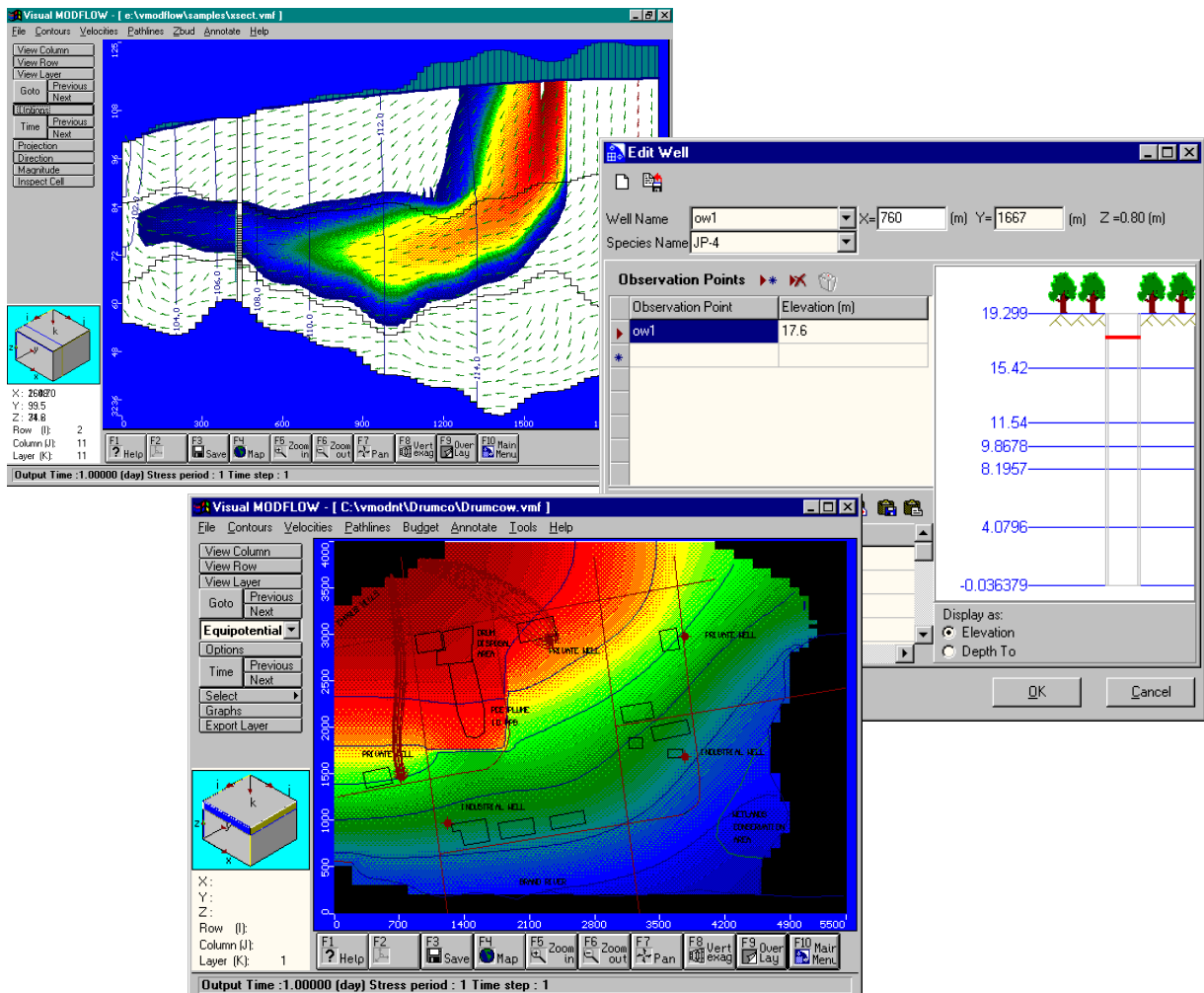


# Visual MODFLOW

## Student Version User's Guide

The fully integrated, three-dimensional, graphical modeling environment for professional groundwater flow and contaminant transport modeling.



## **DISCLAIMER OF WARRANTY**

**This manual and associated software are sold “as is” and without warranties as to performance or merchantability. The seller’s salespersons may have made statements about this software. Any such statements do not constitute warranties.**

**This program is sold without any express or implied warranties whatsoever. No warranty of fitness for a particular purpose is offered. The user is advised to test the program thoroughly before relying on it. The user assumes the entire risk of using the program. Any liability of seller or manufacturer will be limited exclusively to replacement of diskettes defective in materials or workmanship.**

Visual MODFLOW is a trademark, owned by Waterloo Hydrogeologic Inc. Microsoft is a registered trademark, and Windows is a trademark of the Microsoft Corporation. Borland is a trademark of Borland International, Inc. EXml is a trademark of CUESoft. Adobe, the Adobe logo and Acrobat are registered trademarks of Adobe Systems, Inc. MODFLOW and MODPATH are trademarks of the United States Geological Survey. MT3DMS is a trademark of The University of Alabama. MT3D96 and MT3D99 are trademarks of S.S. Papadopoulos and Associates Inc. RT3D is a trademark of the Pacific Northwest National Laboratory and the United States Department of Energy. PEST is a trademark of Watermark Numerical Computing.

### **Contact Information**

#### **Waterloo Hydrogeologic Inc.**

180 Columbia Street West - Unit 1104  
Waterloo, Ontario, CANADA  
N2L 3L3

Phone (519) 746 1798

Fax (519) 885 5262

E-mail: [techsupport@flowpath.com](mailto:techsupport@flowpath.com)

Web: [www.flowpath.com](http://www.flowpath.com)

#### **Visual MODFLOW**

© 2000 Waterloo Hydrogeologic Inc.  
All Rights Reserved.

## **About the Visual MODFLOW Student Version**

The Visual MODFLOW Student Version and the accompanying documentation is intended for educational use. This is a fully-functional version of the standard Visual MODFLOW software package with the following limitations:

- Maximum 40 columns in the X direction
- Maximum 40 rows in the Y direction
- Maximum 3 model layers
- Maximum 3 property zones
- Maximum 2 pumping wells

The Visual MODFLOW Student Version comes with detailed on-line help and a comprehensive step-by-step tutorial exercise to guide you through the steps of building and calibrating a groundwater flow model using Visual MODFLOW, and evaluating advective transport of contaminants from an underground storage tank leak.

Waterloo Hydrogeologic authorized users of the Visual MODFLOW Student Version to reproduce the Visual MODFLOW Student Version User's Guide and the Visual MODFLOW Student Version Tutorial Guide in their entirety (no modifications permitted) for educational purposes only.

## **Software Licencing**

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

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

Documentation



# Table of Contents

<i>Contact Information</i> .....	ii
<i>About the Visual MODFLOW Student Version</i> .....	iii
<i>Software Licencing</i> .....	iii
<b>Waterloo Hydrogeologic Inc.</b> .....	<b>vii</b>
<b>How to Contact WHI</b> .....	<b>vii</b>
<b>Obtaining Technical Support</b> .....	<b>vii</b>
<b>WHI Training and Consulting</b> .....	<b>viii</b>
<b>Other Products by WHI</b> .....	<b>ix</b>
<i>AquaChem for Windows 95/98/NT/2000</i> .....	<i>ix</i>
<i>AquiferTest for Windows 95/98/NT/2000</i> .....	<i>ix</i>
<i>FLOWPATH II for Windows 95/98/NT/2000</i> .....	<i>ix</i>
<i>Visual Groundwater for Windows 95/98/NT/2000</i> .....	<i>ix</i>
<i>Visual HELP for Windows 95/98/NT/2000</i> .....	<i>x</i>
<i>WHI Unsat Suite Plus for Windows 95/98/NT/2000</i> .....	<i>x</i>
<b>1 - Introduction</b> .....	<b>1</b>
<b>Visual MODFLOW</b> .....	<b>2</b>
<i>Hardware Requirements</i> .....	<i>2</i>
<i>Windows 95/NT Swap File</i> .....	<i>2</i>
<i>Installing Visual MODFLOW</i> .....	<i>2</i>
<i>Starting Visual MODFLOW</i> .....	<i>2</i>
<i>About the Interface</i> .....	<i>3</i>
<b>Suggested Reference Books</b> .....	<b>3</b>
<b>2 - General Features</b> .....	<b>5</b>
<b>Getting Around In Visual MODFLOW</b> .....	<b>5</b>
<b>Screen Layout</b> .....	<b>7</b>
<b>Creating a New Model</b> .....	<b>9</b>
<i>Base Maps</i> .....	<i>10</i>
Dual Co-ordinate System .....	<i>12</i>
DXF Base Maps .....	<i>13</i>
BMP Base Map .....	<i>13</i>

<b>Visual MODFLOW Printing Options.....</b>	<b>16</b>
<i>Type 1 Printing – IChart Graphs.....</i>	<i>16</i>
Printer Setup .....	16
Page Layout .....	17
Margins .....	17
Page Design .....	17
Title Blocks.....	18
<i>Type 2 Printing – Model Domain Screen .....</i>	<i>20</i>
<i>Printing to a File.....</i>	<i>22</i>
<b>Numeric Engines.....</b>	<b>22</b>
<i>Transport Engines.....</i>	<i>23</i>
<i>Transport Engine Options .....</i>	<i>25</i>
Species Tab.....	27
General Tab .....	27
Model Params. Tab.....	28
Species Params. Tab .....	29
<i>Variants.....</i>	<i>29</i>
Conditions for Re-Initializing a Variant .....	30
Copying a Variant.....	31
<b>Common Tools .....</b>	<b>33</b>
<i>Overlay Function Features.....</i>	<i>33</i>
<i>IChart Plotting Properties.....</i>	<i>35</i>
The Axis Tabs.....	37
The Titles Tab.....	39
The Series Tab .....	41
The Legend Tab.....	45
The Operations Tab .....	46
The Settings Tab.....	48
The Axis (2) Tab.....	52
The Info Tab .....	52
<i>Cell Inspector.....</i>	<i>53</i>

# Waterloo Hydrogeologic Inc.

---

Waterloo Hydrogeologic, Inc. is a recognized leader in the development and application of software for interpreting, analyzing, managing and protecting groundwater resources. Since 1989, Waterloo Hydrogeologic has been developing powerful, yet easy-to-use, software to meet the advanced technology requirements of groundwater and environmental professionals. Today, with over 7, 000 registered installations in 85 countries, Waterloo Hydrogeologic software products are used and approved by consultants, regulators and educators worldwide.

## How to Contact WHI

---

If, after reading this manual and using Visual MODFLOW, you would like to contact Waterloo Hydrogeologic Inc. with comments, suggestions, or if you need technical assistance on how Visual MODFLOW works, you can reach us at:

**Waterloo Hydrogeologic Inc.**  
180 Columbia Street West - Unit 1104  
Waterloo, Ontario, CANADA  
N2L 3L3

Phone (519) 746 1798  
Fax (519) 885 5262

Email: [techsupport@flowpath.com](mailto:techsupport@flowpath.com)  
Web: [www.flowpath.com](http://www.flowpath.com)

Our technical support hours are 1:30 p.m. to 4:30 p.m. EST. Monday to Friday. To help us handle your problem as quickly as possible, please read the **Obtaining Technical Support** Section.

## Obtaining Technical Support

---

If you have a technical problem with this software, please feel free to contact us. We provide free technical support to registered users of Visual MODFLOW. By technical problem, we mean a problem with the functioning of the software. For advice on numerical modeling please enquire about our extended modeling support service. We also provide numerous public training courses and we will gladly provide a guide for in-house training services.

Generally, we reply to all technical support questions within one business day, however please allow up to two business days for a response.

In addition to a brief description of your problem, on all correspondence please provide your

- name,
- company and office,
- telephone and fax numbers,
- email address,
- serial number, and
- the exact version number.

The version number can be found using the [**About Visual MODFLOW**] button in the Main Menu screen under [**Help**].

If your question is of a "How do I...?" type, the above information is generally sufficient. However, if you are reporting a bug (especially one that causes a system crash) or are having technical problems with the program please provide the following additional information:

- a list of your current hardware setup,
- what operating system you are using (Win95/Win98/WinNT)
- the amount of RAM that you have on your system,
- the amount of free hard disk space remaining on your machine, and
- a detailed description of when and how the problem occurs.

If we are unable to reproduce your problem, or are otherwise unable to solve it, we may ask you to send us a copy of your project files.

## **WHI Training and Consulting**

---

Waterloo Hydrogeologic Inc. offers both individually tailored and public open-enrollment training courses in groundwater modeling. Our modeling courses emphasize how to set up a proper groundwater model (grid design, boundary conditions, etc.), the interpretation of results (calibration, prediction, etc.), and extensive coverage of the mechanics of using Visual MODFLOW. Please contact us for the latest course schedule or to arrange in-house training.

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



## **Other Products by WHI**

---

At WHI we are continually improving our existing products and developing new and innovative software products to meet the advancing technology requirements of groundwater and environmental professionals around the world. The following is a brief description of some other software products developed and distributed by Waterloo Hydrogeologic, Inc.

### **AquaChem** *for Windows 95/98/NT/2000*

AquaChem is a fully-integrated software package developed specifically for graphical and numerical analysis of geochemical data sets. It features a powerful database of common geochemical parameters and automated calculations, a comprehensive selection of 13 different geochemical data plotting techniques, and a built-in interface to PHREEQC geochemical modeling.

### **AquiferTest** *for Windows 95/98/NT/2000*

The most complete and easy-to-use software package for graphical analysis and reporting of pumping test and slug test analysis data. It comes with a comprehensive selection of built-in solution methods for estimating transmissivity, hydraulic conductivity and storage properties for confined, unconfined, leaky and fractured rock aquifers.

### **FLOWPATH II** *for Windows 95/98/NT/2000*

The next generation of powerful, yet easy-to-use software for simulating steady-state groundwater flow with pathlines and contaminant transport. It computes hydraulic heads, pathlines, travel times, velocities and water balances (verified against the USGS MODFLOW, approved by the US EPA, and recommended by the IGWMC).

### **Visual Groundwater** *for Windows 95/98/NT/2000*

Visual Groundwater is the one 3-D visualization software package that any groundwater and environmental professional can use to deliver high-quality, three-dimensional presentations of subsurface characterization data and groundwater modeling results.

## **Visual HELP** *for Windows 95/98/NT/2000*

Visual HELP for Windows 95/98/NT is the most advanced hydrological modeling environment available for designing landfills, predicting leachate mounding and evaluating potential leachate seepage to the groundwater table. Visual HELP combines the latest version of the HELP model (Hydrologic Evaluation of Landfill Performance model v.3.07) with an easy-to use interface and powerful graphical features for designing the model and evaluating the modeling results.

## **WHI Unsat Suite Plus** *for Windows 95/98/NT/2000*

The WHI UnSat Suite is a fully-integrated software package that combines the most popular and widely-used models for simulating one-dimensional groundwater flow and contaminant transport in the unsaturated zone (includes SESOIL, VS2DT, VLEACH, PESTAN and the HELP model). These models have all been compiled and optimized to run as native Windows applications and are seamlessly integrated within the WHI UnSat Suite graphical modeling environment.

### **We want your feedback...**

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

# 1

## 1 - Introduction

---

Visual MODFLOW is the most complete and easy-to-use modeling environment for practical applications in three-dimensional groundwater flow and contaminant transport simulations. This fully integrated package combines MODFLOW, MODPATH, ZoneBudget, MT3Dxx/RT3D, and PEST with the most intuitive and powerful graphical interface available. The intuitive graphical tools and logical menu structure allow you to:

- easily dimension the model domain and select units,
- conveniently assign model properties and boundary conditions,
- run model simulations (MODFLOW, MODPATH, ZoneBudget, MT3Dxx/RT3D, and PEST),
- calibrate the model using manual or automated techniques, and
- visualize and interpret the simulation results

The model grid, input parameters and results can be visualized in cross-section or plan view at any time during the development of the model or the displaying of the results. For complete three-dimensional groundwater flow and contaminant transport modeling, Visual MODFLOW is the only software package you will need.

When you purchase Visual MODFLOW, or any Waterloo Hydrogeologic software product, you not only get the best software in the industry, you also gain the reputation of the company behind the product, and free technical support for the software from our team of qualified modeling professionals. Waterloo Hydrogeologic has been developing groundwater software since 1989 and groundwater professionals in over 80 countries around the world recognize our name. This type of recognition is invaluable in establishing the credibility of your modeling software to clients and regulatory agencies.

Visual MODFLOW was first released in August of 1994 and has quickly become the standard modeling environment for more than 5000 users at consulting firms, educational institutions and government agencies worldwide. It is used by the United States Geological Survey (USGS) and the United States Environmental Protection Agency (USEPA), and is the featured model in many continuing education courses around the world.

# Visual MODFLOW

---

## Hardware Requirements

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

- Pentium-based computer
- 32 MB of RAM (64 MB recommended)
- CD ROM drive
- A hard drive, with at least 100 Mbytes free;
- Windows 95/ Windows 98/ Windows NT 4.0 (Service Pack 3).

If you have any problems with your particular system configuration, please make sure that you followed the installation instructions precisely (see below in **Installing Visual MODFLOW**). If the problem is still unresolved, contact your hardware experts. Finally, if you are still having trouble, see the section in this chapter on **How to Contact WHI**.

## Windows 95/NT Swap File

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

## Installing Visual MODFLOW

Visual MODFLOW must be installed on your hard disk or network disk to run. Please read the section on hardware requirements at the beginning of this chapter to ensure that your system meets the requirements before performing the installation. The **Setup32.exe** executable file is used to install Visual MODFLOW for Windows in Windows 95/98/NT.

The installation procedure is outlined in the Visual MODFLOW Installation Guide and on the Visual MODFLOW Installation CD-ROM in the directory **Network Instructions / Network Installations**. Please refer to one of these documents before proceeding with the installment.

## Starting Visual MODFLOW

Once the program has been installed under Windows 95/98/NT 4.0, simply double click on the Visual MODFLOW 2.8.1 shortcut icon or click on **Start/Programs/WHI Software/Visual MODFLOW 2.8.2/Visual MODFLOW 2.8.2**.

## About the Interface

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

The **Input Module** allows the user to graphically assign all of the necessary input parameters for building a three-dimensional groundwater flow and contaminant transport model. The input menus represent the basic "model building blocks" for assembling a data set for MODFLOW, MODPATH, ZoneBudget, and MT3Dxx/RT3D. These menus are displayed in the logical order to guide the modeler through the steps necessary to design a groundwater flow and contaminant transport model.

The **Run Module** allows the user to modify the various MODFLOW, MODPATH, MT3Dxx/RT3D, and PEST parameters and options which are run-specific. These include selecting initial head estimates, setting solver parameters, activating the re-wetting package, specifying the output controls, etc. Each of these menu selections has default settings, which are capable of running most simulations.

The **Output Module** allows the user to display all of the modeling and calibration results for MODFLOW, MODPATH, ZoneBudget, and MT3Dxx/RT3D. The output menus allow you to select, customize, and overlay the various display options for presenting the modeling results.

## Suggested Reference Books

---

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

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



# 2

## 2 - General Features

---

This chapter presents information on:

- Getting Around Visual MODFLOW,
- Screen Layout,
- Creating a New Model,
- Importing Existing MODFLOW Models,
- Changing Units in an Existing Model
- Printing Options,
- Numeric Engines, and
- Tools Common to the Input, Run, and Output Modules.

### Getting Around In Visual MODFLOW

---

The Visual MODFLOW graphical modeling environment consists of four modules: Main Menu, Input Module, Run Module, and Output Module, plus the system Setup and Help. The Main Menu screen contains the following options:

File Input Run Output Setup Help

- |                 |   |
|-----------------|---|
| <b>[File]</b>   | Select a file utility or exit Visual MODFLOW.   |
| <b>[Input]</b>  | Go to the Input Module to modify the current Visual MODFLOW data set.   |
| <b>[Run]</b>    | Go to the Run Module to modify run time parameters and run numerical simulations in either project or batch mode. |
| <b>[Output]</b> | Go to the Output Module to post-process results from numerical simulations.                                       |
| <b>[Setup]</b>  | Choose the desired variant and transport numerical engine.  |

**[Help]** Main help screen appears and general information on Visual MODFLOW given.

Your Microsoft compatible mouse buttons perform as follows:

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

**Right button:** This button has different functions depending on the context. For example, it closes polygons or ends a line during the assignment of properties, boundaries, etc. During grid design, it also locates grid rows or columns on precise co-ordinates. Within dialog boxes, using the right button brings up a sub-menu to copy, paste, delete etc.

The Alt shortcut keys can be used to access toolbar items and to select many buttons and can be used anytime an item or button text has an underlined letter. Simply press the Alt key and the letter key, on you keyboard, at the same time to access the item or select the button. The number and letter keys are active only when numerical or text input is required; all other keys are ignored by Visual MODFLOW.

After starting Visual MODFLOW, you will be in the Main Menu. To proceed, click on **[File]**, and select one of the nine options shown:

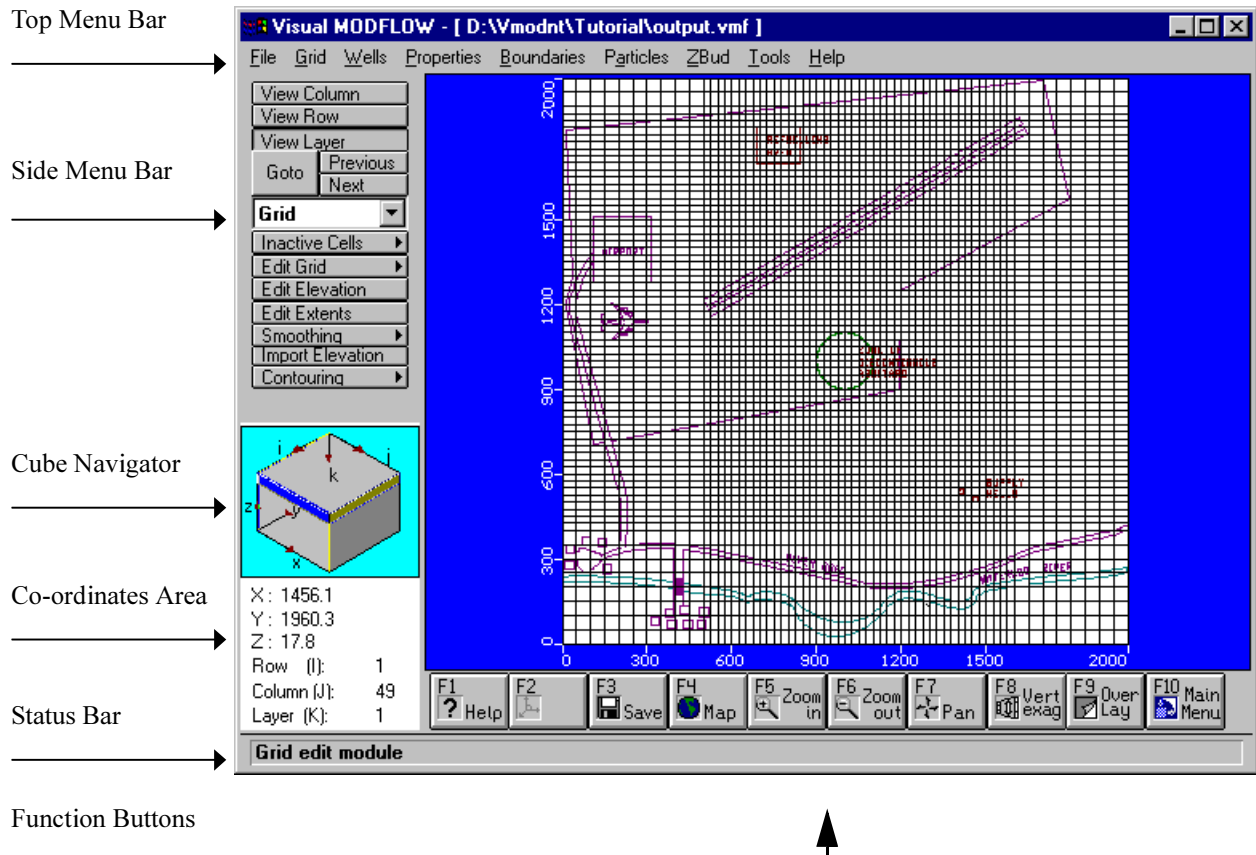
- [1] **[Open...]** to open an existing file; or
- [2] **[New]** to create a new data set;
- [3] **[Save As]** to save the current model to a different filename;
- [4] **[Close]** to close the current model and remain in Visual MODFLOW.
- [5] **[Import MODFLOW]** to import an existing MODFLOW dataset (\*.BAS);
- [6] **[Export...]** to export the graphic as an AutoCAD DXF file in World or Model Co-ordinates (\*.DXF) or an Enhanced Windows Metafile (\*.EMF).
- [7] **[Change Units...]** to change units in an existing model. Note this will not convert existing values to you new system of units.
- [8] **[Print]** to edit page layout and project information (old Visual MODFLOW printing). Prints the current screen to the selected printer.
- [9] **[eXit]** to shut-down Visual MODFLOW.

Choose a file by clicking on the file name and then pressing the **Enter** key or double clicking on the file path. Continue reading this chapter for more information on generating a new data set and importing an existing model. Try the step-by-step example called Visual MODFLOW Tutorial found on your installation CD.



## Screen Layout

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

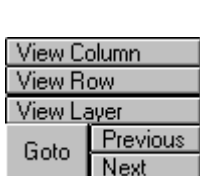


**Top Menu Bar:**

Provides options, which vary depending on the particular module.

**Side Menu Bar:**

Contains the view options plus functions particular to the current screen or module. The view options are as follows:



**[View Column]**

View a cross-section along a column.

**[View Row]**

View a cross-section along a row.

**[View Layer]**

Switch from cross-section to plan view.

**[Goto]**

View a specified column, row or layer.

**[Previous]**

View previous column, row, or layer.

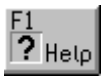
**[Next]** View next column, row, or layer.

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

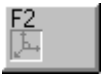
**Co-ordinates Area:** Shows the current location of the cursor in model co-ordinates, and shows the current cell indices on which the cursor is located.

**Status Line:** Describes the functionality and use of the feature currently highlighted by the cursor.

**Function Buttons:** Common functions to the Input, Run, and Output screens can be selected by clicking on the button or by pressing the function key on your keyboard. The function buttons are:



**F1 - Help** Accesses the general help dialog box.



**F2 - 3D** Reserved for future 3-D options.



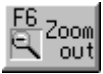
**F3 - Save** Updates the file as previously named.



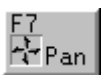
**F4 - Map** Displays a File Selection dialog where you can select a **.DXF** format map file for importing. The map can be toggled on and off in the Overlay dialog box.



**F5 - Zoom In** Allows you to extend a zoom window over the screen with the left mouse button. Click with the right mouse button to specify the co-ordinates of the zoom area for consistent plot windows.



**F6 - Zoom Out** Resets the screen image to the model extent.



**F7 - Pan** Allows the user scroll through and move the location of the model domain on the window. The first click with the left mouse button selects a point on the model and the second click places the selected point on a new point in the window.



**F8 - Vert Exag** Allows the user to specify the amount of vertical exaggeration seen in the row or column view.



**F9 - Overlay** Allows the user to toggle on or off the various plot and map features.



**F10 - Main Menu** Returns you to the Main Screen.

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

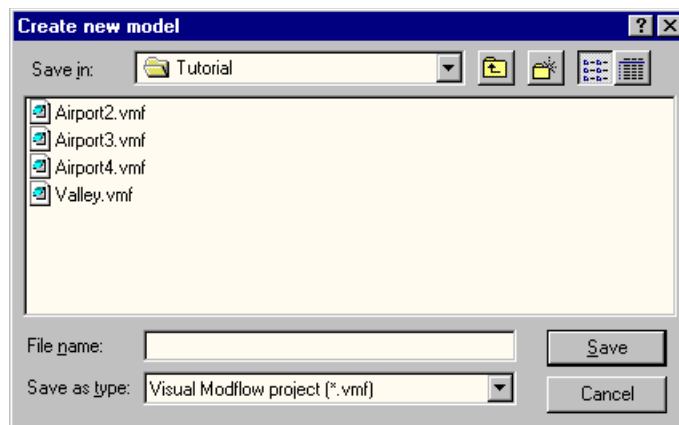
For the CAD environment, a Microsoft (or compatible) mouse is necessary. In all CAD environments, the following keys are active:

<b>Function Buttons</b>	Described above.
<b>Enter Key</b>	Selects the currently highlighted area.
<b>Esc Key</b>	Cancels current window.
<b>Cursor (arrow) keys</b>	Have limited functionality (such as choosing menu options).
<b>Tab Key</b>	Moves from cell to cell.

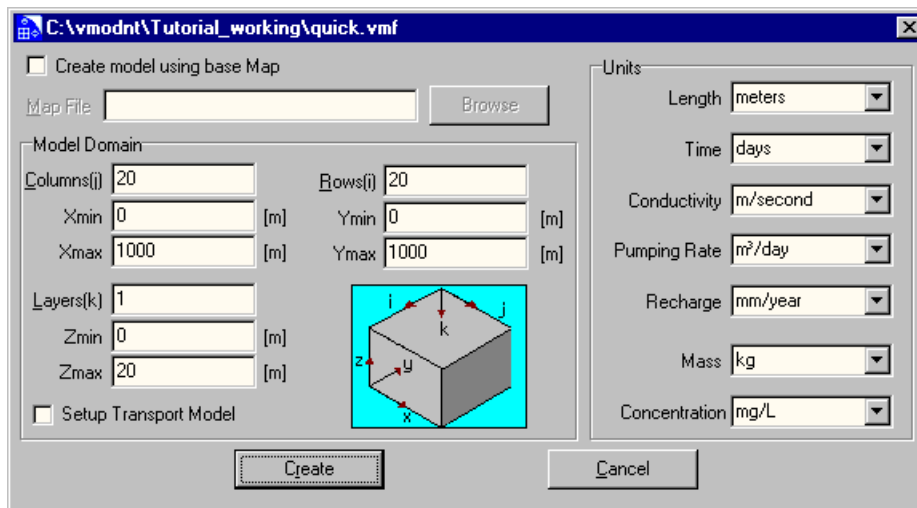
## Creating a New Model

---

Within the Visual MODFLOW Main Menu select **[New]** in the **[File]** drop-down menu to create a new model. A Create New Model dialog will appear, as shown below. Select the directory where you want to save the model files and type in the new model name. Press **[Save]** to continue creating the new model. A warning will appear if the operation was unsuccessful (e.g., if the same name already exists).



Next, another input dialog will appear, as shown below. In this dialog you can assign a base map (\*.BMP, \*.DXF), select the default size of your grid (number and size of columns, rows, and layers), and select the working units for your model.



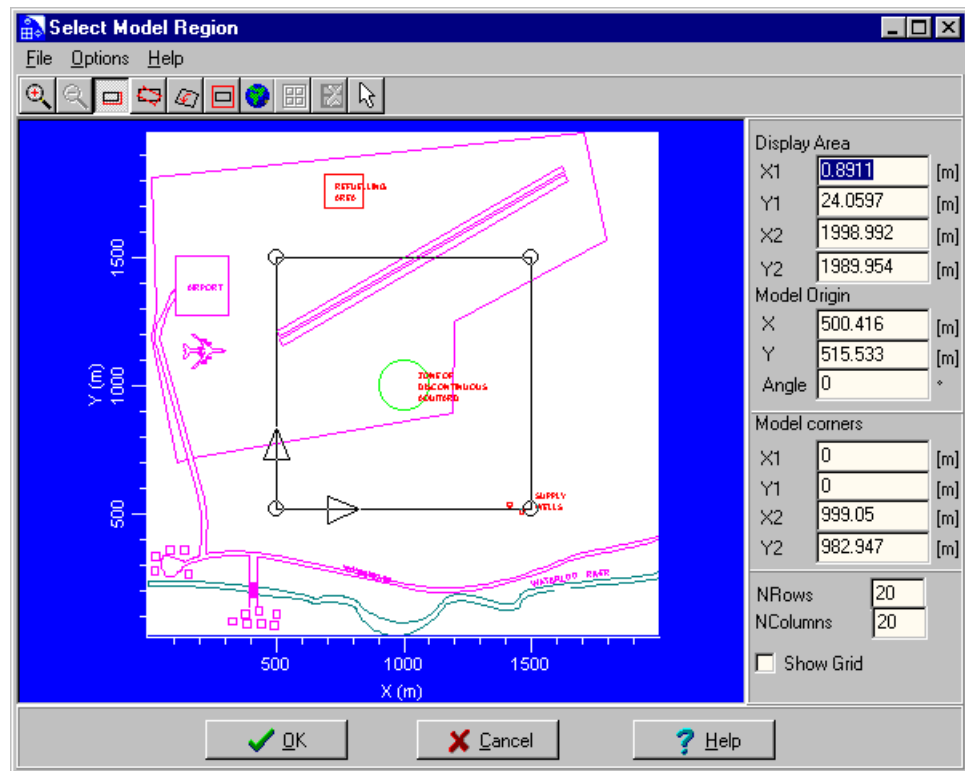
In the Model Domain section of the dialog box, you must enter the number of rows, columns and layers, as well as the minimum and maximum X, Y and Z co-ordinates for the model. If you have selected  **Create model using base Map**, the minimum and maximum X and Y co-ordinates will be taken from the file. Otherwise the minimum and maximum X and Y co-ordinates must be entered. Visual MODFLOW generates a uniform grid based on these dimensions. Later, the grid may be refined (please refer to the Grid section, in Chapter 3).

By selecting  **Setup Transport Module** you will be prompted with the Setup Numeric Engine dialog box. For more information on this procedure please refer to the section later in this chapter.

You must select units for your data set at this time. The units need not be consistent between SI and Imperial. For example, you can choose conductivity in cm/sec and pumping rates in US GPM. After you select the desired units, select [**Create**] to create the model.

## Base Maps

Visual MODFLOW supports the use of base maps in all modules of the program. These base maps can be either \*.DXF (AutoCAD v.14) or \*.BMP format. When  **Create model using base Map** is selected, another dialog appears, the Select Model Region dialog box.



In the Select Model Region dialog box, the following icons are common to both DXF and BMP files:



**Zoom In** Click and drag the mouse to select the zoom area.



**Zoom Out** By pressing the button once, the model region will return to its full extent.



**Resize Region** Allows the user to click on the corner of the region window (at each corner there is a circle, click and drag this) and drag it to the required size. Double clicking on the model region will switch the mode between Resize Region and Rotate Region.



**Rotate Region** Allows the user to click on a corner of the region window (at each corner there is a triangle, click and drag this) and drag it to the required angle. Double clicking on the model region will switch the mode between Rotate Region and Resize Region.



**Align Region** Aligns the model region with the x-axis.



**Full Region** Enlarges the model region to the full extents of the base map.



**New Map** A Select Map for Model dialog will appear, where you can specify a new DXF or BMP file for the base map.



**Pointer** Changes the mouse back to the normal cursor.

### Dual Co-ordinate System

Visual MODFLOW uses two co-ordinate systems, world co-ordinates and model co-ordinates. This allows the model to be rotated while maintaining a UTM based co-ordinate system. At any time you are able to view both the world and model co-ordinates of the mouse by opening the cell inspector (please refer to the **Cell Inspector** section, in this Chapter).

#### Transforming World Co-ordinates to Model Co-ordinates

To transform world co-ordinates to model co-ordinates, use the following formulas. There is no transformation for the Z co-ordinates of the model.

$$X_{model} = (X_{world} - X_{world-origin}) \cdot \cos\theta + (Y_{world} - Y_{world-origin}) \cdot \sin\theta + X_{model-origin}$$

$$Y_{model} = -(X_{world} - X_{world-origin}) \cdot \sin\theta + (Y_{world} - Y_{world-origin}) \cdot \cos\theta + Y_{model-origin}$$

#### Transforming Model Co-ordinates to World Co-ordinates

$$X_{world} = (X_{model} - X_{model-origin}) \cdot \cos\theta - (Y_{model} - Y_{model-origin}) \cdot \sin\theta + X_{world-origin}$$

$$Y_{world} = (X_{model} - X_{model-origin}) \cdot \sin\theta + (Y_{model} - Y_{model-origin}) \cdot \cos\theta + Y_{world-origin}$$

Where,

$X_{\text{model}}$	=	displayed X model co-ordinate
$X_{\text{model-origin}}$	=	model X start point (bottom left corner)
$X_{\text{world}}$	=	displayed X world co-ordinate
$X_{\text{world-origin}}$	=	world X start point (bottom left corner)
$Y_{\text{model}}$	=	displayed Y model co-ordinate
$Y_{\text{model-origin}}$	=	model Y start point (bottom left corner)
$Y_{\text{world}}$	=	displayed Y world co-ordinate
$Y_{\text{world-origin}}$	=	world Y start point (bottom left corner)
$\theta$	=	angle of rotation

### **DXF Base Maps**

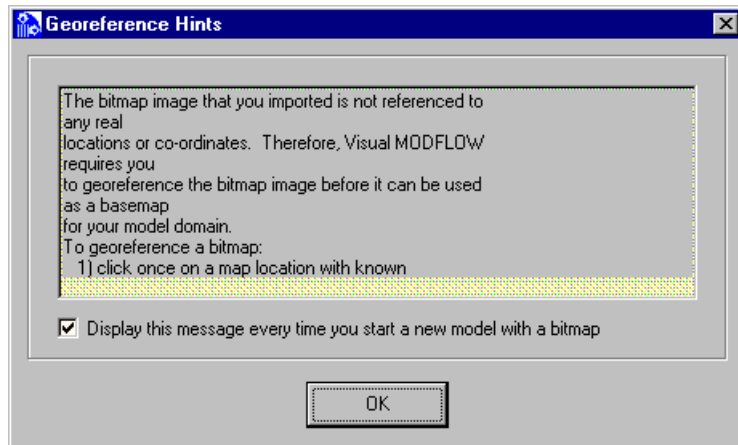
When using DXF files as base maps, you can resize the region with either the mouse or by typing values in the text boxes. The outside co-ordinates of the image are read directly from the DXF file and placed in the Display Area text boxes. The Model Origin that is entered is the world location of the origin of the model. The Model Corners are used as the grid or local co-ordinate system when assigning properties and displaying the model bounds. Lastly, the user is allowed to change the number of rows (NRows) and columns (NColumns) and show the grid. After entering the general model setup information, you will proceed to the Grid screen in the Input Module.

Some of the limitations associated with importing DXF files are:

- BLOCK information, color shaded items, and symbols cannot be imported;
- FROZEN layers must be THAWED; and
- Import single solid line types.

### **BMP Base Map**

When using a BMP file for the base map, the BMP file must be geo-referenced because there is no co-ordinate information in the file. The following dialog will appear advising you on how to geo-reference the image.



Before setting the model region, the BMP image must have two reference points with known world co-ordinates. To set the geo-reference points, click the mouse on a point where you know the world co-ordinates. An input box will appear, asking for the co-ordinates. Repeat for the second geo-reference point. Once both points have been accepted, the remaining text boxes become active to set the model origin, angle of rotation, model corners, number of rows and columns, and the option to show the grid.

The following icons are active when geo-referencing the image:



#### **Set Georeference Point**

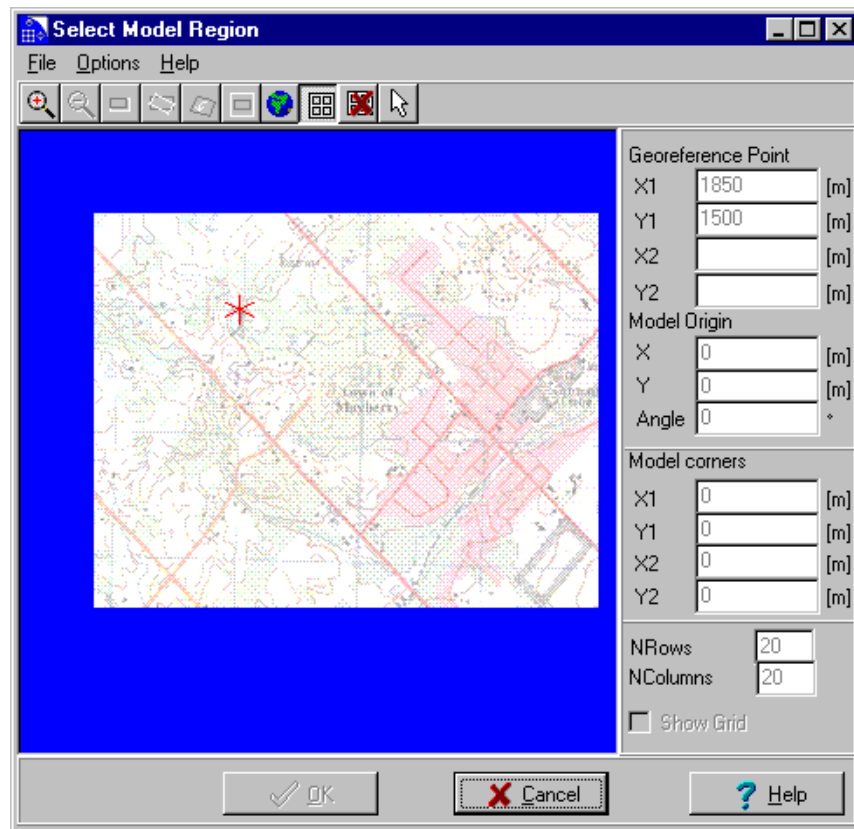
Click on the button and then click on the image to set a geo-reference point. A red star will appear to indicate the geo-reference point.



#### **Delete Georeference Point**

Click on the button and then click on the red star (the geo-reference point) you wish to delete. A dialog will appear to verify you wish to delete the geo-reference point.





When using BMP files as base maps, you can resize the region with either the mouse or by typing values in the text boxes. The co-ordinates of the image are read from the georeferenced points that were assigned. The Model Origin is the world location of the model origin. Grid co-ordinates or Local co-ordinates are used when assigning properties and displaying the Model Corners. Lastly, the user is allowed to change the number of rows (NRows) and columns (NColumns) and show the grid.

After entering the general model setup information, you will proceed to the Grid screen in the Input Module. xxxxxx

## Visual MODFLOW Printing Options

---

There are currently two different types of printing dialogs within Visual MODFLOW. The first type of printing dialog is new and can be accessed from the 2D IChart graphs. The second type of printing dialog is the old Visual MODFLOW printing dialog and can be accessed from the top menu bar **[File] [Print]** option any one of the modules.

### Type 1 Printing – IChart Graphs



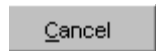
The screen display for the IChart graphs can be printed by selecting **[File] [Print]** or by pressing the **[Print]** shortcut button from the top menu bar in any of the plots. In Visual MODFLOW, the Print dialog appears as shown in the following figure. It contains settings for the printer setup, the page layout, and title blocks.



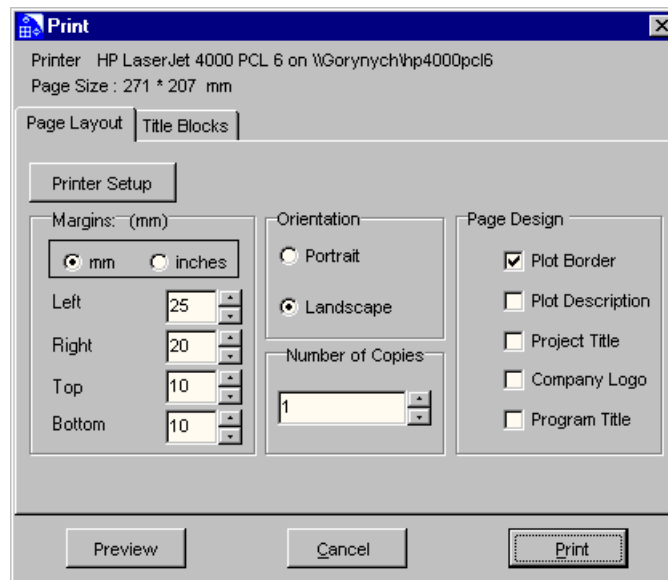
Press the **[Print]** button to accept the settings and print the screen display to the selected printer.



Press the **[Preview]** button for a preview of the how your page will look when it is printed.



Press the **[Cancel]** button to close the **Print** dialog without printing.



### Printer Setup

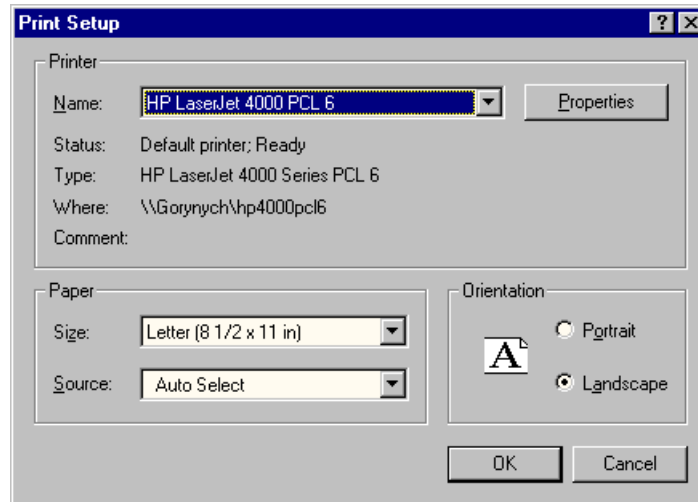
The selected printer and page size are displayed at the top of the Print dialog box as shown in the above figure.

Printer Setup

The printer settings, paper size and source, and page orientation can be modified by pressing the **[Printer Setup]** button, on the Print dialog box, to open the Print Setup dialog as shown in the following figure.

Properties

Pressing the **[Properties]** button, on the Print Setup dialog can access more advanced printer settings such as print quality, watermarks, reduce/enlarge, number of copies, and paper handling options.



## Page Layout

The **Page Layout** tab contains settings for defining the position of the plotted graphics on the page, the page orientation, the number of copies, and designing the page form.

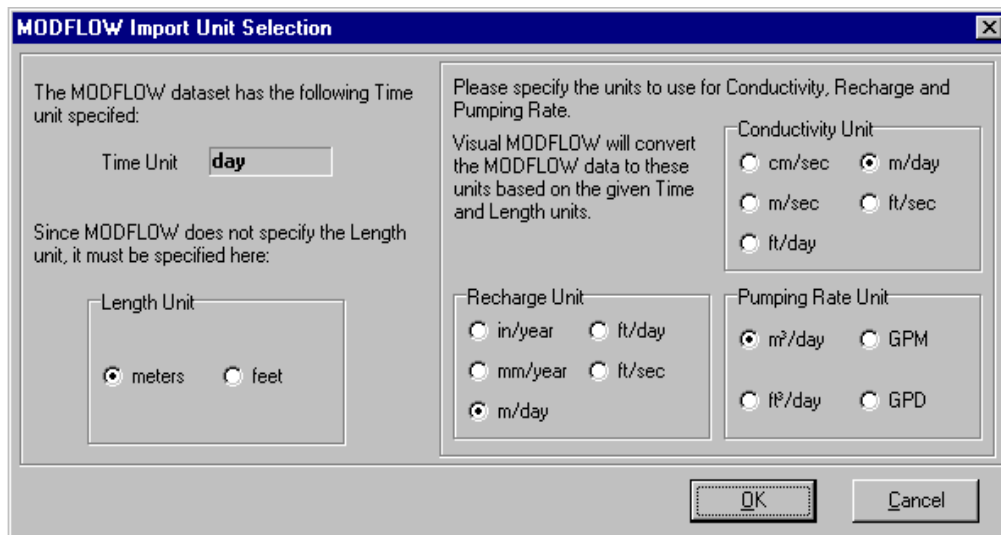
### Margins

The margins define the blank region around the outside of the page. If the figure is being included in a bound report, it is recommended that the margin on the bound edge be slightly larger than margin on the opposite edge. The margins can be set using either  **mm** (millimeters) or  **inches**. A minimum margin of 5 mm will be applied for any margin settings less than 5 mm.

### Page Design

The Page Design settings allow you to design the print template by selecting any or all of the available components to include with the plotted image.

- Borders**      Displays a thin outline along the page margins. Displays a thin outline around each title block when the component is selected.
- Description**      Shows the description of the figure being printed. The description is typed, by the user, in the Description text field.

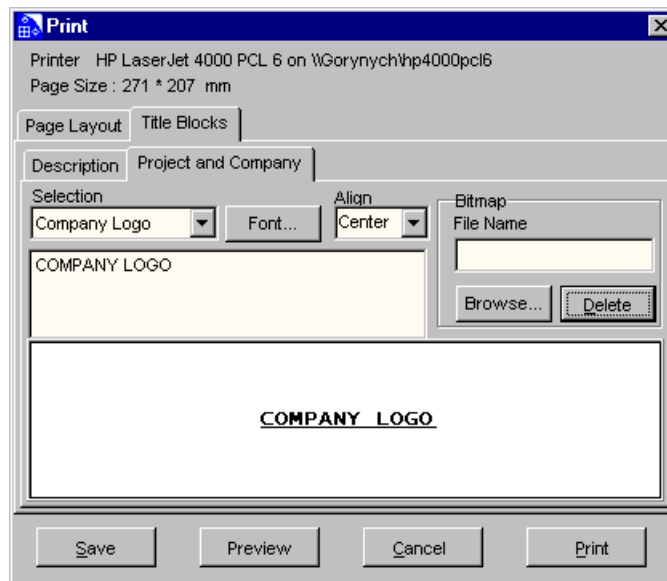


- Project Title** Shows a description of the project and/or client with an optional bitmap image (e.g. client's company logo). By default the current date and time is included in this field.
- Company Logo** Displays a description of your company with an optional bitmap image of your company logo.
- Program Title** Displays a Visual MODFLOW descriptor stating:
 

Waterloo Hydrogeologic Inc.  
Visual MODFLOW 2.8 © 1995 – 1999

### Title Blocks

The **Title Blocks** tab, as shown in the figure below, is used to design the Project Title, Company Logo, and Description title blocks.

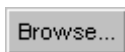


The **Selection** field, found in the top right corner of the Project and Company tab, contains a drop-down menu to choose either the **Company Logo** or **Project Title** title block for editing.

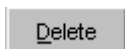
The **[Font]** button is used to set the single font size, style and color for the entire selected title block.

The **Align** field is used to set the alignment (**Left**, **Right**, or **Center**) for the entire text in the selected title block.

The text field, directly below the **[Font]** button, contains the text that will appear in the selected title block. To add text, simply click the mouse pointer in the field and begin typing. This text field can have several lines depending on the size and style of the font. The field on the lower half of the Project and Company tab allows the user to view the title block as it will appear on the page without pressing the **[Preview]** button.



The **Bitmap** field is used to select a single (only one) bitmap file (**.BMP**) to display on the left-hand side of the selected title block. To load a bitmap you can type the path for the bitmap directly into the File Name text field, or you can select the **[Browse...]** button to browse your computer for the appropriate filename. Please note the graphic file must be a bitmap file with a **.BMP** file extension. If the bitmap image already includes text, then you may not need to add text in the selected title block.



The **[Delete]** button is use to remove a bitmap image from the selected title block.

Click on the **Description** tab to edit the **Description** title block. To add text, simply click the mouse pointer in the text field and begin typing. Press on the **[Font]** button to

edit the font size, style and color for the entire title block. The **Align** field is used to set the alignment (**Left**, **Right**, or **Center**) for the entire text in the title block. To preview the **Description** title block, as it will appear on the page, it must be selected on the Page Design tab before pressing the **[Preview]** button.



The **[Save]** button is used to save the title block settings for future use. The Project Title settings are stored in the project **.INI** file such that all plots from the current project will have the same Project Title settings. The Company Logo settings are stored in the Visual MODFLOW **.INI** file such that all plots for any project will have the same Company Logo settings.

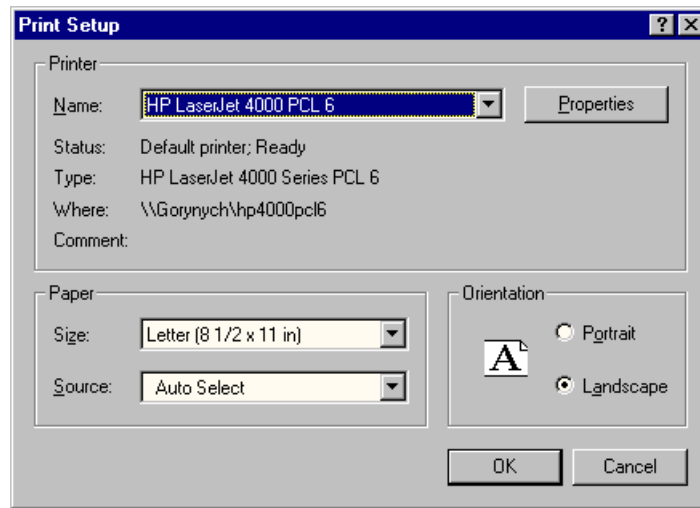
## Type 2 Printing – Model Domain Screen

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

A screenshot of a Windows-style dialog box titled "Print". It has a blue title bar. The dialog is divided into several sections: "Current Printer" with a text field showing "HP LaserJet 4000 PCL 6 on \\Gorynych\hp4000pcl6" and a "Select..." button; "Project Information" with fields for "Project:", "Description:", and "Modeller:", and a "Date:" field with "11-3-1999"; "Margins" with four input fields for "Left:", "Right:", "Top:", and "Bottom:", each containing "1.00"; and a "Units:" section with radio buttons for "inches" (selected) and "cm". There are also two checkboxes: "Full Image" and "Color Legend", both of which are unchecked. At the bottom, there are three buttons: "Print", "Cancel", and "Help".

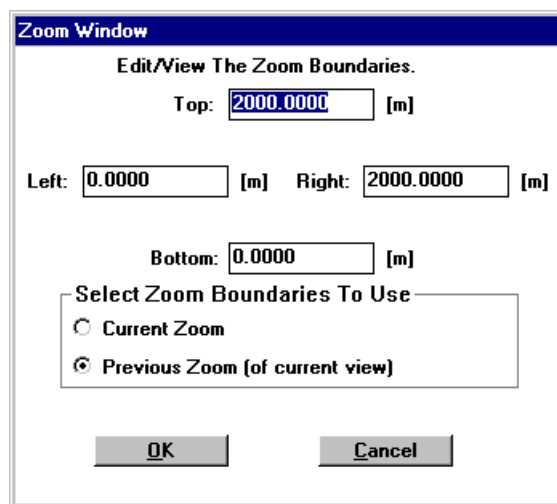
Enter in the project information to provide the text for the title block. When  **Full Image** is toggled on, the title block will not be printed.

The default Windows printer is automatically selected as the current printing device. To change the current printer, click on **[Select...]** and the following dialog will appear.



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

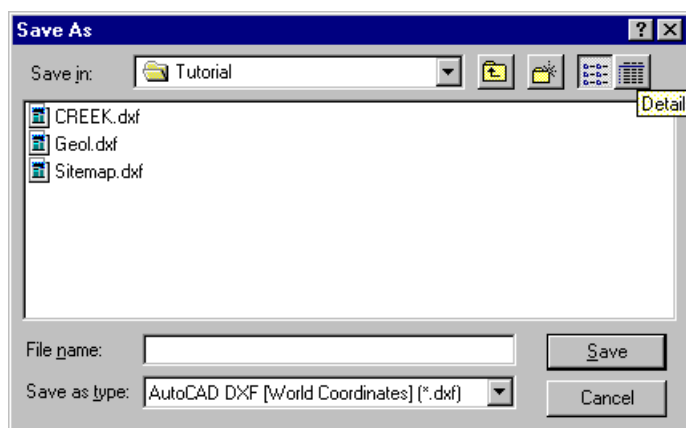
The Zoom Window dialog, which can be displayed by right clicking on the model domain after clicking on the **[F5-Zoom In]** button, allows you to enter a set of grid dimensions for the zoom window. For the specified zoom window, Visual MODFLOW first attempts to scale the rectangular viewing area to the left and right dimensions. If the top and bottom coordinates are not visible then Visual MODFLOW scales the viewing area to the top and bottom dimensions. The current zoom and previous zoom options display the current or previous viewing area coordinates respectively.



## Printing to a File

Visual MODFLOW allows you to export the model domain you are currently viewing to a **.DXF** file for post processing in a CAD program such as AutoCAD.

To export to a **.DXF** file, select **[File]** from the top menu bar and then select **[Export...]**. The following dialog will appear, prompting you for the filename. In the **Save as Type**, there is the option to save the AutoCAD DXF file (**\*.DXF**) using either the world co-ordinates or the model co-ordinates. There is also the option to save as an Enhanced Windows Metafile (**\*.EMF**) or an XYZ ASCII file (**\*.ASC**, **\*.TXT**, **\*.XYZ**) in world or model co-ordinates, depending on the Module.



## Numeric Engines

---

The MODFLOW version included in Visual MODFLOW is the USGS MODFLOW 96, compiled by Waterloo Hydrogeologic for 32 bit applications in Windows NT/95/98/2000. Waterloo Hydrogeologic Inc. has added the WHS Solver and a calibration package for optimizing output of calibration results to the original MODFLOW code.

The MODPATH and Zone Budget versions included in Visual MODFLOW are the USGS versions, compiled for 32 bit applications in Windows NT/95/98 by Waterloo Hydrogeologic Inc.

The following Transport Numeric Engines are supported in Visual MODFLOW:

- DoD MT3D, v1.5 (Public Domain);
- MT3D 96 by S.S. Papadopulos & Associates, Inc.;
- RT3D by T.P. Clement (Public Domain);



- DoD MT3D MS v3.00.A (Public Domain); and
- MT3D 99 by S.S. Papadopoulos & Associates, Inc.

The user should be familiar with the theory of groundwater flow and mass transport, including the finite-difference methods, explicit and implicit numerical methods, and the structure of the MODFLOW, MODPATH and MT3D codes before running the models. **This is the responsibility of the user.**

Basic texts covering the theory of groundwater flow and mass transport include Freeze and Cherry (1979), Domenico and Schwartz (1990), and Fetter (1993). Additional information on the method of characteristics may be found in Zheng (1993). The user should be aware of the strengths and weaknesses of MODFLOW/MT3D when deciding whether it may be appropriate for a particular problem. The user is also strongly encouraged to read each of the transport numeric engine manuals. In the next section there is a brief overview of the transport numeric engines in Visual MODFLOW.

## **Transport Engines**

Each of the Transport Numeric Engines have specific sorption and reaction functions. The following figure outlines the sorption and reactions packages that are available to each of the Transport Engines included with Visual MODFLOW.

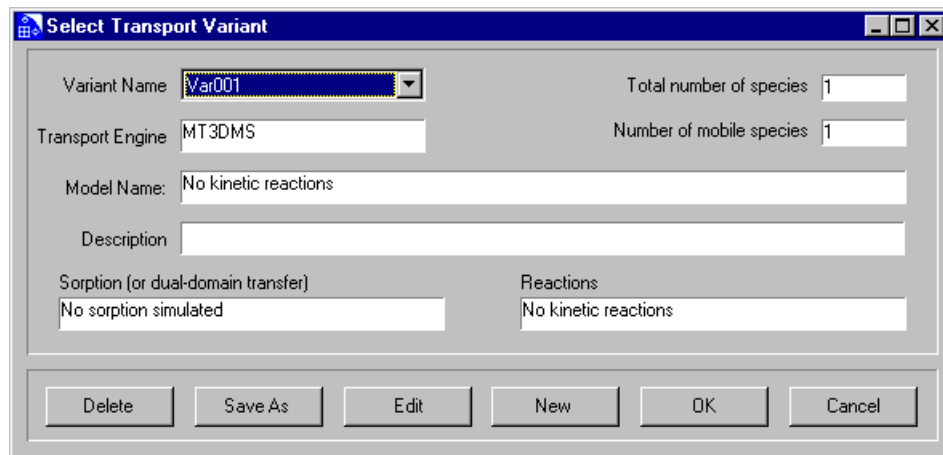
**Table 2.1: Summary of sorption and reaction options available to each of the Transport Engines**

		MT3D150	MT3DMS	MT3D99	RT3D	MT3D96
<b>Sorption Options</b>	No Sorption is simulated	✓	✓	✓	✓	✓
	Linear Isotherm (equilibrium-controlled)	✓	✓	✓	✓	✓
	Freundlich Isotherm (equilibrium-controlled)	✓	✓	✓	✓	✓
	Langmuir Isotherm (equilibrium-controlled)	✓	✓	✓	✓	✓
	First-order kinetic sorption (non-equilibrium)		✓	✓		
	First-order kinetic dual-domain mass transfer		✓	✓		
<b>Reactions Options</b>	No kinetic reaction	✓	✓	✓	✓	✓
	First-order irreversible decay	✓	✓	✓		✓
	Monod kinetics			✓		
	First-order parent-daughter chain reaction			✓		
	Instantaneous reactions among species			✓		
	Instantaneous aerobic decay of BTEX				✓	
	Instantaneous multi-path degradation of BTEX				✓	
	Kinetic limited multi-path degradation of BTEX				✓	
	Rate-limited sorption reactions				✓	
	Double-Monod degradation model				✓	
	Sequential decay reactions				✓	
	Aerobic/anaerobic model for PCE/TCE degradation				✓	

## Transport Engine Options

Visual MODFLOW Version 2.8.2 allows you to have multiple transport scenarios (also referred to as variants) referenced to a single MODFLOW simulation. This allows you to more efficiently manage your transport calibration and analysis.

When you select [**Setup**] from the Main Menu and then [**Numeric Engines**] from the drop-down menu, the following dialog will be displayed.



This dialog shows you the basic information for the currently selected transport variant and allows you to switch to another variant if you have more than one variant defined. The buttons along the bottom of the dialog function as follows:

**[Delete]** Removes the variant currently selected permanently.

**[Save As]** Creates a new variant with the settings of the current variant.

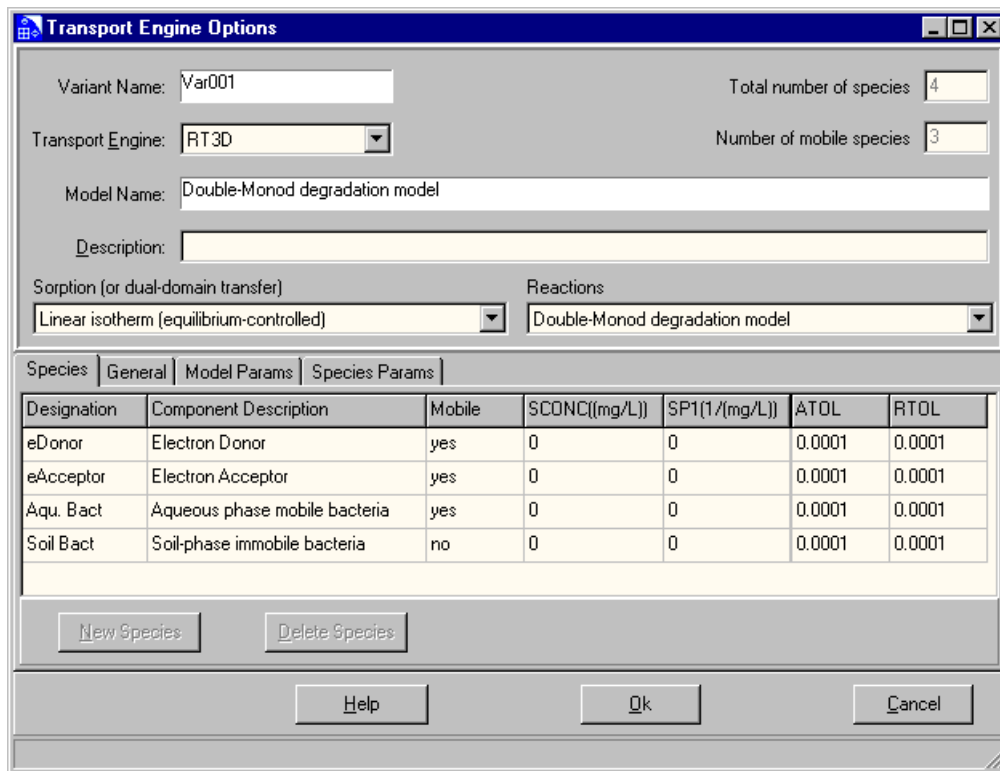
**[Edit]** Allows you to alter the currently selected variant.

**[New]** Creates a new variant with default settings.

**[OK]** Accepts the transport variant selection and settings, and returns you to the Main Menu.

**[Cancel]** Returns you to the Main Menu without changing the variant or the settings.

If you press either the **[Edit]** or **[New]** button the following dialog will appear.



In the upper half of the Transport Engine Options dialog, you can select which transport engine (FORTRAN executable) you wish to use, edit the variant description, and select from the available sorption and reaction modules. The available sorption and reaction modules depend on which transport engine is selected. These options are summarized in Table 2.1 on page 24.

The lower half of the Transport Engine Options dialog contains four tabs: **Species**, **General**, **Model Params**, and **Species Params**. The reference documentation for each of the numerical engines includes detailed descriptions of the parameters referenced on these tabs. The parameter names and descriptions have been chosen to make it easy to cross-reference parameters with the technical documentation. All the technical documentation can be found on the Visual MODFLOW Installation CD-ROM in an Adobe Acrobat .PDF file format.

This dialog represents the initial default values for a new variant. Many of the parameter values listed on these tabs are also editable in the Input mode. If the values are edited in the Input module, any subsequent changes in the Variant Options dialog will be ignored.

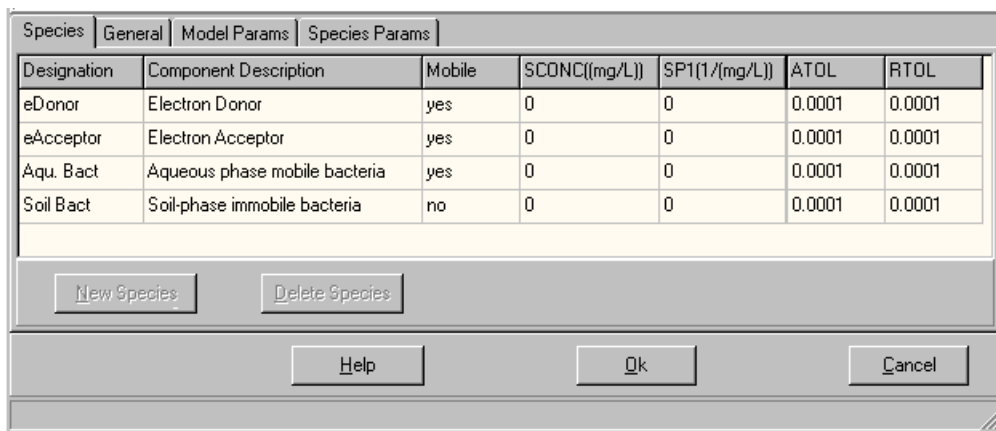
## Species Tab

The Species Tab contains a list of current species in the variant. For multi-species engines (MT3DMS and MT3D99) you can add or delete species from the list. RT3D is a special case because the reaction model that you select defines the list of species. If no reaction model is selected then you can also add and delete species from the species list for RT3D.

The species Designation and Description can be edited in all cases, except for the predefined RT3D models.

The available parameters for each species depends on the transport engine selected. For example, in MT3DMS, each species is defined as either Mobile or Immobile, has an initial default concentration (SCONC), and initial default values for sorption (SP1 and SP2) and reactions (RC1 and RC2), when sorption or reactions are selected.

All multi-species models must have the mobile species written to the input files first, followed by the immobile species. Therefore, when entering species verify that their order reflects this constraint. Immobile species must always follow Mobile species.



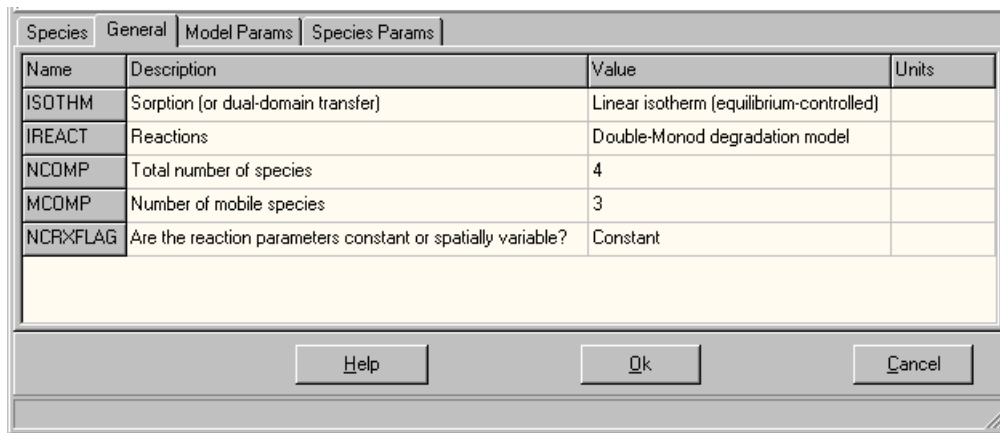
The screenshot shows a dialog box titled "Species" with four tabs: "General", "Model Params", "Species Params", and "Species". The "Species Params" tab is active, displaying a table with the following data:

Designation	Component Description	Mobile	SCONC((mg/L))	SP1(1/(mg/L))	ATOL	RTOL
eDonor	Electron Donor	yes	0	0	0.0001	0.0001
eAcceptor	Electron Acceptor	yes	0	0	0.0001	0.0001
Aqu. Bact	Aqueous phase mobile bacteria	yes	0	0	0.0001	0.0001
Soil Bact	Soil-phase immobile bacteria	no	0	0	0.0001	0.0001

Below the table are buttons for "New Species" and "Delete Species". At the bottom of the dialog box are buttons for "Help", "Ok", and "Cancel".

## General Tab

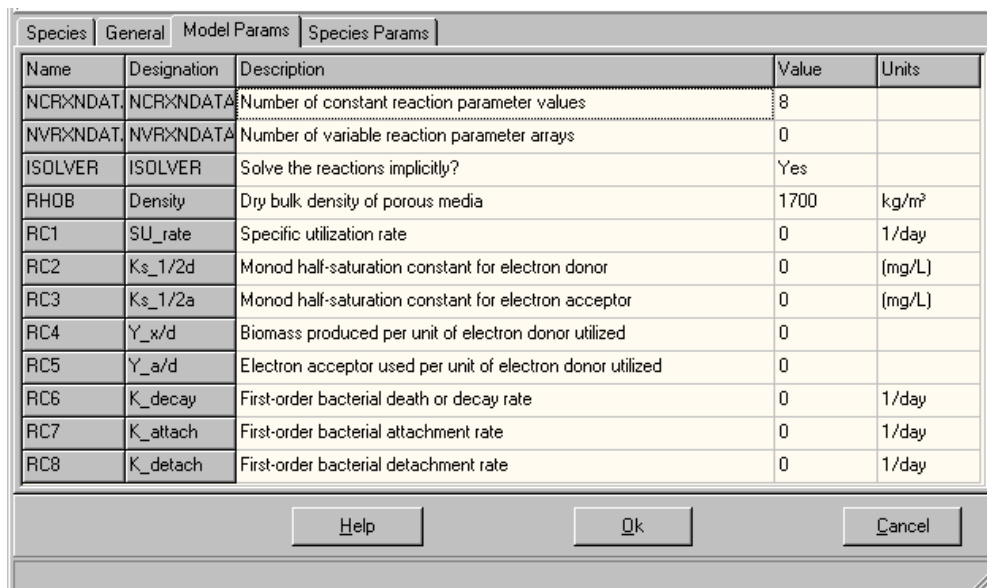
The General Tab contains basic information for the variant. This tab states the total number of species and the number of mobile species, as determined from the Species Tab. This tab states the reaction/sorption models selected from the combo boxes and allows the user to re-select the reaction/sorption model they wish to use. On this tab, when RT3D is selected, the user can choose whether or not the reaction parameters are constant or spatially variable. **If the RT3D reaction parameters are selected to be spatially variable, then spatially distributed zones can be defined in the [Model Params] mode in [Input].**



### Model Params. Tab

The Model Params. Tab is dependent on the transport engine and reaction/sorption options selected by the user. From the Model Params. Tab the user is able to edit the reaction constants associated with the reaction model.

In the following dialog, the user has specified RT3D with Model 5. The first two variables are taken from the Species Tab and are not editable. Next, the ISOLVER variable is editable, but there is only one choice for Model 5. This is followed by the initial default value for the Bulk Density. The eight reaction parameters are only editable here unless they are defined as spatially variable in the General Tab.



## Species Params. Tab

The Species Params. Tab further details the sorption constants and reaction parameters required for the reaction/sorption model selected for the variant. The Species Params. Tab displays the same information as the Species Tab but instead is organized by parameter rather than by species. It is possible to edit the parameter values in either the Species Tab or the Species Params. Tab.

Name	Designation	Parameter Description	Units	eDonor	eAcceptor	Aqu. Bact	Soil Bact
SCONC	SCONC	Initial concentration	(mg/L)	0	0	0	0
SP1	Kd	Distribution coefficient (Kd)	1/(mg/L)	0	0	0	0
ATOL	ATOL	Absolute Tolerance		0.0001	0.0001	0.0001	0.0001
RTOL	RTOL	Relative Tolerance		0.0001	0.0001	0.0001	0.0001

## Variants

In Visual MODFLOW 2.8.2 variant information is saved in the *project.vmf* file. For each variant a file called *project.var001* is saved to the working directory where *var00#* is the variant number. When entering species specific information in the Input Module, the files containing the transport information are created in the working directory. When a new variant is created the current information from the Input Module is compressed and saved to the *project.var00#* file. All species specific information in Input will be reset for the new variant to the default values entered in the Transport Engine Options dialog. As you move between your variants, the input and output for the variants is compressed and uncompressed to and from the variant repositories.

The reason for this data structure is to allow multiple scenarios for contaminant transport using the same flow model. This allows different remediation alternatives to be examined given an identical set of flow conditions.

For example, say you have a complex, transient calibrated flow model and you want to simulate the transport of BTEX. To simulate the transport over a ten year period, you might initially set up your model using MT3DMS and a single species. This would allow you simulate the transport using the new and very fast GCG solver included with MT3DMS. Next, your client may want you to determine the impact of intrinsic remediation using RT3D. If you create a new variant, your transport input and output

files will be compressed into a variant repository file (*project.var001*). The information for the new variant can now be entered. If you switch back to the initial variant, the information from variant 2 will be compressed in a second variant repository (*project.var002*) and the *project.var001* will be extracted.

### **Variants and Model Translation**

When a model is translated only the files from the active variant are used for translation. All dormant variants are invisible to the transport engine.

### **Variants and Altering the Flow Model**

After creating a number of variants it may be necessary to alter the structure of the flow model (e.g. you may wish to refine the grid). These types of changes to the flow model will only affect the active variant. Once these changes are made, you will receive an error message upon opening an old variant, due to the incompatibility between the old transport variant and the new flow model.

When the variant information is condensed and saved to the repository, the species information is save in arrays of size  $x, y, z$ . Once the model is refined, the active array size changes to  $x+x_i, y+y_i, z+z_i$ . The old variant, once extracted from the repository, will no longer fit the correct array size. This error does not occur for the active variant because the working files are written to the working directory and thus updated. It is the invisible variant conditions that are not updated because they have been filed to the repository.

Contaminant transport simulations can become sophisticated. Losing the information due to a poorly defined flow model is time consuming. **It is recommended that the user wait until the flow model is mature and stable before experimenting with different transport scenarios using multiple variants.**

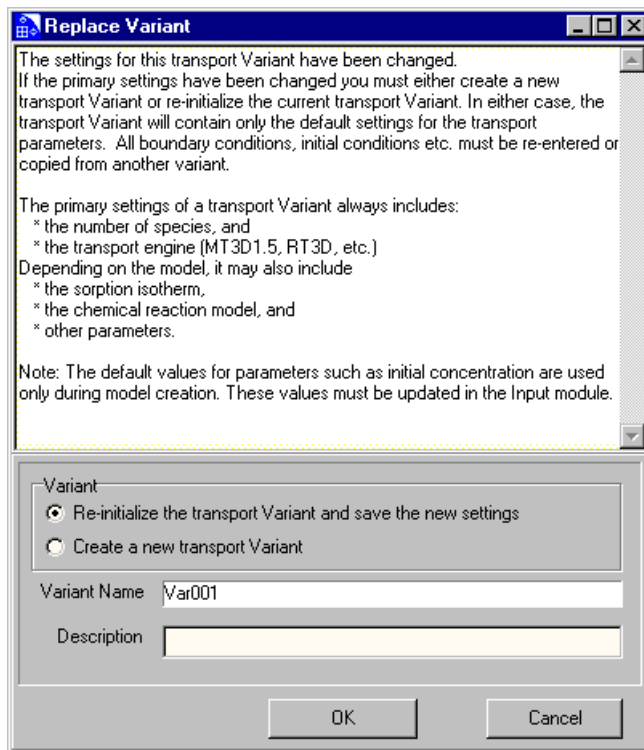
### **Conditions for Re-Initializing a Variant**

Any of the following conditions will cause the variant to be re-initialized.

- Changing the transport engine
- Changing the sorption method
- Changing the reaction
- Adding or deleting species

If a variant is to be re-initialized, Visual MODFLOW will present you with the following dialog.





The user will have two options: to re-initialize the transport variant or to create a new transport variant. Regardless of which option is selected, the species specific information in the Input Module will be re-initialized. To avoid re-initializing the Input Module, press **[Cancel]** on each window until you find yourself in the Main Menu.

### Copying a Variant

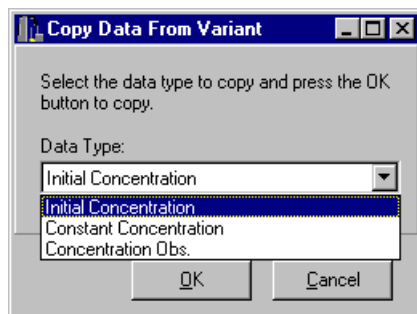
If changes are to be made to the transport engine, the sorption method, the reaction or species are to be added or deleted, it may be desirable to copy a previously defined variant. This will prevent the need to re-enter variant information which is required when the variant is re-initialized or a new variant created.

To copy a previously defined transport variant:

From the **Replace Variant** dialog box:

Choose **Ⓒ Create a new transport Variant** by selecting the radio button. A new variant name will appear in the Variant name field. Click **[OK]**

To copy existing variant information to the newly defined variant select **[Setup] [Copy from Variant]**. The following dialog box will appear:



From here the variant data to be copied may be selected. Highlighting the desired variant component and clicking [OK] will bring up the following dialog:



From the pull down box, the variant to be copied may be selected. By selecting the three different variant data components for the **Copy Data from Variant** dialog box, all the components of the “old” variant can be copied to the new variant.

### Re-initializing the Transport Variant

Re-initializing the transport variant will save all the changes made within the Transport Engine Options dialog to the same variant name. This will overwrite all of the information contained in the variant repository and reset the species specific information within the Input Module.

### Creating a New Transport Variant

Creating a new transport variant will perform two tasks. One, all of the old variant information will be saved and condensed to its variant repository (*project.var001*).

Two, the new variant will be created, re-initializing the species specific portions of the Input Module.

## Common Tools

---

There are several tools that are common to more than one module. They can be accessed through either the Input, Run, and/or Output Modules. These tools are:

- Overlay Function,
- IChart, and
- Cell Inspector.

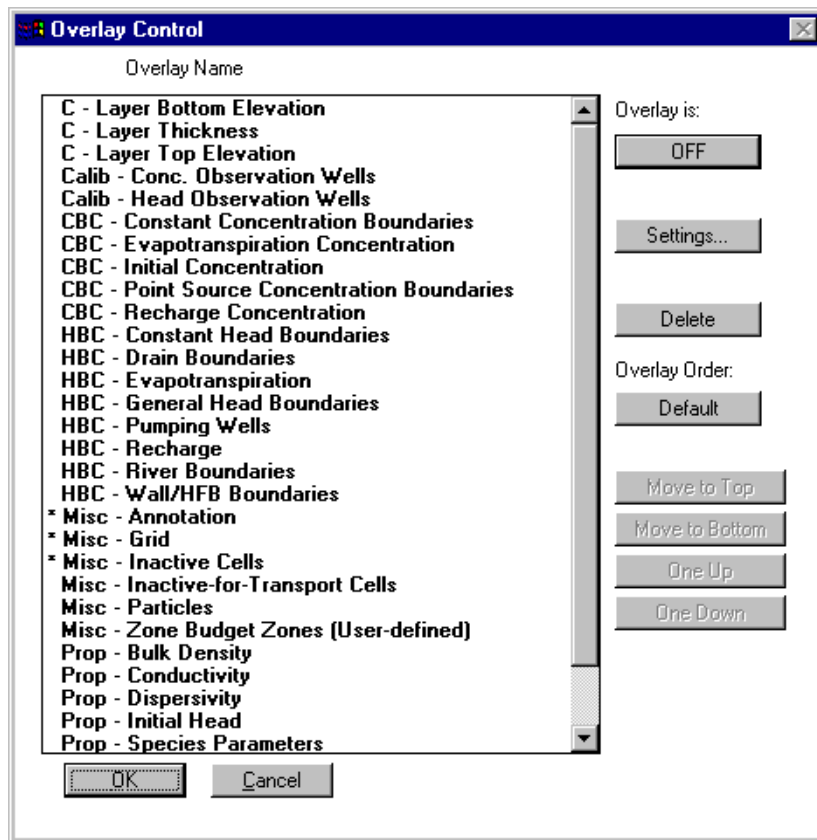
### Overlay Function Features

The Visual MODFLOW display is organized as a series of layers. The various features, such as gridlines, observation wells and conductivity zones, are drawn independently and can be turned on or off individually to make your plots or screen more readable. These various layers are called overlays.

The overlays with your results are only available in the Output Module. Otherwise, the Input and the Output overlays are identical.

When you move between Main Menu, Input, Output, and Run there are particular defaults used for the overlays. In the Main Menu only the background **.DXF** and **.BMP** overlays are displayed along with the model boundaries. The Grid Overlay is added by default when you first enter the Input Menu, but as you navigate through the Input items, the default overlays are context sensitive. In Output, the default overlays are also context sensitive but as you navigate, the previous overlays remain visible unless explicitly turned off.

The Overlay dialog appears when F9 is pressed or when the **[F9 – Overlay]** button at the bottom of the screen is clicked on with the cursor. The Overlay dialog contains a list of all the overlays available in the Module. Select an overlay by moving the cursor overtop of the overlay name and click once. The overlay will be highlighted by a blue line. If the overlay is turned on, an asterisk will be to the left of the overlay name and the top button on the dialog will read **[On]**. To turn an overlay on either double click on the overlay name (an asterisk will appear) or select the overlay and press the top button until it reads **[On]**. To turn an overlay off either double click on the overlay name to remove the asterisk or select the overlay and press the top button until it reads **[Off]**.



The default overlay list is alphabetical but grouped by overlay type. The following list outlines the meaning of the prefixes for the overlay types.

- C** - Contour
- Calib** - Calibration Plots
- CBC** - Concentration Boundary Condition
- HBC** - Head Boundary Condition
- Misc** - Miscellaneous
- Prop** - Property
- Results** - Output Overlays

**Note:** Map files (i.e. \*.DXF and \*.BMP files) have no prefix and appear last in the groupings.

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

The various overlays allow you to cross check your data or to add or remove features from your plots to make them more readable. Maps that are imported can also be deleted from the Overlay dialog box by pressing the **[Delete]** button.

Overlay order does not reflect the drawing order. To see and modify the drawing order, toggle the **[Default]** button to **[User Defined]**. The Overlay list can be modified using the group of four buttons located on the lower right hand side of the Overlay Control dialog box. To modify the order, simply select an active overlay and press one of the four buttons: **[Move to Top]**, **[Move to Bottom]**, **[One Up]**, or **[One Down]**.

## **IChart Plotting Properties**

Visual MODFLOW uses a charting component, IChart, for all plots that have been developed by Waterloo Hydrogeologic. IChart allows the user to customize a graph's properties. Although there may be subtle differences in the appearance of the various plot windows, IChart will maintain its features.

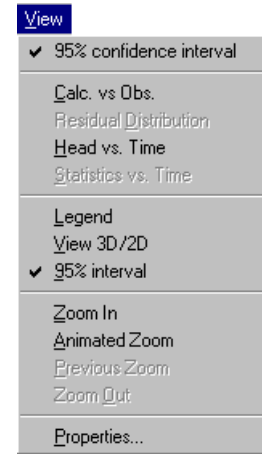
IChart can be found in the following sections of Visual MODFLOW:

- [Input][Grid][Edit Elevations][**Grid by Row**] or [**Grid by Column**]
- [Input][Wells][Pumping Wells][**Graph**]
- [Run][**Run MODFLOW**]
- [Run][**Run ZoneBudget**]
- [Output][Head Equipotentials][**Graphs**]
- [Output][Drawdown][**Drwdn vs. Time**]
- [Output][Concentration][**Graphs/Conc. vs. Time**]
- [Output][Budget][**Flow/Mass Budget/Balance Graphs**]

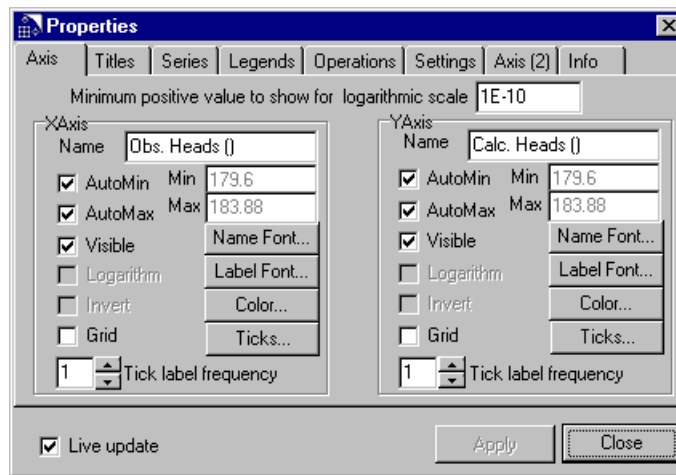
To edit IChart plotting properties, from these sections, simply,



- right click on the plot area and select the **[Properties]** option on the pop-up menu,
- press the **[Properties]** button in the toolbar, or
- select **[View][Properties]** from the top menu bar of the plot window.



Next, the Properties dialog will appear as shown below.



The Properties dialog consists of eight tabs,

- [Axis]** Used to edit axis names and range of axis values, format axis and grid, and show or hide tick marks, for the primary axis.
- [Titles]** Allows you to show or hide titles and change and format the graph title and plot footer.
- [Series]** Allows you to select, format, and set the order of the series appearing on the graph.
- [Legend]** Allows you to show or hide legend and format legend type, font, frame, and position relative to plot area.
- [Operations]** Used to add new series by performing mathematical operations on existing series.
- [Settings]** Used to add a bitmap to the plot area background and format plot area, plot background, or plot frame.

**[Axis (2)]** Used to edit axis names and range of axis values, format axis and grid, and show or hide tick marks, for a secondary axis.

**[Info]** Allows the user to select where, how, and what information is displayed in the data point information bubble.

There are several features with this dialog that remain constant regardless of the tab selected. These are:

**Live Update** When turned on (indicated with a ✓) any change made to the property settings will be updated simultaneously.

**[Apply]** Press to update the plot area to reflect any changes made to the property setting. This button is deactivated when  **Live Update** is selected.

**[Close]** Press to close the Properties dialog box.

### The Axis Tabs

To edit the graph axes, access the Properties dialog using one of the three possibilities mentioned above. The **Axis** tab is the default tab in the Properties dialog box, and is shown above. This tab allows the user to customize the general appearance, dimensions, color and content of the X and Y axes. The following is a list and description of all the options offered to edit the graph axes:

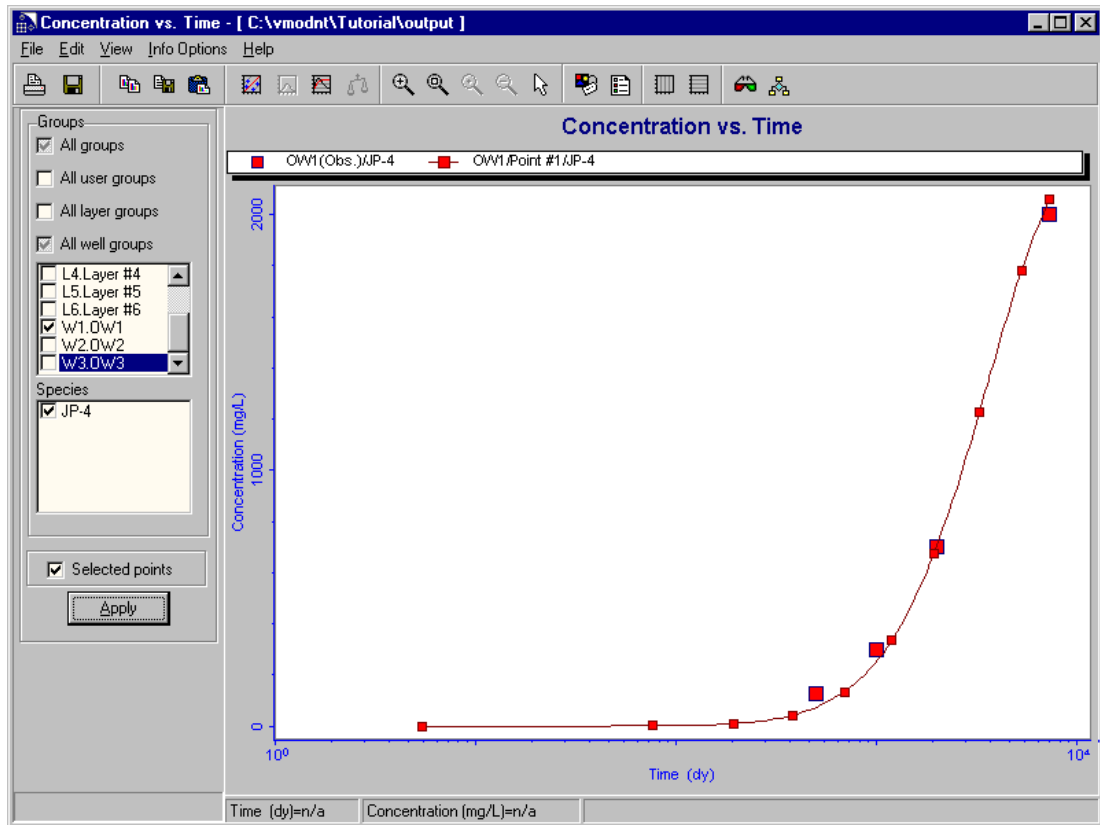
<b>Name</b>	Contains the name for the axis.
<input checked="" type="checkbox"/> <b>Automin/</b> <input checked="" type="checkbox"/> <b>Automax</b>	When turned on (indicated by a ✓), the minimum/maximum value for the axis is automatically set.
<b>Min/Max</b>	Contains the minimum/maximum value for the axis. Only when Automin/Automax is turn off, can these values be customized.
<input checked="" type="checkbox"/> <b>Visible</b>	Shows (indicated by a ✓) or hides the tick marks and labels for the X axis.
<input checked="" type="checkbox"/> <b>Logarithm</b>	When turned on (indicated by a ✓), a logarithmic scale is applied to the axis. When the minimum value is negative, it will automatically be set to the minimum positive value to show for logarithmic scale, specified by the user in the input box at the top of the <b>Axis</b> tab.
<input checked="" type="checkbox"/> <b>Invert</b>	When turned on (indicated by a ✓), the axis values become inverted. For instance, along the X axis the maximum value will be plotted on the left and the minimum value will be plotted on the right.

<input checked="" type="checkbox"/> <b>Grid</b>	Shows (indicated by a ✓) or hides the gridlines connected to the ticks along the axis.
<b>Inside Position</b>	Specifies where the X axis crosses the Y axis.
<b>Tick label frequency</b>	Sets the major ticks' labelling frequency. By default, every major tick is labelled; denoted by a frequency of one. A frequency of two, labels every second major tick.
<b>[Name Font...]</b>	Sets the font type, style, size, and color for the axis name. The default font is 8 point, regular Arial in blue. Use only True Type fonts for the Y axis.
<b>[Label Font...]</b>	Sets the font type, style, size, and color for the axis labels. The default font is 8 point, regular Arial in blue. Use only True Type fonts for the Y axis.
<b>[Color...]</b>	Sets the color of the axis line and tick marks.
<b>[Ticks...]</b>	Allows the user to show or hide the major and minor tick marks, set the tick mark length, and specify their direction with respect to the axis.

The following plot area is a graph from the **[Concentration] [Graphs]** option. It has been altered from the default setting of the Axis tab by:

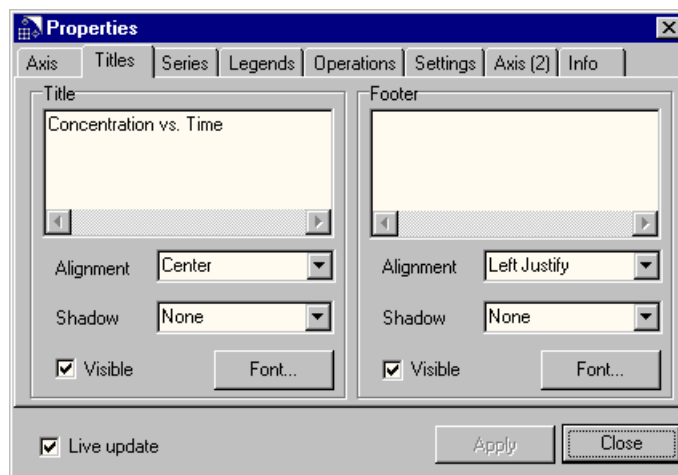
- [1] Turning off the  **Automin** option, and specifying a **Min** value of 1 for the X axis.
- [2] Turning on the  **Logarithmic** option for both the X axis and the Y axis.
- [3] Setting a **Tick label frequency** of 4 to the Y axis, to clear-up the labels for easier reading.





### The Titles Tab

To edit the graph titles, simply access the Properties dialog using one of the three alternatives mentioned previously. Once the dialog appears, select the **Titles** tab.

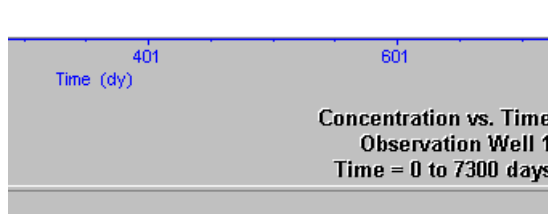


The **Titles** tab allows the user to customize the general appearance and content of the graph title and the plot footer. The following is a list and description of all the options offered to edit the titles:

- Title/ Footer** Contains the text for the plot title/ footer. The text will appear on the plot area exactly the way it appears in the text box. Additional lines of text can be added by pressing the <Enter> key. A live update of the changes can come to view by pressing the <Tab> key.
- Alignment** Places the title/footer in one of three options with respect to the plot: **Left Justify**, **Center**, or **Right Justify**.
- Shadow** Writes the text with a shadow in one of three positions: **Lowered**, **Raised**, or **None**.
- Visible** Shows (indicated by a ✓) or hides the plot title or footer.
- [Font...]** Sets the font type, style, size, and color for the plot title and footer. The default title font is 12 point, bold Arial in navy. The default footer font is 9 point, regular Arial in navy.

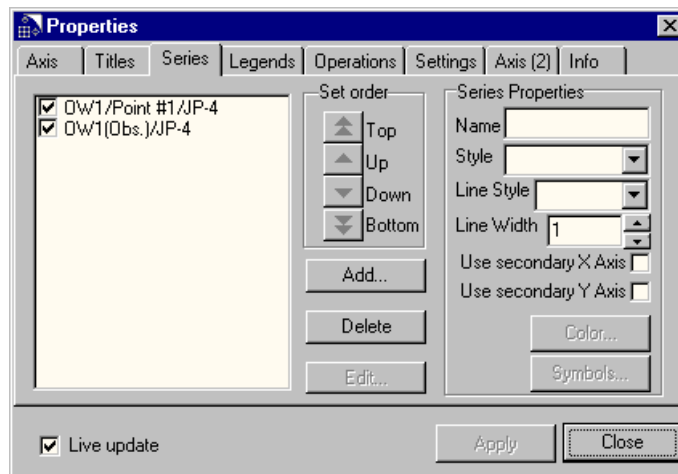
The following is a plot footer of a graph and is located below the plot area. It has been altered from the default setting of the Titles tab by:

- [1] Typing new footer text in the **Footer** text box.
- [2] Selecting the **Right Justify** option to for the footer **Alignment**.
- [3] Selecting the **Raised** option for the text **Shadow**.
- [4] Setting the **Font** as black, 10 point, bold.



## The Series Tab

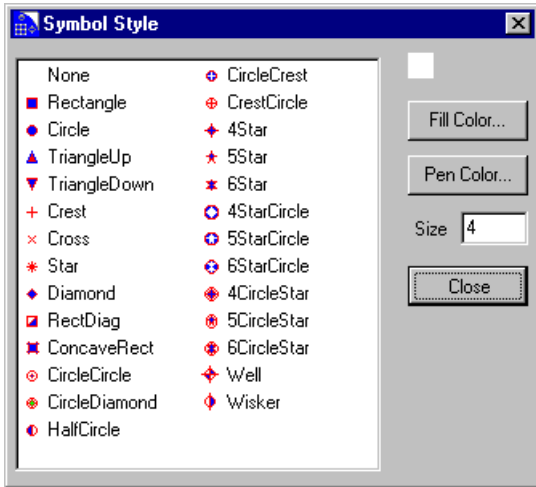
To edit the series contained within your plot area, access the Properties dialog using one of the ways previously mentioned. Select the **Series** tab.



The **Series** tab allows you to customize the appearance of each individual series in your plot. Each available series is listed, by order they appear and are updated, in the box on the left-hand side of the dialog. Notice each series has a  beside them. The  indicates the series is activated and appears on the plot. To edit the options available on this tab, the desired series must be highlighted. The following is a list and description of the options for editing each series:

**Set Order** Changes the order of the series in the list box. To change the order of the series, highlight the desired series in the list box and click on the appropriate direction button. For each move, these steps must be repeated. The **[Top]**, **[Up]**, **[Down]**, and **[Bottom]** buttons move the series to the top of the list, one series closer to the top of the list, one series closer to the bottom of the list, and to the bottom of the list, respectively.

<b>[Add...]</b>	Press to add another series to the plot. For more detail please refer to the <b>Adding/ Editing Series</b> section of this Chapter.
<b>[Delete]</b>	Press to remove the highlighted series from the list of available series for plotting. The data is not actually deleted.
<b>[Edit...]</b>	Press to modify the data in the selected series. For more detail please refer to the <b>Adding/ Editing Series</b> section of this Chapter.
<b>Name</b>	Contains the series name displayed in the list box and in the legend.
<b>Style</b>	Select the style used for displaying the series data ( <b>Points, Line, Line and point, Volume, Piecewise Constant, Area, and Bar</b> ).
<b>Line Style</b>	Select the line style used for displaying the series data ( <b>Solid, Dash, Dot, DashDot, DashDotDot, and Clear</b> ). When <b>Point, Area, or Bar</b> are selected for the <b>Style</b> , this option is not available.
<b>Line Width</b>	Select the line width used for the series, by using the neighbouring arrow keys or typing in a number. This option is only available when the <b>Line Style</b> option is available.
<b>Use Secondary X Axis <input checked="" type="checkbox"/>/ Use Secondary Y Axis <input checked="" type="checkbox"/></b>	When turned on (indicated by a ✓), the series is placed on the secondary X Axis/ Y Axis.
<b>[Color...]</b>	Sets the color of the lines, bars or fill areas for each series. When <b>Points</b> is selected for the <b>Style</b> , this option is not available.
<b>[Symbols...]</b>	Select the shape, size, and color of the symbols used when one of <b>Point, Line and point, Volume, or Piecewise Constant</b> is select as the <b>Style</b> for a series. When this button is pressed the Symbol Style dialog appears.  Click on the symbol you wish to use, to represent the series, from the list located on the left side of the dialog box. The selected symbol will become highlighted and its image will appear in the top right corner of the dialog box. This image of the selected symbol is for display purposes only and is not updated with changes to its color or size. Changes made within this dialog are updated immediately on the plot area. The features available to customize symbols are:



**[Fill Color...]** Select the color used to shade in the symbol. For example, the symbols to the left have a blue fill color.

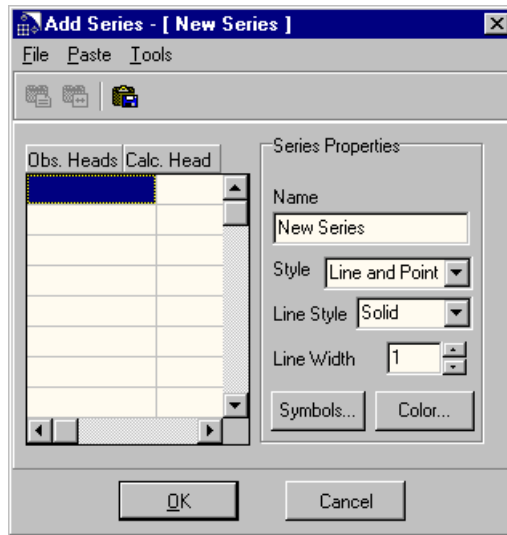
**[Pen Color...]** Select the color used to outline the symbol. For example, the symbols to the left have a pink pen color.

**Size** Type the desired point size of the symbol.

**[Close]** Press to close the Symbol Style dialog box.

### Adding/ Editing a Series

From the **Series** tab in the Properties dialog, you can add a new series by pressing the **[Add...]** button or edit an existing series by pressing the **[Edit...]** button. A new dialog will appear. Both the Add Series and the Edit Series dialog boxes have the same layout and functionality.



The Add/ Edit Series dialog provides a spreadsheet with two columns. The left column contains the X Axis data and the right column contains the Y Axis data. Each row represents one data point (X, Y) in the series. The right side of the dialog contains editing options for the series. For more detail please refer to the beginning of this section.

The top menu bar of the Add/Edit Series dialog consists of the following items: **[File]**, **[Paste]**, and **[Tools]**.

The **[File]** menu item has the following functions:

- [Open]** Allows the user to select an ASCII file, specify the files delimiters, match the file columns to the required spreadsheet columns, and import data into the schedule.
- [Exit]** Closes and saves changes made in the Add/ Edit Series dialog box.

The **[Paste]** menu item has the following functions:

- [As Is]** Pastes data from the clipboard to the spreadsheet without editing.
- [Use Field Name]** Allows the user to paste data by using the Link Fields dialog. The Link Fields dialog allows the user to match the X and Y required fields with the desired input fields. To link the **Required Field** to the appropriate **Input Field**, highlight one of the required fields, highlight the corresponding input field, and press the arrow button to add the matched pair to the link fields list box. Once all the required fields are matched with their corresponding input fields press **[OK]** to accept and paste the data in the spreadsheet.

The **[Tools]** menu item has the following spreadsheet functions:

- [Insert Row]** Inserts a blank row above the selected row.
- [Delete Row]** Deletes the selected row.
- [Erase Column]** Erases the entries belonging to the column where the highlighted cell is located.
- [Erase All]** Erases all of the entries in the spreadsheet.

In the top left corner of the Add/ Edit Series dialog box, there are several icon buttons.



- [Paste from Clipboard]** Allows the user to paste data from the clipboard to the spreadsheet without editing.



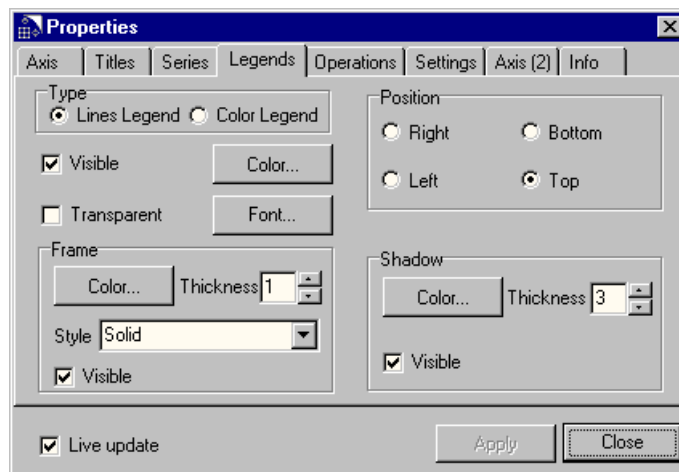
- [Paste from Clipboard by Fields]** Allows the user to paste data from the clipboard to the spreadsheet by using the Link Fields dialog box. For more detail refer to the above function **[Use Field Name]**.



- [Load from ASCII File]** Allows the user to select an ASCII file, specify the files delimiters, match the file columns to the spreadsheet columns required, and import data into the schedule.

## The Legend Tab

To edit the legend, first access the Properties dialog box. Then select the **Legend** tab.



This tab allows the user to customize the appearance of the legend. The following is a list and description of all the options available to edit the legend:

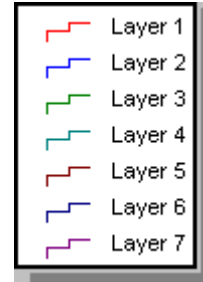
- Type** Select between a  **Lines Legend** or a  **Color Legend**.
- Visible** Shows (indicated by a ✓) or hides the legend.
- Transparent** When turned on (indicated by a ✓), makes the legend background transparent.
- [Color...]** Sets the color to the legend background. The default color is white.
- [Font...]** Sets the font type, style, size, and color for the legend text. The default legend text is 8 point, regular Arial in black.
- Frame [Color...]** Sets the color of the line around the legend box. The default color is black.
- Frame Thickness** Sets the point thickness of the legend frame.
- Style** Sets the frame line style (**Solid**, **Dash**, **Dot**, **DashDot**, **DashDotDot**, or **Transparent**).
- Frame  Visible** Shows (indicated by a ✓) or hides the legend frame.
- Position** Sets the position of the legend relative to the plot area ( **Right**,  **Left**,  **Bottom**, or  **Top**). Depending on the legend position, the legend items could be listed horizontally or listed vertically.
- Shadow [Color...]** Sets the color of the legend box shadow.

**Shadow Thickness** Specifies, using a line point thickness, the distance between the legend box center and the shadow center. For example a shadow thickness of 0, will place the shadow directly underneath the legend box, out of view.

**Shadow  Visible** Shows (indicated by a ✓) or hides the legend box shadow.

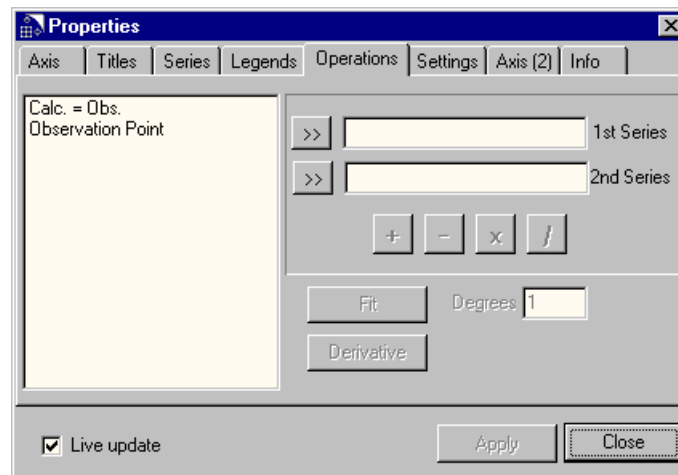
The following is a legend taken from the right side of a plot area. It has been altered from the default setting of the Legend tab by:

- [1] Setting a **Frame Thickness** of 2.
- [2] Placing the legend to the **Right** of the plot area.
- [3] Specifying a **Shadow Thickness** of 2.



### The Operations Tab

To formulate a series, first access the Properties dialog using one of the three possibilities mentioned earlier in this section, and then select the **Operations** tab.



This tab allows the user to create new series by performing mathematical operations on existing series. To perform an operation, you must first highlight a series and it to the 1st Series box. At this point you may use the **[Fit]** or **[Derivative]** buttons or, highlight another series and move it to the 2nd Series box. Now you may select an operation and a new series will be created. To edit the new series using the Property Tabs, you must first press **[Close]** and then re-open the Properties dialog. The following is a list and description of the options offered in the **Operations** tab:



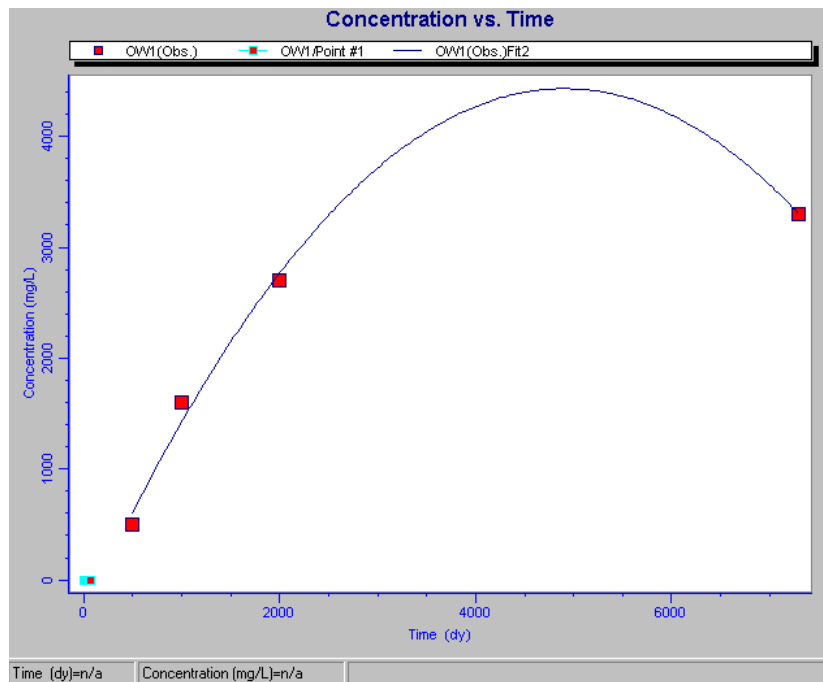
<b>1st Series</b>	Places the desired series as the first series in a mathematical operation. For example, if the <b>[/] Operation</b> button was chosen the desired series would become the numerator.
<b>2nd Series</b>	Places the desired series as the last series in a mathematical operation. For example, if the <b>[/] Operation</b> button was chosen the desired series would become the denominator.
<b>[+] Operation</b>	Adds the <b>1st Series</b> and the <b>2nd Series</b> together to form a new series.
<b>[-] Operation</b>	Subtracts the <b>2nd Series</b> from the <b>1st Series</b> to form a new series.
<b>[x] Operation</b>	Multiplies the <b>1st Series</b> and the <b>2nd Series</b> together to form a new series.
<b>[/] Operation</b>	Divides the <b>2nd Series</b> into the <b>1st Series</b> to form a new series.
<b>[Fit]</b>	Adds a new series representing a best fit curve for the set of data points in the highlighted series. The curve is a polynomial to the order of the number of degrees specified.
<b>Degrees</b>	Dictates the order of the polynomial curve.
<b>[Derivative]</b>	Adds a new series representing the first derivative for the set of data points in the highlighted series.

**To add a new series using the Operations tab:**

- [1] Select a series, from the left-hand side list of the **Operations** tab, by clicking the name of the desired series. Press the **[>>]** button associated with the **1st Series**. The highlighted series' name will appear in the **1st Series** box.
- [2] Select another series, from the left-hand side list of the **Operations** tab. Press the **[>>]** button associated with the **2nd Series**. The highlighted series' name will appear in the **2nd Series** box.
- [3] Press the **[-] Operation** button to subtract the **2nd series** from the **1st series**. This newly created series (**1st Series – 2nd Series**) will appear immediately on the plot area if you have selected  **Live Update**. If you have not, press **[Apply]** to have the new series appear on the plot area.

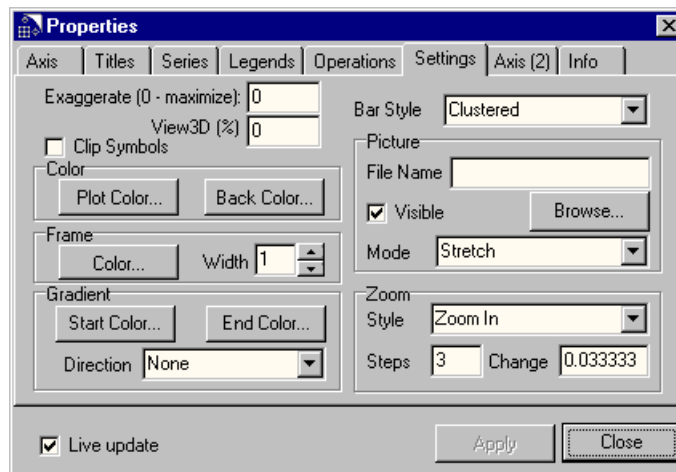
The following graph has a newly created series. This new series was created using the **Operations** tab by:

- [1] Selecting the **OW1(Obs.)** series.
- [2] Specifying a **Degrees** value of 2.
- [3] Pressing the **[Fit]** button to create the new series **OW1(Obs.)Fit2**.



### The Settings Tab

To edit miscellaneous items on the plot area, access the Properties dialog using one of the three previously mentioned options. Select the **Settings** tab.



The **Settings** tab allows the user to customize the general appearance of the plot area. The following is a list and description of all the miscellaneous options available with the IChart plotting component:

**Exaggerate (0-maximize)**

Sets the vertical exaggeration factor for the plot. The value entered increases or decreases the unit length of the Y axis with respect to the X axis. For example, setting the exaggeration value to,

- 0, will maximize the plot area and there will be no unit length relation between the Y and X axis.
- 1, will set the unit length of the Y axis equal to the unit length of the X axis.
- 2, will set the unit length of the Y axis double the unit length of the X axis.

**View3D(%)**

Specifies the percentage the plot area is rotated around the Y axis, giving a 3D view. 100% rotates the plot area 90 degrees from the 0% (2D) view.

**Clip Symbols**

When turned on (indicated by a ✓), the minimum and maximum values for the axes will be set equal to the minimum and maximum data point values. This will clip the symbols of these data points. When turned off, the axes are offset slightly to include the minimum and maximum data point values. This options is available only when  **Automin** and  **Automax** are turned on, in the **Axis** tab.

**[Plot Color...]**

Sets the color for the plot area background. The default color is white.

**[Back Color...]**

Sets the color for the area surrounding the plot area. This area includes the plot title and footer area. The default color is grey.

**Frame [Color...]**

Sets the color of the line around the plot area. The default color is blue.

**Width**

Specify the point thickness of the frame around the plot area.

**Gradient [Start Color...]**

Sets the color to start the background gradient. At the beginning of the gradient this color will be strong and at the end of the gradient this color will be weak. The gradient will only cover the area surrounding the plot area. The default [Start Color...] is white.

**Gradient [End Color...]**

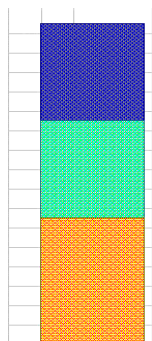
Sets the color to end the background gradient. At the beginning of the gradient this color will be weak and at the end of the gradient this color will be strong. The default [End Color...] is grey.

**Direction** Specifies the direction the gradient will follow (**None**, **LeftRight**, or **TopBottom**). If the direction **None** is selected, the background color will be dictated by **[Back Color...]**.

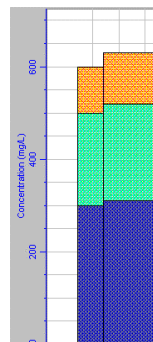
**Bar Style** Specifies the bar styles used to plot bar charts (Clustered, Z-Ordered, Stacked, or 100% Stacked). The following details the bar styles:



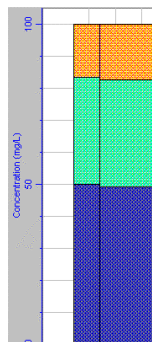
Clustered



Z-Ordered



Stacked



100% Stacked

- the **Clustered** bar style, places each bar separately starting at the X axis and ending at its value. The first bar in the group is on the bottom of the series list from the **Series** tab.
- the **Z-Ordered** bar style, places each bar on top of one another, starting at the X axis and ending at its value. The first series is the top of the series order.
- the **Stacked** bar style, places each bar on top of one another, starting with the series on the bottom of the series list. The value of each series is determined by the difference between its own starting and ending point.
- the **100% Stacked** bar style, places each bar on top of one another, starting with the series on bottom of the series list. The series starting and ending point difference is its percentage of the series' total values at an X point.

Each of these bar styles affects the plotted series when more than one series is specified as a bar.

**Picture File Name** Allows the user to specify a bitmap image to be depicted on the background plot area.

**Picture  Visible** Shows (indicated by a ✓) or hides the bitmap image on the background plot area.

**Picture [Browse...]** Allows the user to search and select the bitmap image file to be depicted on the background plot area.

**Picture Mode** Sets the manner in which the bitmap image is displayed in the plot area. The **Stretch** mode fits the image to the plot area. The **Fix**

mode fits the plot area to the image's dimensions. The **Tile** mode fills the entire plot area with small squares of the image.

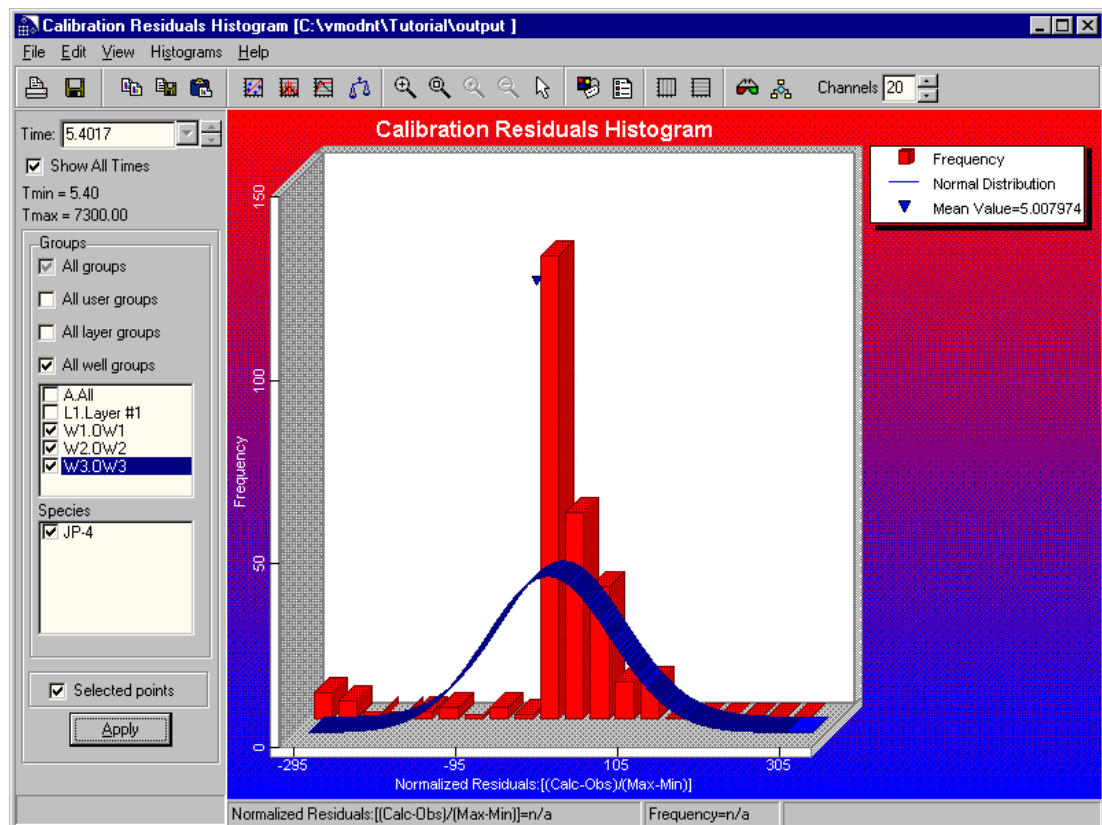
**Zoom Style** Sets the animated zoom direction to either Zoom In or Zoom Out.

**Zoom Steps** Sets the number of steps used in the animated zoom sequence.

**Zoom Change** Sets the zoom factor for each step in the animated zoom sequence.

The following is a chart that has been altered from the default setting of the **Settings** tab by:

- [1] Specifying a **View3D (%)** of 10.
- [2] Setting a **Frame Width** of 4.
- [3] Setting the **Gradient Direction** to **LeftRight**.
- [4] Altering the **[Start Colour...]** and the **[End Colour...]** of the gradient to red and blue (note the title, axis labels, and axis names were also changed to white).



## The Axis (2) Tab

Please refer to the **Axis Tab**.

## The Info Tab

<b>Style</b>	In the style combo box you can pick from one of three style types: <b>Round Rectangle</b> , <b>Rectangle</b> , and <b>No Bubble</b> .
<b>Position</b>	In the position combo box you can pick from one of 6 locations for the bubble tag. Your options are <b>RightTop</b> , <b>TopCentre</b> , <b>LeftTop</b> , <b>RightBottom</b> , <b>BottomCentre</b> , and <b>LeftBottom</b>
<b>[Color...]</b>	Sets the color for the info bubbles.
<b>[Font...]</b>	Sets the font and its style, size and color for the info bubble text. The default legend text is 8 point, regular Arial in black.
<b>Formats</b>	Format specifiers have the following form: "% " ["-"] [width] [". " prec]

A format specifier begins with a % character. After the % come the following, in this order:

- An optional left justification indicator, ["-"]
- An optional width specifier, [width]
- An optional precision specifier, [". " prec]

The following summarizes the possible values for type:

<b>e = Scientific</b>	The value is converted to a string of the form "-d.ddd...E+ddd". The resulting string starts with a minus sign if the number is negative. One digit always precedes the decimal point. The total number of digits in the resulting string (including the one before the decimal point) is given by the precision specifier in the format string--a default precision of 15 is assumed if no precision specifier is present. The "E" exponent character in the resulting string is always followed by a plus or minus sign and at least three digits.
<b>f = Fixed</b>	The value is converted to a string of the form "-ddd.ddd...". The resulting string starts with a minus sign if the number is negative. The number of digits after the decimal point is given by the precision specifier in the format string--a default of 2 decimal digits is assumed if no precision specifier is present.
<b>g = General</b>	The value is converted to the shortest possible decimal string using fixed or scientific format. The number of significant digits in the resulting string is given by the precision specifier in the format string--

a default precision of 15 is assumed if no precision specifier is present. Trailing zeros are removed from the resulting string, and a decimal point appears only if necessary. The resulting string uses fixed point format if the number of digits to the left of the decimal point in the value is less than or equal to the specified precision, and if the value is greater than or equal to 0.00001. Otherwise the resulting string uses scientific format.

**n = Number** The value is converted to a string of the form "-d,ddd,ddd.ddd...". The "n" format corresponds to the "f" format, except that the resulting string contains thousand separators.

Conversion characters may be specified in uppercase as well as in lowercase--both produce the same results. For all floating-point formats, the actual characters used as decimal and thousand separators are obtained from the DecimalSeparator and ThousandSeparator global variables. Index, width, and precision specifiers can be specified directly using decimal digit string (for example "%10d"), or indirectly using an asterisk character (for example "%\*.\*f"). When using an asterisk, the next argument in the argument list (which must be an integer value) becomes the value that is actually used. For example,

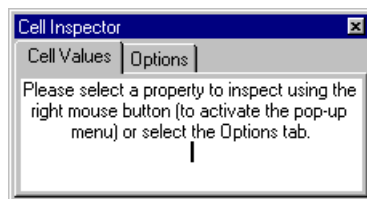
```
Format("%*.*f", [8, 2, 123.456])
```

is the same as

```
Format("%8.2f", [123.456]).
```

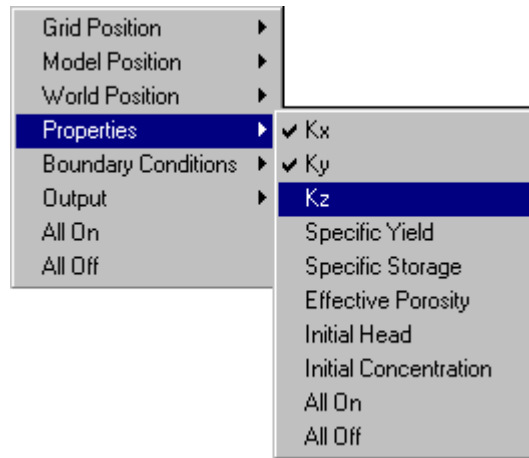
A width specifier sets the minimum field width for a conversion. If the resulting string is shorter than the minimum field width, it is padded with blanks to increase the field width. The default is to right-justify the result by adding blanks in front of the value, but if the format specifier contains a left-justification indicator (a "-" character preceding the width specifier), the result is left-justified by adding blanks after the value.

## Cell Inspector



The Cell Inspector can be called up in the Input and Output Modules. It can be modified to display the desired information as you move your cursor over the model domain. To access the Cell Inspector, click on **[Tools]** from the top menu bar in either the Input Module or the Output Module, and then select **[Cell Inspector]** from its drop-down menu. The following message will appear on the dialog when the **Cell Values** tab is selected and before the desired information is chosen.

The Cell Inspector allows you to turn on/off the information that you want displayed. There are two ways to turn on/off displayed values. If you click the right mouse button in the **Cell Values** tab, a pop-up menu appears allowing the user to select and deselect the parameters that are to be displayed. A ✓ will appear next to the information that has been turned on.



To turn on all the sub-parameters, select the All On option at the bottom of the submenu. If all the sub-parameters are turned on then a ✓ appears next to the main parameter name. To turn on all the main parameters (this turns on all the sub-parameters), select the All On option at the bottom of the pop-up menu.



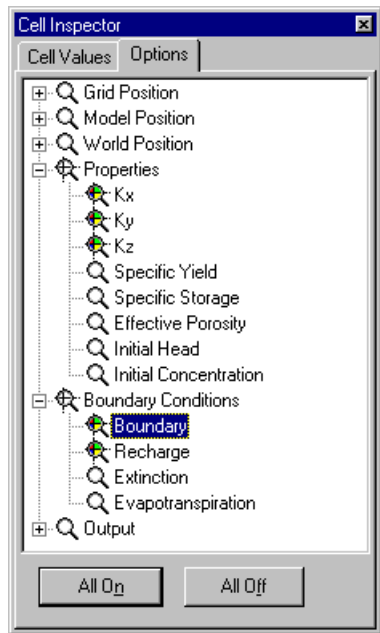
The second option is to select the **Options** tab and click on the cell parameters to be displayed. If you double click on the main parameter heading, all of the sub-parameters will be selected or deselected. Also you can click on the **[All On]** or **[All Off]** buttons at the bottom of the dialog to select or deselect all of the parameters. To select the sub-parameters individually, click on the plus sign beside the main parameter name to expand the tree. If a sub-parameter is selected the magnifying glass will be multicolored and its main parameter's magnifying glass will have a cross through it. When all sub-parameters are turned on, their main parameter's magnifying glass will become multicolored.



The method of selecting and deselecting the parameters is consistent in the Input and Output Modules. The parameters that are available for the Input Module will, however, be slightly different than the parameters available for the Output Module.

Once the parameters have been turned on, select the **Cell Values** tab and move the cursor over the model domain. The values in the dialog will be updated to reflect the cell contents.





The screenshot shows the 'Cell Inspector' dialog box with the 'Options' tab selected. A table displays the values for various properties. The 'Boundary' property is set to 'Constant Head'.

Property	Value	Units
Row	1	
Column	58	
Layer	1	
Model X	1756.37	
Model Y	1966.01	
Model Z	18.325	
World X	1756.37	
World Y	1966.01	
World Z	18.325	
Kx	0.0002	m
Ky	0.0002	m
Kz	0.0002	m
Specific Yield	0.2	m
Specific Storage	0.0001	m
Effective Porosity	0.15	m
Initial Head		
Initial Concentration	0	m
Evapotranspiration		
Boundary	Constant Head	
Extinction		
Recharge	100	mm/yr
Head		
Concentration		

To exit Cell Inspector, press the close button, in the dialog's top right-hand corner.



## A

About the Interface 3  
AquaChem ix  
AquiferTest viii, ix

## B

Base Maps 10  
BMP 13  
DXF 13

## C

Cell Inspector 53  
Common Tools 33  
Co-ordinates  
Dual Co-ordinate System 12  
Transforming Model to World 12  
Transforming World To Model 12  
Co-ordinates Area 8  
Copying a Variant 31  
Creating a New Model 9  
Creating a new Variant 32  
Cube Navigator 8  
Cursor (arrow) keys 9

## E

Enter Key 9  
Esc Key 9  
Export  
DXF File 22  
EMF File 22  
Windows Metafile 22

## F

Feedback x  
Finite Difference Methods 23  
FLOWPATH ix  
Function Buttons 8  
F1 - Help 8  
F10 - Main Menu 8  
F2 8  
F3 - Save 8  
F4 - Map 8  
F5 - Zoom In 8  
F6 - Zoom Out 8  
F7 - Pan 8  
F8 - Vert Exag 8  
F9 - Overlay 8, 33

## G

General Features 5  
General Tab 27  
Georeference Point 13  
Delete 14  
Set 14  
Getting Around Visual MODFLOW 5  
Graphs  
Properties 35

## H

Hardware Requirements 2

## I

IChart 35  
Adding/Editing series 43  
Axis 37  
Axis(2) 52  
Info Tab 52  
Legend 45  
New series 47  
Operations 46  
Series 41  
Settings 48  
Titles 39  
Immobile Species 27  
Input Module 3, 5  
Installation  
Visual MODFLOW 2  
Introduction 1

## M

Mass Transport Theory 23  
Mobile Species 27  
Model Params. Tab 28

## N

Numeric Engines  
Setup 5, 22  
Numerical Methods  
Explicit 23  
Implicit 23

## O

Output Module 3, 5  
Overlay Function Features 33

## P

Printing 16  
Old Visual MODFLOW Printing 20  
Page Layout 17  
Printer Setup 16  
Title Blocks 18  
To DXF File 22  
To EMF File 22  
To Windows Metafile 22  
project.var001 file 29

## R

Reference Guide 3  
Re-intialiazing the Variant 32  
Run Module 3, 5

## S

Screen Layout 7  
Side Option Bar 7  
Goto 7

Next 8  
Previous 7  
View Column Button 7  
View Layer Button 7  
View Row Button 7  
Species Params. Tab 29  
Species Tab 27  
Starting Visual MODFLOW 2  
Status Line 8

## T

Tab Key 9  
Technical Support vii  
Top Menu Bar 7  
Transport Engine Options 25  
General Tab 27  
Model Params. Tab 28  
Species Params. Tab 29  
Species Tab 27

## U

Units  
Defining 10

## V

Variant repository 30  
Variants 25, 29  
Changing Flow Model 30  
Copying 31  
Creating 32  
Re-initializing 30  
Translating Model 30

## View

column 7  
layer 7  
row 7  
Visual Groundwater ix  
Visual HELP x

## W

Waterloo Hydrogeologic Inc.  
How to Contact WHI vii  
Other Products by WHI ix  
Training and Consulting viii