.

To determine the zone of contribution for a particular time of travel, use reverse particle-tracking. Start the particles in a circle around each well and set the maximum travel time to the desired value.

WinTran Application Guide

Introduction

This chapter presents the major assumptions inherent in WinTran and guidelines for the use of the transport model. These guidelines include estimating memory requirements, dealing with model instabilities, and suggestions for simulating various transport scenarios.

WinTran Assumptions

It is important to understand the many simplifying assumptions inherent in any model before the model can be applied to a real-world problem. This chapter presents potential applications of WinTran to the solution of contaminant fate and transport problems. First, however, some important assumptions are discussed as they apply to practical application of WinTran. For easy identification, the primary assumptions are underlined.

WinTran is designed to solve two-dimensional ground-water flow and transport problems in a horizontal plane. It is not designed for two-dimensional cross-sections (2D vertical plane). The two primary assumptions are that ground-water flow is horizontal and contaminant concentrations are the same throughout the entire aquifer thickness. WinTran should not be applied to aquifers exhibiting strong vertical gradients unless the scale of the problem is such that horizontal flow can still be considered dominant. WinTran can be used even in cases where there are significant vertical gradients if the horizontal scale of the model is much larger than the vertical scale, such as in regional studies.

Another assumption is that the aquifer hydraulic conductivity is assumed to be isotropic and homogeneous. The base of the aquifer is horizontal and fixed at a given elevation. The top of the aquifer is also horizontal and fixed at a given elevation. Unconfined conditions are simulated when the hydraulic head is below the top of the aquifer.

The reference head in the flow model is constant throughout all calculations. The reference head is analogous to a constant head boundary condition in a numerical model. It is therefore very important to keep the reference head far from the area of interest so that model predictions are not impacted.

All pumping rates, linesink fluxes, pond recharge, and elliptical recharge rates are constant through time. The transport model simulates transient movement of the contaminant in this steady-state velocity field.

All wells are assumed to fully penetrate the aquifer. Wells are assumed to be perfectly efficient and linesinks are in perfect hydraulic communication with the aquifer. Both assumptions are rarely encountered in practice. There is often head loss around the well screen or stream bottom caused by clogging of the pore-space by fine-grained material (clay). There are two important consequences of imperfect hydraulic communication.

(1) Pumping rates predicted by WinTran to achieve a desired response may not be attainable because more drawdown will be encountered in the actual well. The increased drawdown encountered in the field is caused by inefficiency around the well screen. The same effect will happen using linesinks to simulate trenches or drains.

(2) The amount of water produced or injected by a linesink to maintain a specified head in the linesink will be overestimated if the actual drain has less than 100 percent efficiency.

Particle traces and streamlines are two-dimensional. In cases where the aquifer receives recharge, the capture zone of a pumping well will be large enough to capture the amount of recharge equaling the pumping rate of the well (Larson et al. 1987). In two-dimensional analyses, such as in WinTran, the capture zone extends upgradient until encountering a ground-water divide or infinity. This is an important consideration in designing a containment system.

Chemical reactions are reduced to two types, (1) linear, fully-reversible sorption using a retardation coefficient, and (2) first-order decay. WinTran can be used to simulate biological decay of organic compounds only if the biological reactions can be reduced to a first-order decay reaction. That is, a contaminant half-life is estimated for the compound.

Memory Requirements

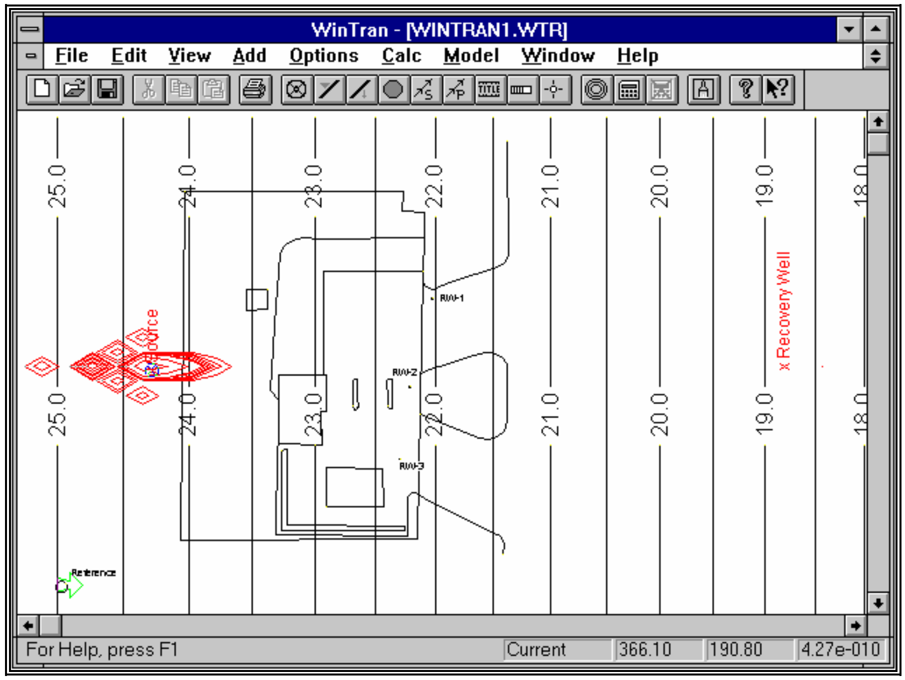
WinTran uses a substantial amount of computer memory to solve the finite-element transport model. The amount of memory required for each model is determined by the size of the contour matrix. The default size of the contour matrix is 35 x 35 (35 nodes in both the X- and Y-directions). In this case, the model requires about 1 megabyte of memory. The maximum matrix size allowed in WinTran is 100 x 100, requiring about 18 megabytes of memory. Other matrix sizes and memory requirements are shown below:

Matrix Size	Memory Required
35 x 35	1 megabyte
50 x 50	2.6 megabytes
75 x 75	8 megabytes
100 x 100	18 megabytes

Problems with Model Stability

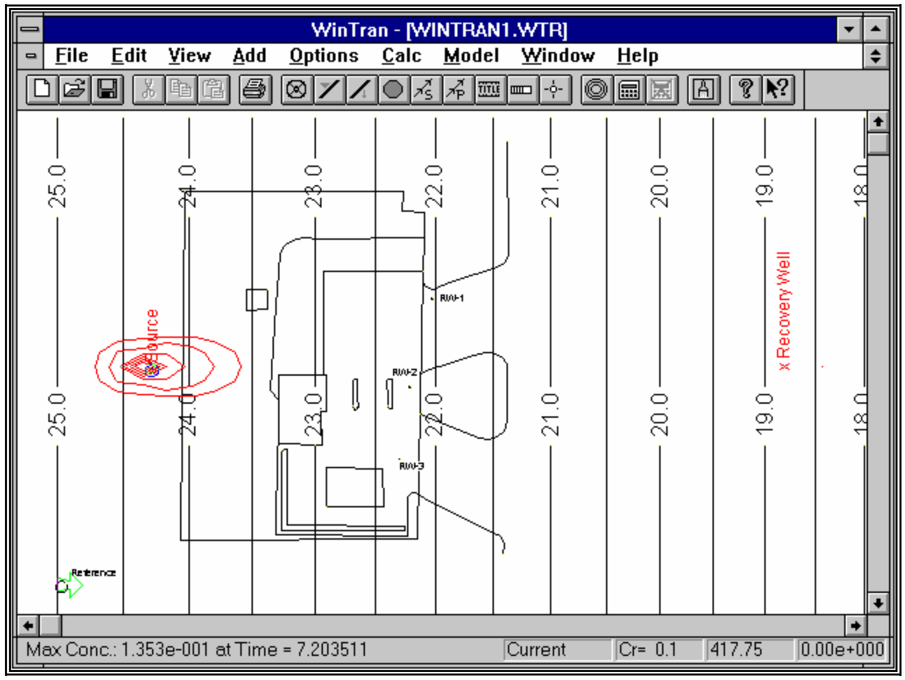
Numerical transport models require the user to carefully evaluate each simulation for potential errors. WinTran assists you in evaluating model error by displaying the mass balance error on the status bar when the transport model is running. The mass balance error is expressed as a percentage and should be less than 10 percent for a valid simulation. Usually, the mass balance error is less than 1 percent.

Even if the mass balance error is below 10 percent, there can be oscillations in the transport solution. Oscillations are indicated by negative concentrations computed by WinTran. In extreme cases, alternating nodes will have positive and negative concentrations producing diamond-shaped contours. The following screen shows a contour pattern that is typical of numerical oscillations:



Note the diamond shaped contours upgradient of the source. These contours are produced because alternating nodes are positive and negative. The contouring routine draws "bulls-eyes" around these high and low points producing the diamond-shaped contours. This is very typical of oscillating solutions and is probably the most common problem you will run into with WinTran.

The pattern above was produced in the tutorial model by lowering the time-step size to 0.1 days, using centered-in-time, and reducing the longitudinal dispersivity to 3 ft. This produces a Peclet number of 6.2, which is above the recommended limit of 2. In the screen shown below, the dispersivity value was increased to 30 ft, dropping the Peclet number to 0.62. This was enough to remove the oscillations.



When the transport solution oscillates, check the following:

(1) The Peclet number is displayed on the status bar as "Pe=" and is computed by dividing the nodal spacing (the distance between nodes in the contour matrix) by the longitudinal dispersivity. The Peclet number should generally be less than 2 for a stable solution. If you are experiencing mass balance problems or oscillations, increase dispersivity until the Peclet number is less than 2, as described above.

(2) The Courant number is another criterion used to judge the stability of a transport simulation. The Courant number is computed as the velocity times time-step size divided by nodal spacing. This criterion is displayed as "Cr=" on the status bar and should generally be less than 1. Again, if you are experiencing mass balance or oscillation problems, try decreasing the initial and maximum time-step sizes.

There are also times when the Courant number is too low. In cases where the Courant number is less than 0.1, there can be round-off errors in the matrix solver. In this case, you should increase the initial and maximum time-step sizes until the Courant number is close to 1.

There are two other WinTran options that can aid in model stability. These include the time discretization method (backward and centered in time) and upstream weighting. The time discretization methods are selected using the **Edit->Time Stepping** menu. Backward in time is unconditionally stable but is only first-order accurate, while centered in time is second-order accurate but may be subject to instability (Javandel et al., 1984). It is usually best to start with backward in time.

Upstream weighting factors in the X- and Y-directions are edited from the **Edit->Transport Parameters** menu. Upstream weighting factors of 1.0 indicate full upstream weighting, while a weighting factor of 0.0 turns off upstream weighting. Upstream weighting adds stability to the solution (helps eliminate oscillations) at the expense of added numerical dispersion. Numerical dispersion is artificial dispersion that produces similar results to an increase in the dispersivity coefficient.

Analysis of Remedial Actions

Setting Up the Flow Model

WinTran can provide valuable guidance in designing a ground-water remediation system. The most obvious remedial action that WinTran can simulate is "pump & treat" where the goal is to contain a volume of contaminated aquifer. WinTran can simulate the effects of both pumping and injection wells.

WinTran can simulate trenches and drains using linesinks. There are two options in simulating drains: (1) specify a head to be maintained in the drain and WinTran will compute the discharge rate necessary to achieve the given head; or (2) specify the discharge rate and compute the resulting head in the drain. To illustrate the capture zone of the drain, use reverse particle-tracking and start the particles along two lines on either side of the linesink.

WinTran can simulate a lagoon closure by using ponds. To do this, set up the initial analytical model with ponds that simulate the lagoon. Adjust the pond recharge rate to match field-measured heads. Finally, remove the pond (or set the pond recharge equal to zero) to simulate the effects of closure.

The effects of capping can be simulated with a combination of elliptical recharge and circular ponds. Set up the initial analytical model using recharge to match field-measured heads. A circular cap can then be simulated with a pond that has a recharge rate equivalent to the regional recharge rate but opposite in sign (e.g. negative).

Setting Up the Transport Model

Remedial alternatives are usually simulated in several stages, as described below:

(1) Calibrate the transport model to the observed contaminant plume. This is accomplished by adding source terms to the model (injection wells, infiltrating ponds, or injecting linesinks) and adjusting the source concentration until the desired plume is simulated. The length of the simulation should be chosen to approximate the length of time that the source of contamination has been effecting the groundwater system.

An alternative approach to calibrating the plume configuration is to import a SURFER grid file (e.g. test.grd) containing the contaminant distribution data (use **File->Import** from the main menu). The contoured concentrations are then used as initial conditions for the remedial simulation.

(2) Save the calibrated concentrations as initial conditions using the **Calc->Restart** option on the main menu. Skip this step if you have imported a SURFER grid file for initial conditions.

(3) Add the remediation system (pumping wells or linesinks, etc.) and rerun the transport model. To simulate source removal, delete the source terms added in State 1 above. This is accomplished by moving the cursor over the source element (well, pond, or linesink) until the four-arrow cursor () is displayed. Click the left mouse button to select the element and then press the delete key or select **Edit->Delete** from the main menu. Now, rerun the transport model to simulate source removal.

At any time during the simulations, you may save concentrations for later restart using the **File->Export** menu. Exporting concentration as a restart file (*.rst) will allow you to **Import** these concentrations in later simulations.

Simulating Biodegradation

Simulating the biodegradation of organic compounds is a popular modeling scenario, especially for dissolved hydrocarbons. WinTran does not simulate these complex degradation processes; however, the decay term in WinTran can be used to approximate biodecay. The biodegradation process is reduced to specifying a half-life for the compound. The half-life is the time required to remove half of the original mass. While the half-life is most often used for radioactive elements, such as uranium, it can also be used to express the decay of organic compounds through biodecay. The *Handbook of Environmental Degradation Rates* (Howard et al., 1991) is a good reference for contaminant half-life data.

Performing Risk Assessments

WinTran is not a risk assessment model but can be useful in risk assessments by providing concentration data over time at receptor locations. To obtain the concentration over time at these receptor locations, you must add a well at the receptor. Specify the flow rate as zero (0.0) and check the "Observation well" option on the well dialog. These concentration-time data may then be saved to a file for use in other programs. To save these data, select **File->Export** and choose the file time **Conc-Time** (*.cvt). The file is a DOS text file delimited by commas. The first line contains the well names and subsequent lines list the time and concentration for each well.

Digitized Maps

Digitized Map File Format

Digitized base maps increase the efficiency of site-specific modeling by placing the modeling results in context with the area to be modeled. As shown in the tutorial, WinFlow overlays the base map on head contours and streamlines, making it easier to interpret the results.

WinFlow uses a very simple file format for the digitized base map, as shown in Table 9. The file is made up of two sections. The first defines a series of line segments, while the second set of data defines a series of text strings. Each line segment requires the following data (1) the beginning and ending **X** and **Y** coordinates, (2) the line style, e.g., dashed or solid, and (3) the line color. The data for each line segment should appear on one line and be separated by at least one space between each data item. Commas may not be used to separate data items.

The following data items are required for each text item (1) **X** and **Y** coordinates of the lower left corner of the text, (2) angle of rotation of the text string, (3) height of the text, (4) color, and (5) a text string. The first four data items are entered on one line separated by at least one space between each data item. The text string is located on the following line and the height of the text string is in map coordinates (not in inches!).

Line and text colors are defined as integer numbers from 0 through 15. Each integer defines a unique color. The possible colors are shown in Table 10. These colors are all displayed on VGA color displays.

The digitized map file is a simple ASCII file that may be created in any text editor. You may also find it advantageous to write a simple program to convert files from your digitizing software to the WinFlow format. WinFlow also has the ability to convert DXF files directly. Simply choose **File** from the main menu and **Map** from the pull-down menu. Next select **DXF** from the menu. Specify the DXF file name and a conversion factor, which is explained below. The DXF file format is a relatively standard file format for CAD packages, such as AutoCad.

Table 9 File Format for WinFlow Digitized Maps.
Line 1 NLS, NTEXT
NLS = Number of line segments in map
NTEXT = Number of Text Strings in map
Lines 2 to NLS+1 (Enter one line for each line segment)
X1, Y1, X2, Y2, NDASH, NCOLOR
X1, Y1 = Beginning line coordinates
X2, Y2 = Ending line coordinates
NDASH = Positive integer for solid line, negative for dashed
NCOLOR = Color index (integer)
Lines NLS+2 to end (Enter one set per text item)
X1, Y1, ANGLE, HEIGHT, NCOLOR
TEXT
X1, Y1 = Coordinates of left side of text string
ANGLE = Angle of text string
HEIGHT = Height of text string
NCOLOR = Color index of text string
TEXT = Text string

Table 10 Definition of color indices.	
Index	Color
0	BLACK
1	BLUE
2	GREEN
3	CYAN
4	RED
5	MAGENTA
6	BROWN
7	WHITE
8	GRAY
9	LIGHT BLUE
10	LIGHT GREEN
11	LIGHT CYAN
12	LIGHT RED
13	LIGHT MAGENTA
14	YELLOW
15	BRIGHT WHITE

DXF Translator

The DXF (Drawing Interchange Format) file is a fairly standard format for exchanging data between CAD systems. In particular, the popular AutoCAD software uses DXF files extensively. A translator is provided with WinFlow to extract digitized information from DXF files and convert it to the WinFlow digitized map format.

The DXF file contains detailed data describing numerous CAD entities. An entity is a line or symbol placed on the drawing by the CAD system. The WinFlow DXF translator supports the following CAD entities:

LINES

POLYLINES

POINTS

ARCS

CIRCLES

TEXT

Certain aspects about these entities are ignored by the translator, such as elevation (for 3D CAD software such as AutoCAD Release 10), line style, and line thickness. In addition, the curve-fit and spline options applied to POLYLINES are ignored. The coordinates and color of the entity are preserved, however.

Many CAD drawings contain entities called BLOCKS, which are a collection of other entities (e.g., lines, circles, text, etc.). WinFlow will not interpret BLOCKS properly, so make sure that these are converted to other entities before creating the DXF file in your CAD package. In AutoCAD terminology, this is called "exploding" the blocks.

The DXF translator is activated from the File menu, as described above. Next, specify the DXF file name and a Map file name using standard Windows file dialogs. You only have to answer one additional prompt after starting the DXF translator - a conversion factor for the translation. Normally, a conversion factor of 1.0 will work; however, sometimes your CAD software will store coordinates in the DXF file in units of inches. If this happens, use a conversion factor of 0.0833333 (1.0/12.0). Each coordinate in the DXF file is multiplied by the conversion factor before being written to the WinFlow map file.

After all entities are processed in the DXF file, the digitized map file is created. A message to that effect is displayed at the bottom of the screen. After the translation is finished, the map file is imported into the model and displayed on your screen.

References

ASTM Standards

D 4104 Test Method (Analytical Procedure) for Determining Transmissivity of Nonleaky Confined Aquifers by Overdamped Well Response to instantaneous Change in Head (Slug Tests), ASTM, 4 p.

D 4105 Test Method (Analytical Procedure) for Determining Transmissivity and Storage Coefficient of Nonleaky Confined Aquifers by the Modified Theis Nonequilibrium Method, ASTM, 5 p.

D 4106 Test Method (Analytical Procedure) for Determining Transmissivity and Storage Coefficient of Nonleaky Confined Aquifers by the Theis Nonequilibrium Method, ASTM, 5 p.

D5920-96. Test Method (Analytical Procedure) for Tests of Anisotropic Unconfined Aquifers by Neuman Method, ASTM, 8 p.

Books

Bear, J. And A. Verruijt, 1987, *Modeling Groundwater Flow and Pollution*, D. Reidel Publishing Company, Boston, 414 p.

Butler, J. J., Jr., 1998, *The Design, Performance, and Analysis of Slug Tests,* Lewis Publishers, CRC Press, Boca Raton, Florida, 252 p.

Horne, R. N., 1995, *Modern Well Test Analysis*, Petroway, Inc., Palo Alto, California, 257 p.

Howard, P., R. Boethling, W. Jarvis, W. Meylan, and E. Michalenko, 1991, *Handbook of Environmental Degradation Rates*, Lewis Publishers, Inc., Chelsea, MI.

Huyakorn, P.S., A.G. Kretschek, R.W. Broome, B.H. Lester, J.W. Mercer, 1984, Testing and Validation of Models for Simulating Solute Transport in Groundwater: Development, Evaluation, and Comparison of Benchmark Techniques, International Groundwater Modeling Center Report GWMI 84-13, Golden, Colorado.

Javandel, I., C. Doughty, and C.F. Tsang, 1984, *Groundwater Transport: Handbook of Mathematical Models*, American Geophysical Union, Water Resources Monograph 10, Washington, D.C.

Kruseman, G.P. and N.A. de Ridder, 1990, *Analysis and Evaluation of Pumping Test Data*, Second Edition, ILRI publication 47, International Institute for Land Reclamation and Improvement, The Netherlands, 377 p.

Lohman, S.W., 1979, *Ground-Water Hydraulics*, U.S.G.S. Professional Paper 708, 70 p.

Reed, J.E., 1980, *Type Curves for Selected Problems of Flow to Wells in Confined Aquifers*, USGS TWRI Book 3, Chapter B3.

Strack, O.D.L., 1989, Groundwater Mechanics, Prentice Hall, Englewood Cliffs, New Jersey, 732 p.

Wexler, E.J., 1992, *Analytical Solutions for One-, Two-, and Three-Dimensional Solute Transport in Ground-Water Systems with Uniform Flow*, USGS Techniques of Water Resource Investigations (TWRI), Book 3, Chapter B7.

Journal Papers

Abramowitz, M., and I.A. Stegun, 1965, Handbook of Mathematical Functions, Dover Publications, New York.

Case, C.M., D.L. Ghiglieri, and K. Fallon, 1979, Tables of the Leaky Aquifer Well Function, Desert Research Institute Publication No. 45016, Water Resources Center, Desert Research Institute, Reno, Nevada, 31 p.

Birsoy Y.K. and Summers W.K., 1980, *Determination of aquifer parameters from step tests and intermittent pumping data.* Ground Water, 18, 137-146.

Black, J H, 1978, *The use of the slug test in groundwater investigations*, Water Services, March, p. 174-178.

Bouwer, H. and R.C. Rice, 1976, *A slug test for determining hydraulic conductivity of unconfined aquifers with completely or partially penetrating wells*, Water Resources Research, v. 12, p. 423-428.

Case, C.M., D.L. Ghiglieri, and K. Fallon, 1979, Tables of the Leaky Aquifer Well Function, Desert Research Institute Publication No. 45016, Water Resources Center, Desert Research Institute, Reno, Nevada, 31 p.

Cooper, H.H., J.D. Bredehoeft, and I.S. Papadopulos, 1967, *Response of a finite-diameter well to an instantaneous charge of water*, Water Resources Research, v.3, no. 1, p. 263-269.

Eden, R.N. and Hazel C.P., 1973, *Computer and graphical analysis of variable discharge pumping tests of wells.* Inst. Engrs. Australia, Civil Eng. Trans., 5-10

Hantush, M.S., 1956, *Analysis of data from pumping tests in leaky aquifers*, Am. Geophys. Union Trans., v. 37, no. 6, p. 702-714.

Hantush, M.S., 1960, *Modification of the theory of leaky aquifers*, Journal of Geophysical Research, v. 65, no. 11, p. 3713-3725.

Hantush, M.S., 1964, *Hydraulics of Wells*, In *Advances in hydroscience* (V.T. Chow, editor), Vol. 1, pp. 281-432, Academic Press, New York.

Hantush, M.S., 1967, Growth and decay of groundwater-mounds in response to uniform percolation, Water Resources Research, Vol. 3, No. 1, pp. 227-234.

Hantush, M.S. and C.E. Jacob, 1955, *Non-steady radial flow in an infinite leaky aquifer,* Am. Geophys. Union Trans., v. 36, no. 1, p. 95-100.

Hvorslev, M.J., 1951, *Time lag and soil permeability in ground water observations*, U.S. Army Corps of Engineers Waterway Experimentation Station, Bulletin 36.

Hyder, Z., Butler, J. J., Jr., McElwee, C. D., and Liu, W. Z., *Slug tests in partially penetrating wells*, Water Resour. Res., 30(11), 2945, 1994.

Jacob, C.E., 1944, *Notes on determining permeability by pumping tests under watertable conditions*, USGS Open File Report, In: USGS Water Supply Paper 1536-I, 1963, pp. 245-271.

Jacob, C.E., 1963, *Determining the permeability of water-table aquifers, in Bentall, Ray, compiler, Methods of determining permeability, transmissibility, and drawdown*, U.S. Geol. Survey Water-Supply Paper 1536-1, p. 245-271.

Kipp, K. L., Jr., 1985*, Type curve analysis of inertial effects in the response of a well to a slug test*, Water Resources Research, v.21, no. 9, p. 1397-1408.

Larson, S.P., C.B. Andrews, M.D. Howland, and D.T. Feinstein, 1987, Three-dimensional modeling analysis of ground water pumping schemes for containment of shallow ground water contamination, Proceedings of the Solving Ground-Water Problems with Models Conference, National Ground Water Association, Denver, pp. 517-531.

McDonald, M.G. and A.W. Harbaugh, 1988, A Modular Three-Dimensional Finite-Difference Ground-Water Flow Model, USGS Techniques of Water Resource Investigations, Book 6, Chapter A1, 528 p.

Moench, Allen F., 1997, *Flow to a well of finite diameter in a homogeneous, anisotropic water table aquifer.*, Water Resources Research, vol. 33, no. 6, pp 1397-1407.

Moench, A.F., 1996, *Flow to a well in a water-table aquifer: An improved Laplace Transform solution*, Ground Water, vol. 34, no. 4, pp. 593-596.

Moench, A.F., 1985*,Transient Flow to a Large-Diameter Well in an Aquifer With Storative Semiconfining Layers*, Water Resources Research, vol. 21, no. 8, pp. 1121-1131.

Moench, A.F., 1984, *Double-porosity models for a fissured groundwater reservoir with fracture skin*, Water Resources Research, vol. 20, no. 7, pp. 831-846.

Neuman, S.P., 1972, *Theory of flow in unconfined aquifers considering delayed response of the watertable*, Water Resources Research, vol. 8, pp 1031-1045.

Neuman, S.P., 1974, *Effect of partial penetration on flow in unconfined aquifers considering delayed gravity response*, Water Resources Research, vol. 10, pp 303-312.

Papadopulos, IS. and H.H. Cooper, 1967, *Drawdown in a well of large diameter*, Water Resources Research, vol. 3, no. 1, pp. 157-168.

Reilly, T.E., O.L. Franke, G.D. Bennett, 1987, The Principle of Superposition and Its Application in Ground-Water Hydraulics, USGS Techniques of Water Resource Investigations, Book 3, Chapter B6, 28 p.

Rumbaugh, J.O., 1991, QuickFlow - Analytical 2D Ground-Water Flow Model, Version 1.0, Geraghty & Miller, Inc., Reston, Virginia.

Shafer, J., 1987, GWPATH: Interactive Ground-Water Flow Path Analysis, Bulletin 69, Illinois State Water Survey, Champaign, Illinois.

Spane, F.A. and S.K. Wurstner, *DERIV: A Computer Program for Calculating Pressure Derivatives for Use in Hydraulic Test Analysis*, Ground Water, vol. 31, no. 5, pp. 814-822.

Theis, C.V., 1935, *The relation between the lowering of the piezometric surface and the rate and duration of discharge of a well using groundwater storage*, Trans. Amer. Geophys. Union, Vol. 16, pp. 519-524.

Software

Huyakorn, P.S., R.W. Broome, B.H. Lester, J.W. Mercer, 1984, SEFTRAN: Simple Efficient Flow and Transport Model, GeoTrans, Sterling, Virginia.

Rumbaugh, J.O., 1991, *QuickFlow - Analytical 2D Ground-Water Flow Model*, Version 1.0, Geraghty & Miller, Inc., Reston, Virginia.

Index

3

A

B

C

G

H

I

L

M

N

T

U

V

W

Z