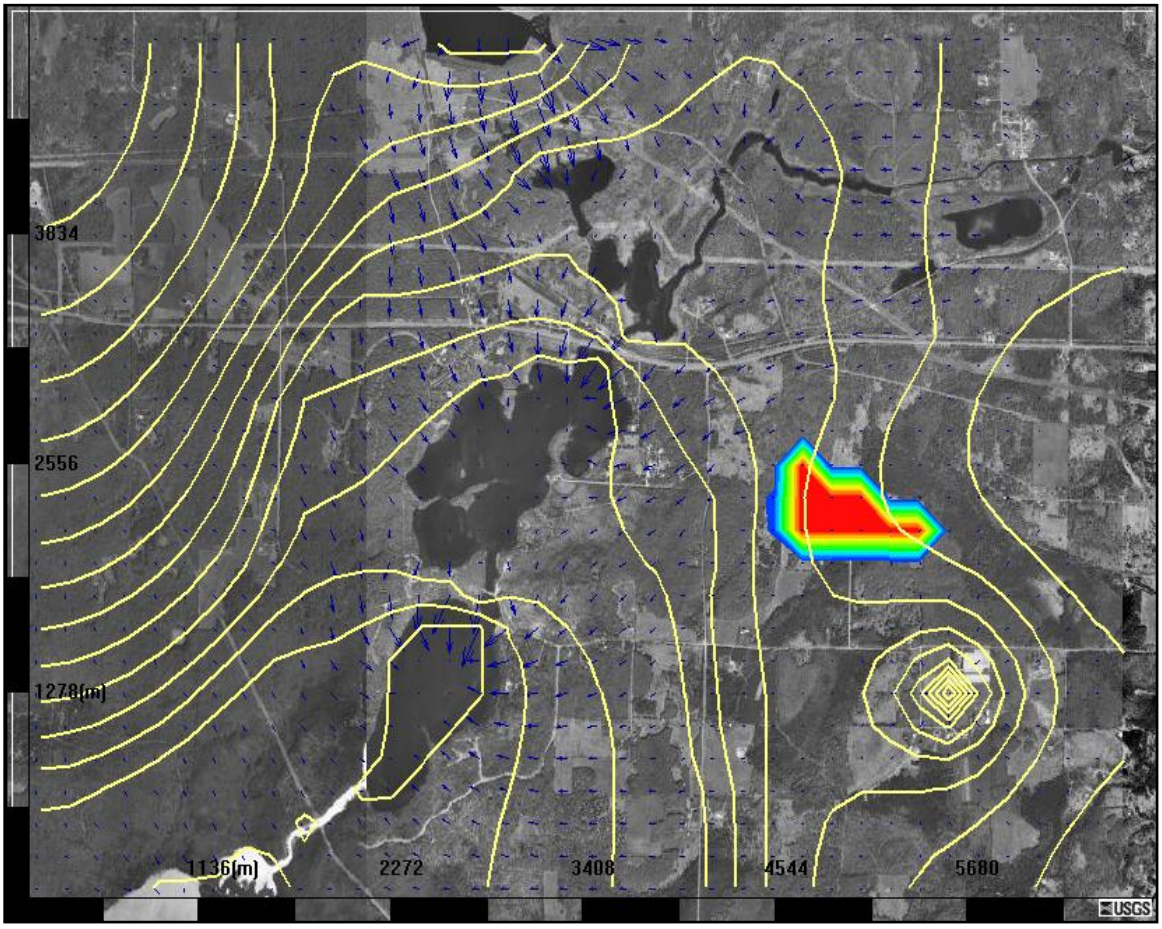
DR. SHUGUANG LI AND ASSOCIATES

INTERACTIVE GROUNDWATER MODELING (IGW)



IGW VERSION 5.0P

USER'S MANUAL

Dr. Shuguang Li and Associates at Michigan State University

IGW User's Manual for *Version 5.0P*

Authors / Editors:
Hassan Abbas
Dr. Shuguang Li
Justin Scheidt

Special Thanks to:
Dr. Hausheng Liao
Dr. Qun Liu

DOCUMENT VERSION: **2006-1**

http://www.egr.msu.edu/~lishug

TABLE OF CONTENTS

TABLE OF FIGURES

Chapter 1 INTRODUCTION

This brief chapter provides an introduction to the capabilities and requirements of Interactive Groundwater Modeling (IGW Version 5.0P), in addition to pointers for using this document and contact information.

1.1. IGW Version 5.0P Software Synopsis

IGW Version 5.0P is a real-time, interactive and visual software system for unified deterministic and stochastic groundwater modeling. Letter "P" in the version number stands for "Parallel Computing". The software is capable of 'parallel computing', making optimal use of computing power of multiple processors in the networked systems. This capability reduces computing time by many folds for complex stochastic simulation models.

Taking advantage of recent developments in groundwater research, numerical simulation techniques, and visual programming, Dr. Li and his group are developing a software system for unified deterministic and stochastic groundwater modeling. The system is designed to simulate unsteady flow and reactive transport in general groundwater formations, subject to both systematic and 'randomly' varying stresses, along with geological and chemical heterogeneity. The current version of the software, characterized by its full interactivity and built-in real-time and animated visualization, has the following distinct capabilities:

- High level and grid-independent conceptual modeling. This includes interactive visual specification and editing of model domain, aquifer properties and stresses (over any arbitrarily-shaped area, at any point in time during the process of model construction), simulation, and analysis.
- Automatic grid generation and conversion of conceptual representation to numerical model(s).
- Interactive simulation and 'real-time' visualization and animation of flow, in response to both deterministic and stochastic stresses.
- Interactive, visual, and 'real-time' particle tracking, random walk, and reactive plume modeling in both systematically and randomly fluctuating flow.
- Interactive and visual conditional simulation of hydrogeologic and geochemical spatial fields; 'real-time', visual, and parallel conditional flow and transport simulations.
- Interactive scattered data interpolation, regression, and variogram modeling.
- Interactive water and contaminant mass balance analysis with visual and real-time flux update.
- Interactive, visual, and 'real-time' monitoring of head hydrographs and concentration breakthroughs.
- Real-time modeling and visualization of aquifer transition: confined or unconfined to partially de-saturated, or completely dry and rewetting.
- Interactive, visual, and 'real-time' sub-scale modeling using a telescopic approach.
- Interactive, visual, and 'real-time' stochastic Monte Carlo simulations.

The user-friendly software system dramatically simplifies the process of groundwater modeling and provides virtually 'instant' analysis with visual solution representation. The software provides an effective tool for: 1) Groundwater professionals, site planners, managers, and regulators to conduct site investigation and experiment in 'real-time' with sampling strategies, management options, and remedial schemes, 2) Researchers to study field-scale physical and chemical groundwater processes in heterogeneous soils, and 3) Academics to teach groundwater flow transport, site investigation, and remediation, using vivid interactive and real-time simulations.

1.2. User's Manual Introduction

This manual is intended to give in-depth information concerning **IGW Version 5.0P**, including its interface and implementation. It is assumed that the reader has a basic knowledge of the Microsoft® Windows® Operating System.

For more detailed information concerning technical content of the model (i.e., mathematics and theory), please consult the *IGW Version 4.7 Reference Manual*.

The *IGW Version 5.0P Tutorials* offer step-by-step instructions for implementing some of the procedures/methods presented in this document.

1.2.1. User's Manual Layout

This manual is organized according to the logical order the user would employ the features of the software interface, if building a model from scratch. For example, **Chapter 3** describes the main interface and the functions of the buttons. These descriptions then direct the user to later chapters of the document that deal with each representative topic.

On the other hand, the document is not exclusively sequential. Each chapter is written to stand alone and any information necessary to understand the chapter is cross-referenced. Therefore, the easiest way to learn a particular aspect of the software is to:

1) Use the **Table of Contents** to find the appropriate chapter or sections,
2) Read the appropriate selection, and
3) Reference the indicated chapters/sections/documents to 'fill in the blanks'.

i Throughout the manual, there are numerous **INFO (i)** texts that give important information to enhance the **IGW** experience.

R Throughout the manual, there are **RECOMMENDATIONS** that give an alternative way to run a particular feature or perform a procedure in **IGW,** in order to make its functionality easier and more effective.

! Boxed **WARNING** texts are used to warn the users about groundwater modeling aspects and operational use of **IGW** that may require more careful attention and evaluation during the operation.

Light gray text indicates features of the software that have not yet been implemented (but may be incorporated in the later versions of its interface).

Throughout the manual, some of the screen view styles may be different from each other, depending on the Windows version in which they were created.

It is important to note that the **IGW 3D** versions (recently, **Version 5.0P**) and the associated documents are undergoing constant revision. Check the website (see **Section 1.3**) often for updates.

1.2.2. User's Manual Acronyms/Abbreviations

There are a large number of acronyms used throughout this text. The acronyms, their meanings, and the section in which they are first discussed are presented in **Table 1-1**.

Table 1-1 User's Manual Acronyms

ACRONYM / ABBREV.	DEFINITION	REFERENCE
AE	Attribute Explore (AE) window	**Section**
AEA	Attribute Entry Area	**Section 7.6**
ASCII	American Standard Code for Information Interchange	**Section 20.3**
BMP	Bitmap	**Section 3.3.1**
CAT	Cursor Activated Table	**Figure 3-1**
dpi	dots per inch	**Section 1.2.3**
DT	Time Step	**Section 3.5**
DXF	Drawing Exchange Format	**Section 5.2**
FAQ	Frequently Asked Questions	**Section 2.3**
GIA	General Information Area	**Section 7.6.2**
GIF	Graphics Interchange Format	**Section 5.2**
IGW	Interactive Groundwater Modeling	**Chapter 1**
IW	Interactive Wizard	**Section 2.3**
JPEG	Joint Photographic Experts Group	**Section 3.3.1**
LHP	Left-hand Pane	**Section 4.1.1**
LMA	Left Message Area	**Section 3.13**
MB	Megabyte	**Section 1.2.3**
MGD	Million Gallons per Day	**Section 9.4.2**
MHz	Megahertz	**Section 1.2.3**
MMOC	Modified Method of Characteristics	**Section 13.3.1**
N/A	Not Applicable	**Section 3.12**
PP	Plot Pane	**Appendix E**
RAM	Random Access Memory	**Section 1.2.3**
RBSP	River Bottom Sediment Properties	**Section 7.6.1.4.2**
RHP	Right-hand Pane	**Section 4.1.2**
RMA	Right Message Area	**Section 3.13**
SATDI	Step Adjustment and Time Display Interface	**Figure 3-1**
SHP	Shape	**Section 5.2**
SOR	Successive Over Relaxation	**Section 13.1.1**
SP	Statistics Pane	**Appendix E**
TPS	Time Process Selector	**Section 4.1.3**
VCI	Vertex Coordinate Interface	**Section 3.13**
VOA	Visualization / Option Area	**Section 7.6**
WAAD	Working Area Attribute Display	**Figure 3-1**
XXXX	Implies a number between 000 and 999	**Section 3.13**
ZCA	Zone Color/Pattern Area	**Section 7.6.2**
ZTA	Zone Type Area	**Section 7.6.2**

1.2.3. System Requirements

IGW Version 5.0P is designed for PCs running Microsoft® Windows®. The software runs on the following versions of Windows®: 95, 98, NT, 2000, ME, XP, and Vista.

Table 1-2 lists the minimum and recommended system requirements for reliably using the **IGW Version 5.0P** software.

Users of Windows® 95, 98, NT and 2000 will need access to zip decompression software. See **Section 2.1** for more information.

Table 1-2 Minimum and Recommended System Requirements

ATTRIBUTE	MINIMUM	RECOMMENDED
Processor Speed	300 MHz	800 MHz
Memory (RAM)	64 MB	256 MB
Hard Drive Space	100 MB	200 MB
Display Resolution	640 x 480 dpi	1024 x 768 dpi
Color Setting	8-bit	32-bit

1.3. Additional Information

Additional information concerning **IGW** can be obtained from the **IGW website**:

http://www.egr.msu.edu/igw/

The site contains numerous topics, including:

- Exploring the capabilities and algorithms associated with the software,
- Viewing software demonstrations and associated presentations,
- Downloading verification papers,
- Obtaining software documentation,
- Downloading the software,
- Providing feedback,
- Accessing the IGW Forum,
- Contacting Dr. Li and his associates, and
- Acknowledgements and team members.

Chapter 2 GETTING STARTED

This brief chapter outlines the recommended steps to obtain and begin using IGW Version 5.0P.

2.1. Obtaining the Software

As mentioned in **Section 1.3**, the software can be obtained from the IGW homepage. The specific link for IGW Version 5.0P is:

http://www.egr.msu.edu/igw/ [1]

At this point, the three-dimensional Version 5.0P is not available on the web.

Notice that the files have the 'zip' extension. This is because the files are in 'zipped' format to reduce the downloading size and protect the file integrity during transmission. Windows® ME and above have built-in zip capability and will have no problem extracting the program files embedded. However, users of other versions will need decompression software that supports zip files. It is recommended to use the version 8.1 or higher of the WinZip® archive utility. A fully functional evaluation version is available on the Internet at

http://www.winzip.com/downauto.cgi?winzip81.exe

For more information concerning downloading, installing, or using WinZip® please consult the WinZip® homepage at

http://www.winzip.com

If any trouble is encountered attempting to obtain the software, please use the links on the IGW website to contact Dr. Li or his associates. All feedback is appreciated.

2.2. Installing the Software[2]

Once the setup files have been extracted from the zip file, simply execute the setup application located in the folder where the files were extracted.

!	The icon or font may not exactly match the picture at right, depending on the software version and target computer appearance settings.	

Alternatively, some decompression software packages allow the user to install the software as a continuation of the unzipping process. In this case, it is not necessary to execute the setup file, as the decompression software does it automatically.

The steps involved in the installation process (after initiating the setup application) are:

1. Authorizing installation continuation.
2. Closing other open programs and continuing.
3. Reading the License Agreement and accepting it.
4. Reading an important notice and continuing.
5. Selecting the install folder (and verifying overwrite if folder already exists).

[1] There are also movies associated with the tutorials document that may be loaded as a package with the software by using the following link: http://www.egr.msu.edu/~lishug/research/igw/download/ igw3.x/igw3.x.movie.zip.
[2] Versions 3.1 and earlier should be uninstalled before installing of IGW Version 5.0P .

6. Selecting the Start Menu group.
7. Authorizing continuation (last chance to review options) and installation.
8. Reading important information and continuing.
9. Reading notification of successful install and finishing.

Once the installation is complete, the software is ready to use. There is no need to restart the computer[3].

R It is recommended to use only one 3D version of IGW (4.0 or higher) on a single machine at a given time. Before installing a different version of the software please be sure to uninstall any previous version(s).

2.3. Starting the Software[4]

The easiest way to start the program is to use the Windows® Start Menu. Alternatively, the program can be started by executing the IGW Version 5.0P application from the C:\Program Files\Interactive Groundwater 5.0P folder – the default – or the one chosen during installation. The software shortcuts are installed in the Programs→Interactive Groundwater folder in the Start Menu (again, by default – or in the folder chosen during installation). The folder will contain a shortcut 'IGW 5.0P' that is used to start the program and a shortcut 'Uninstall IGW 5.0P' that is used to uninstall the program.

Upon starting the software, a splash screen with the software credits will appear (see **Figure 2-1**). The user may click the 'Continue' button or just wait a few seconds. The splash screen disappears and the full screen IGW Version 5.0P window appears, with a 'Tip of the Day' window (see **Figure 2-2**). From the 'Tip of the Day' window, one can:

- Scroll through the tip list by selecting Next Tip;
- Access the Frequently Asked Questions (FAQ) list by clicking Frequently Asked Questions;
- Open the step-by-step tutorial by clicking Step by Step Tutorial;
- Open the on-line help by clicking Online Help;
- Close the window Close; and,
- Decide whether or not the 'Tip of the Day' window should be displayed next time the software starts Show tips on startup.

Completing the step by step tutorial lessons is a very good way for beginners to become familiar with the IGW Version 5.0P interface and basic functionality. The step-by-step tutorial is a help file version of the ***IGW Version 5.0P Tutorials*** document. It contains a number of movies associated with the tutorial lessons that show how the steps should be completed and what should occur after each step is taken (available only with the movie-software download bundle – see footnote 1 on **page 5**). Closing the 'Tip of the Day' window provides a complete view of the main window (it can be reopened at any time through the **Help** menu – see **Section 3.3.7**). This is the starting point for all features, and provides a focus point for discussion of the software implementation – continued in **Chapter 3**.

i Step by step tutorial and FAQ is not fully functional in 'Tip of the Day' in *IGW Version 5.0P*

[3] There are some Versions of Windows® 95 and 98 that may require the system to be restarted.
[4] Refer to **Section 2.1** of the ***IGW Version 5.0P Tutorials*** document for a step-by-step example of starting the software.

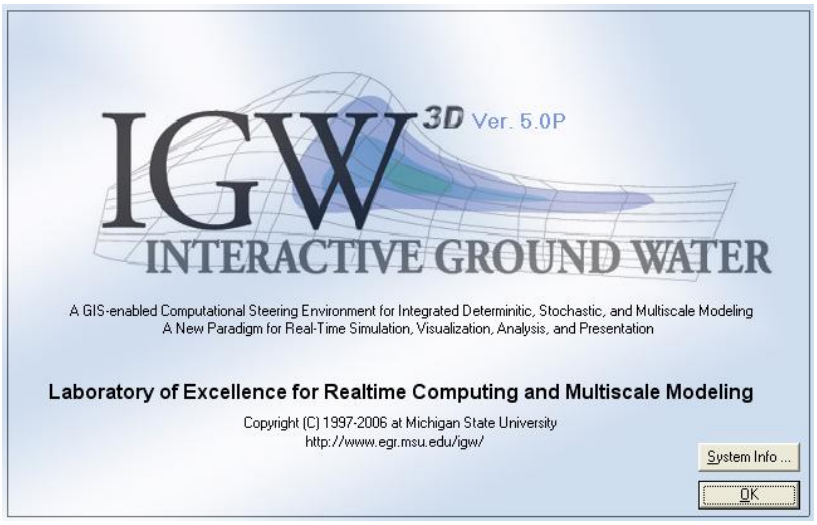


Figure 2-1 IGW Version 5.0P Splash Screen

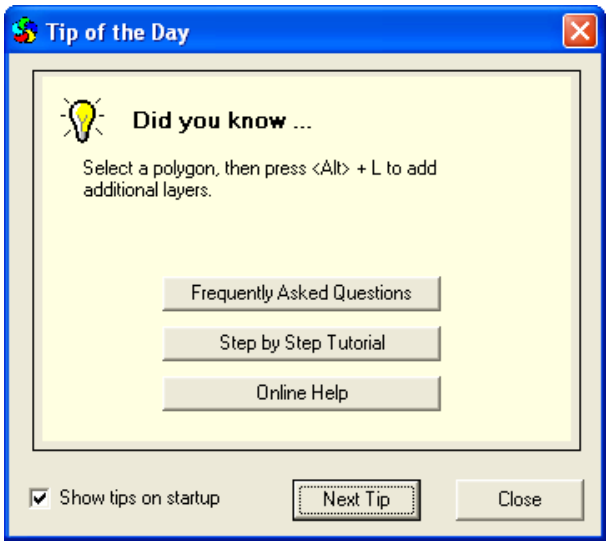


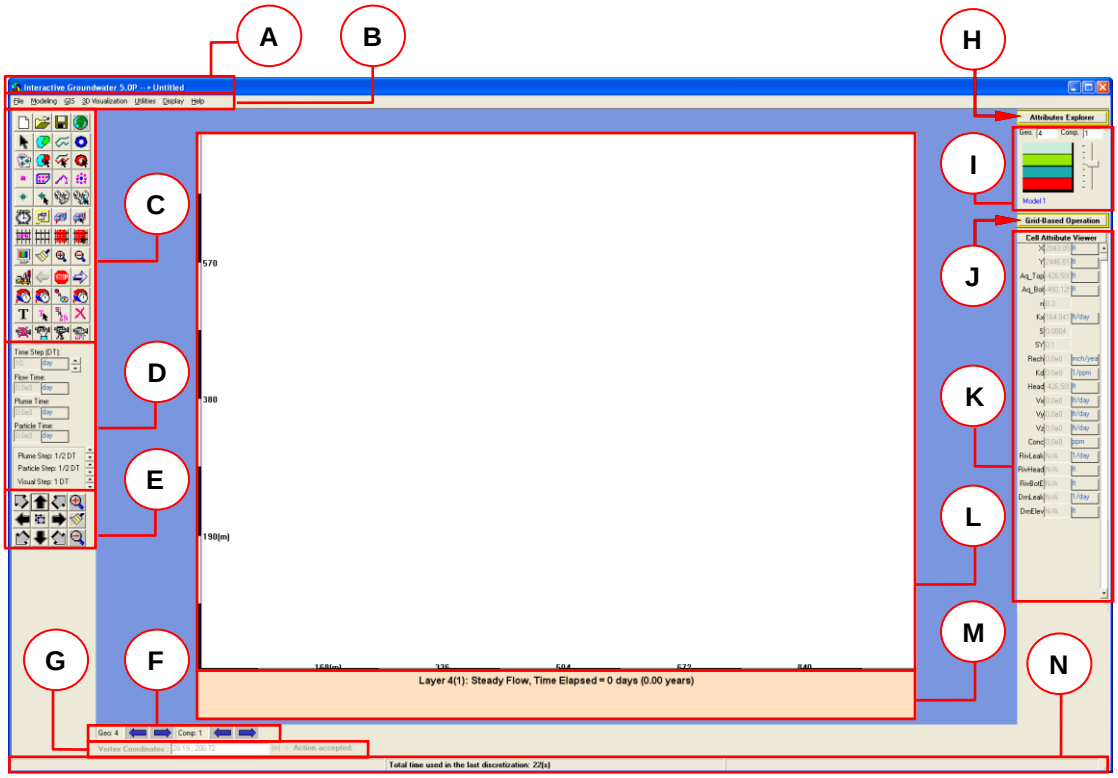
Figure 2-2 Tip of the Day

Chapter 3 MAIN WINDOW INTERFACE[5]

One of the primary features of IGW Version 5.0P is that it is very intuitive, helping the user to grasp the functionality of the software and the interface, due to its logical layout. Therefore the ideal place to begin a discussion of the software implementation is with the interface.

3.1. Main Window Layout

IGW Version 5.0P main window shown in **Figure 3-1** is divided into 14 parts (A through N) to facilitate its functionality and explanation of items/buttons/zones etc. The subsequent sections explain each of these 13 parts of the main window.



A- Title Bar
B- Menu Bar
C- Button Palette
D- Step Adjustment and Time Display Interface (SATDI)
E- Working Area Display Tools
F- Layer Navigator
G- Vertex Coordinates Interface
H- Attributes Explorer Button
I- Layer Selector
J- Grid-Based Operations Button
K- Cursor Activated Table (CAT)
L- Working Area
M- Working Area Attributes Display (WAAD)
N- Status Bar/Message Bar

Figure 3-1 IGW Version 5.0P Main Window

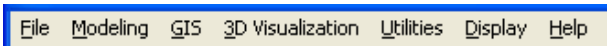
[5] Refer to **Section 2.2** of the ***IGW Version 5.0P Tutorials*** document for a short interactive walkthrough of the Main Window interface.

3.2. Title Bar

The 'title bar' shows the name of the software, the version (in this instance, 5.0P), and the name and path of the open file (in this case the software default 'Untitled'). It also contains the Windows® buttons that allow the user to 'minimize' the window[6], toggle the window between full-screen and small window, and close the program (the same as selecting 'Exit' from the **File** menu – see **Section 3.3.1**). The remaining attributes of the main window are discussed in the following sections.

3.3. Menu Bar

The 'menu bar' is located just below the title bar. Seven menus are accessible from the menu bar: **File**, **Modeling, GIS, 3D Visualization, Utility**, **Display**, and **Help**.



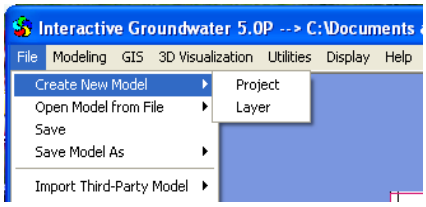
Each of these menu items is explained below.

3.3.1. File Menu

The **File** menu contains the following operations:

Create New Model Selecting this opens a new file. This is the same as clicking the 'Create a New Model' button (see **Section 3.4**). New to Version 5.0P is the option to select opening either an entire project or a single layer. More discussion on layers will be given in subsequent chapters.

R	The user should save work prior to creating a new model as the software will not prompt the user to save it.

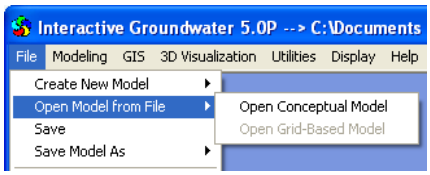


Open Model From File Selecting this opens an existing IGW Version 5.0P file. A window titled 'Open' appears, therefore allowing the user to browse to the location of the desired file. The user only has the option to open a conceptual model, as the grid-based model option is not yet offered.

!	The user should remember that 2D version files are not compatible with 3D versions, or vice versa.

[6] This function minimizes only the Main Window. Other IGW Version 5.0P interfaces are not affected.

i IGW can run multiple models through different sessions, simultaneously. Previously-saved files can be opened in separate IGW sessions at the same time.



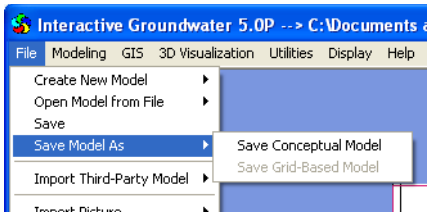
i In IGW, you cannot double-click on an existing model file to run it. Instead, the user should go into **File** menu.

R If the Working Area for an existing modeling session contains a large amount of data, then it may be difficult to create a new model using the same session due to memory issues. Instead, the user is recommended to open a fresh IGW session and then create the new model in that session.

Save Saves the latest changes in the model currently open.

i The first time you click on Save in a new model will automatically open the 'Save As' window. Once the model is saved, subsequent use of Save will save the model with the latest changes

Save Model As Selecting this saves all present work. A window titled 'Save As' appears, allowing the user to save the file in the desired location. A valid name must be supplied, as no default name is given. The user can also select (in a subsequent 'Message' window) whether or not to include file parameter comments (see **Section 23.2** for more information).



Import Third-Party Model This option is not active in IGW 5.0P.

Import Picture Selecting this allows the user to import a picture into the model. The cascading options allow the user to choose 'As BMP' or 'As JPEG'. (See **Chapter 5** for instructions on how to import pictures permanently.)

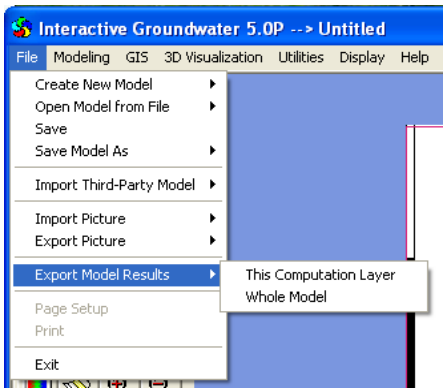
> **i** This function will only temporarily places a picture in the Working Area.

Export Picture

Selecting this allows the user to export the Working Area (discussed in **Section 3.13**) as a picture. The cascading options allow the user to choose 'As BMP' or 'As JPEG'. See **Section 23.1** for more advanced graphics capture options.

Export Model Results

Selecting this allows the user to save data for hydraulic head (m), concentration (ppm), seepage velocity (m/d) in x, y, and z directions and hydraulic conductivity (m/d) from the current model. A window titled 'Save As' appears, allowing the user to save the data file with a desired file name and specify the path. This selection is available only after the model has been discretized (see **Chapter 12**). See **Section 23.4** for more information.



> **i** Since the parameter units are internally assigned to metric system values in the model exporting interface, the user can not change them.

Additionally, Version 5.0P allows the user the option to save either one computational layer of their model, or the entire model altogether.

Page Setup

Selecting this allows the user to set the page up with their preferences. At this time the selection is inactive.

Print

Selecting this allows the user to print the active screen. A window prompting the printer function will appear, allowing the user to select the desired printing feature.

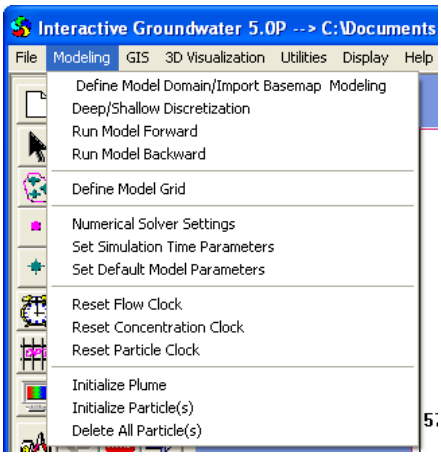
Exit

Selecting this exits the program. The software will prompt the user to confirm the desire to exit the software, if the model has not been saved.

> **i** Closing the Main Window is another way to exit the software.

3.3.2. Modeling Menu

The **Modeling** menu contains the following operations, all of which can be found on the button palette. The names of each operation are listed along with their respective location on the button palette, given in a (row, column) format:



Functions in this menu will be explained in later chapters.

3.3.3. GIS Menu

The **GIS** menu allows the user to bring files stored in GIS format into IGW. This menu contains **GIS Model Importer**, **County-Based Assistant**, and **GIS Exporter** buttons. See **Chapter 22** for further details.

3.3.4. 3D Visualization Menu

The **3D Visualization** menu is briefly explained below. More details are given in **Chapter 21**.

Show as 3D Surface This feature allows the user to view the results of the model as a three-dimensional surface, which can be manipulated in plan view for clarity.

Show as 3D Volume This feature opens a new window with many options that allow the user a complete toolbox of operations for viewing the model in three dimensions. Such options include cross-sectional and fence diagram manipulations of the model.

3.3.5. Utilities Menu

The **Utilities** menu contains the operations shown in **Figure 3-2**. Each of these operations is explained below:

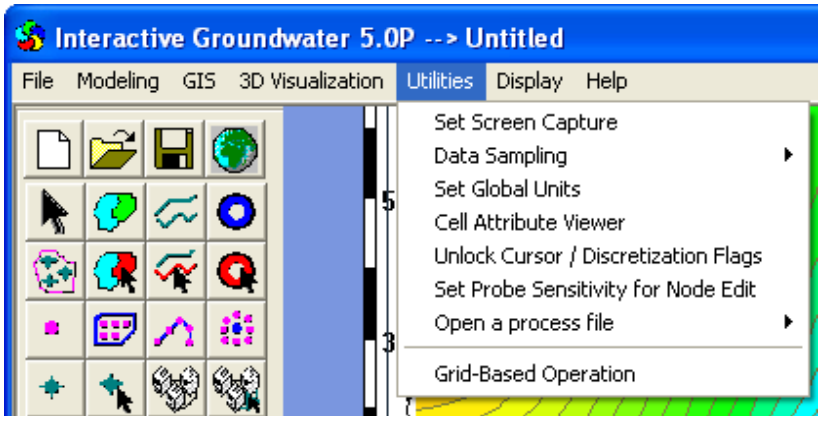


Figure 3-2 Utilities Menu

Set Screen Capture Selecting this brings up the 'Automatic Capture Window'. See **Section 23.1.1** for a detailed discussion of the window and its functionality.

Data Sampling Selecting this brings up two options: **Random Sampling**, and **Sampling at Calibhead**.

'Random Sampling' opens the 'Data Random Sampling' window (**Figure 23-3**) and allows the user to take a random sample of the data from the Working Area. This selection is available only after the model has been discretized (See **Chapter 12)**. Also see **Section 23.5** for more information.

'Sampling at Calibhead' compares samples at well locations and models the data as samples versus the predicted head. This feature will allow the user to see if the samples have been over-predicted or under-predicted, based on the results.

Set Global Units Selecting this feature allows the user to set the unit types on any of the variables needed in the model (**Table 3-1** and **Figure 3-3**).

> **i** To change the unit of a parameter, simply click the mouse to cycle through the available units until the desired one appears. m/day

Table 3-1 Available Units in IGW

UNIT TYPES	UNIT SYSTEMS		
	Metric	English	Customized
Length	m	ft	ft inch km mile m cm
Conductivity	m/day	ft/day	ft/day m/day cm/sec
Time	day	day	day hour sec month year
Concentration	kg/m^3	ppm	ppm ppb g/m^3 kg/m^3
Head	m	ft	ft inch km mile m cm
Recharge	m/day	inch/year	inch/year m/day cm/day cm/year ft/year
Leakance	1/day	1/day	1/day 1/hour 1/sec 1/month 1/year
Pumping Rate	m^3/day	GPM	GPM MGD m^3/day m^3/sec l/sec
Velocity	m/d	ft/day	ft/day m/year m/day
Solute	kg/day	lb/year	lb/year kg/year lb/day kg/day
Angle	degree	degree	degree radian
Area	m^2	acre	acre m^2 cm^2 ft^2 $inch^2$ km^2 $mile^2$
Specific Storage	1/m	1/ft	1/ft 1/inch 1/m 1/cm
Partioning Coef (K_d)	m^3/day	1/ppm	1/ppm m^3/kg m^3/g 1/ppb 1/ppm
Bulk Density	kg/m^3	g/m^3	g/m^3 kg/m^3 mg/m^3

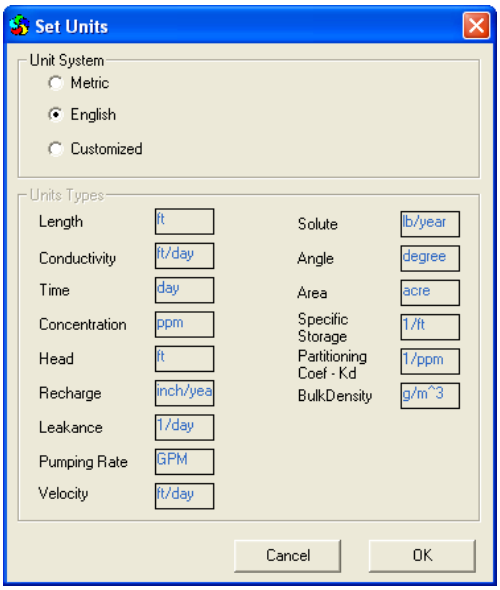


Figure 3-3 Set Units window

R

When entering numbers, interface first change the unit and then input the numerical value. When changing units, software automatically converts the existing numerical value into the new ones. Getting in this habit will prevent input errors when inputting data into the software.

Also, after changing the unit, be sure to delete all numbers from the numerical field before entering the new ones. Some numbers may be present in the field, but not visible, due to the number having many decimal places (and therefore not being completely displayed in the given field). Again, this helps prevent data entry errors.

Cell Attribute Viewer Selecting this allows the user to choose the desired parameters for the model. More information is found on this topic in **Section 3.12**.

Unlock Cursor/Discretization Flags Selecting this feature allows the user to unlock the cursor to reveal values in the Cursor-Activated Table. This feature also allows the user to make changes to the model without having to discretize it, offering flexibility to the user in terms of approach variations for a given model.

Set Probe Sensitivity for Node Edit This feature is used to increase or decrease the sensitivity area for cursor activation for node edit. Clicking on this item opens the 'Probing Setting for Node Edit' window as shown in **Figure 3-4**. See **Node Edit Mode** in **Section 3.16** for details.

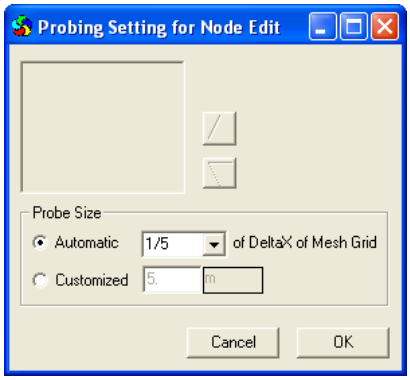


Figure 3-4 Probing Setting for Node Edit window

Open a process file This feature allows the user to open a Monitoring Well or Pline Flux data file. Please refer to **Section** for details**.**

Grid-Based Operation Selecting this feature allows the user to modify the grid using different array features. Grid based operations are explained in **Chapter 17**

3.3.6. Display Menu

The **Display** menu contains the following operations:

Zoom out Selecting this shrinks the Working Area and Working Area Attribute Display within the Model Screen. The 'Zoom out' button (see **Section 3.4**) performs the same function. See **Section 3.13** for information concerning the Working Area, the Working Area Attribute Display, and the Model Screen.

Zoom in Selecting this enlarges the Working Area and Working Area Attribute Display within the Model Screen. The 'Zoom in' button (see **Section 3.4**) performs the same function. See **Section 3.13** for information concerning the Working Area, the Working Area Attribute Display, and the Model Screen.

Refresh Selecting this updates the Working Area and other IGW Version 5.0P windows to include all recently changed parameters. The 'Refresh' button (see **Section 3.4**) performs the same function.

Display Options Selecting this opens the 'Main Model -- Draw Option' window. **Section 19.1** details the options available in this window for changing the IGW Version 5.0P display.

Change Display Property This feature is currently inactive in IGW Version 5.0P.

Attributes Explorer Selecting this feature access the Attributes Explorer window. For detailed information on this feature and all of its uses, see Error! eference source not found..

Show Toolbar By default, this feature is active. Un-selecting it will remove the button palette, SATDI and Working Area Display Tools from the Model Screen.

Show Model Notepad By default, this feature is active. Un-selecting this removes the Working Area Attribute Display (WAAD) from the Model Screen.

3.3.7. Help Menu

The **Help** menu contains the operations shown in **Figure 3-5**. Each of these operations is explained below:

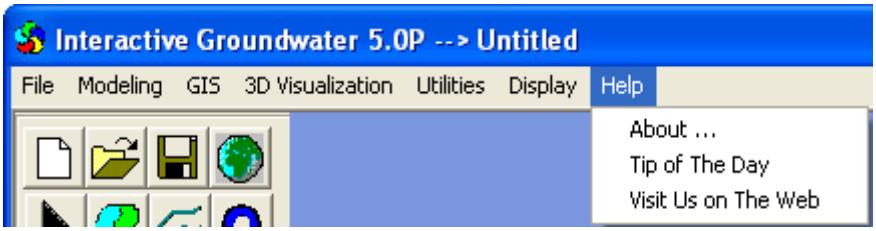


Figure 3-5 Help Menu

About … Selecting this opens the **Credits** window (see **Figure 3-6**). It is very similar to the splash screen that appears when the software is started. The **Credits** window lists the full software title, the names of the developers, and the home institution of current software development – the Department of Civil and Environmental Engineering at Michigan State University.

There is also a 'System Info…' button that opens the Windows® 'System Information' console window (or other appropriate interface – depending on Windows® version), and an 'OK' button used to close the 'About…' window. The IGW website may be opened by clicking on the web address.

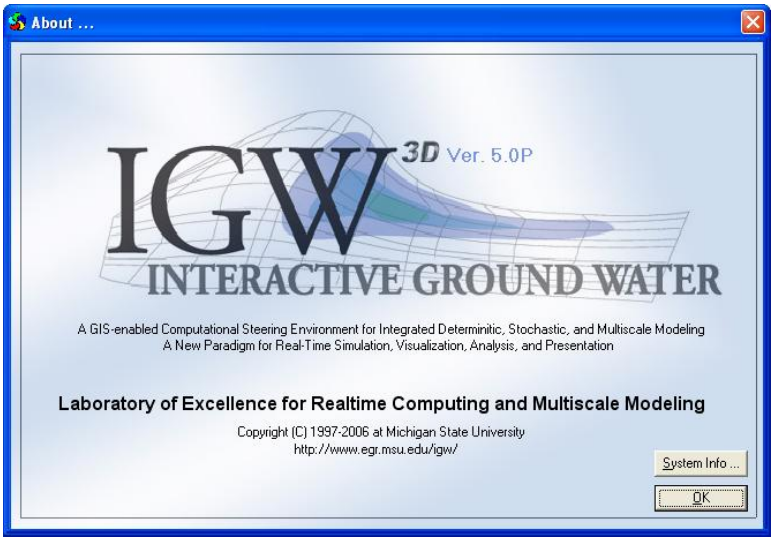


Figure 3-6 IGW Credits

Visit Us on The Web Selecting this opens the computer's web browser to http://www.egr.msu.edu/~lishug/research/igw. This is the IGW home page.

Tip of the Day Selecting this toggles the 'Tip of the Day' window on and off. A check mark appears in this window if it is active. The 'Tip of the Day' window is discussed in **Section 2.3**.

3.4. Button Palette

The 'Button Palette' is located just below the menu bar on the left-hand side of the main window (see **Figure 3-1**).

In the initial state, the software displays 48 buttons, all of which are active in IGW 5.0P. The button functions are explained further in the remainder of this section. The title of the button is appears in a bubble-box when the cursor is placed over it.

> **i** Due to differences in display resolution settings, the **Button Palette** may appear differently on some machines.

The location of each button is given next to its name in a **(row, column)** format that coincides with its location in the 12 x 4 palette.

Create New Project (1, 1)

Clicking this button opens a new model without saving the current work. It performs the same function as the 'New Model' operation in the **File** menu (**Section 3.3.1**).

Open Model (1, 2)

Clicking this button allows the user to open a previously-saved model in IGW.

Save model (1, 3)

Clicking this button allows the user to save the current model into any desired location

Define model domain/Import Basemap (1, 4)

Clicking this button initiates the process of setting a picture file as a basemap in the Working Area (see **Section 3.13**) by opening the 'Model Scale and Basemap' window. This process is discussed further in **Chapter 5**.

Reset Toolbar (2, 1)

Clicking this button resets the cursor from any previous state such as 'help mode' or 'draw mode'. The cursor is initialized to select new buttons or perform other functions

Create Zone/Assign Properties (2, 2)

Clicking this button allows the user to define a zone (polygon) within the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply click within the Working Area to define points that denote the outline of the zone. The zone creation process and other zone implementation information is discussed further in **Chapter 7**.

Create Polyline/Assign Properties (2, 3)

Clicking this button allows the user to define a polyline (a series of line segments) within the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply click within the Working Area to define points that correspond to the line-segment endpoints. The polyline creation process and other polyline implementation information is discussed further in **Chapter 8**.

Add Well (2, 4)

Clicking this button allows the user to define a well (point) within the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply click within the Working Area to define a point that corresponds to the location of the well. The well creation process and other well implementation information is discussed further in **Chapter 9**.

Modify Existing Zone (3, 1)

Clicking this button allows the user to replace the active zone with another zone, without having to redefine the zone attributes or any associated scatter points (**Section 7.3**). The first click of the button brings up one of two windows: 1) a 'Message' window appears with the text, 'You should select a zone first!' if no zone is currently selected; or 2) a 'Warning' windows appears with the text, 'Are you sure that you want to replace the current zone with a new one?', if there is an active zone.

If the 'Message' window appears, the user should click 'OK', select a zone and then re-select the 'Redefine Applied Area for a Zone' button.

If the 'Warning' window appears, verify that the correct zone is selected. If not, then select 'No', activate the desired zone, click the 'Reset toolbar buttons state' button, then re-select the 'Redefine Applied Area for a Zone' button. If the correct zone is selected, then click 'Yes'. The cursor enters 'draw mode' and the user may define a completely new zone to replace the old one. Once draw mode has been entered, the user may continually replace the previous zone until satisfied. This process is described further in **Section 7.5.**

Select/Edit Zone (3, 2)

Clicking this button allows the user the select a zone within the Working Area (see **Section 3.13**). The cursor is set to 'select mode' (see **Section 3.16**) and the user may simply click within a zone in the Working Area to select it. This is alternatively referred to as 'making the zone active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the Attribute Explore (AE) window. This process is described further in **Section 7.2**

Select/Edit Polyline (3, 3)

Clicking this button allows the user the select a polyline within the Working Area (see **Section 3.13**). The cursor is set to 'select mode' (see **Section 3.16**) and the user may simply click on a polyline in the Working Area to select it. This is alternatively referred to as 'making the polyline active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the Attribute Explore (AE) window (see **Section** Error! Reference source not ound.). This process is described further in **Section 8.2**.

Select/Edit Well (3, 4)

Clicking this button allows the user the select a well within the Working Area (see **Section 3.13**). The cursor is set to 'select mode' (see **Section 3.16**) and the user may simply click on a well in the Working Area to select it. This is alternatively referred to as 'making the well active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the Attribute Explore (AE) window (see **Section** Error! Reference source not ound.). More detail found in **Section 9.2.**

Add Single Particle (4, 1)

Clicking this button allows the user to add a single particle in the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply click within the Working Area to define a point that corresponds to the desired location of the particle. Particle implementation is discussed further in **Chapter 10.**

Add Particles Inside Polygon (4, 2)

Clicking this button allows the user to add a group of particles in the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply define a zone that outlines the desired location of the particles within the Working Area. Once the zone is defined, the 'Particles' window appears prompting the user to enter the number of particle columns desired. Once the number is entered, either click 'OK' to create the particle zone or 'Cancel' to abort. Particle implementation is discussed further in **Chapter 10**.

Add Particles Along Polyline (4, 3)

Clicking this button allows the user to add a polyline (a series of line segments) of particles in the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply define a polyline that indicates the desired location of the particles within the Working Area. Once the polyline is defined, the 'Particles' window appears prompting the user to enter the number of particles to be released along the polyline. Once the number is entered click 'OK' to create the particle polyline or 'Cancel' to abort. Particle implementation is discussed further in **Chapter 10**.

Add Particles Around Well(s) (4, 4)

Clicking this button allows the user to add particles to existing wells. This is discussed further in **Section 10.1.4**. Particle implementation is discussed further in **Chapter 10.**

Add Scatter Point (5, 1)

Clicking this button allows the user to add a scatter point (associated with a zone) to the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may click at any point in the Working Area to define a scatter point. This process may be repeated as desired without having to re-select the 'Add Scatter Point' button (however, be sure to wait for the crosshair cursor to reappear before defining another scatter point).

If no zone is active, attempting to add scatter points in the Working Area will bring up the 'Message' window with the text 'You should select a zone first!'. In this case, click the 'OK' button, select the desired zone, and then re-click the 'Add Scatter Point' button. Scatter point implementation is discussed further in **Section 7.7.**

> **i** Before adding scatter points, the user should first select the polygon onto which they will add the points. In order to access to scatter points, the corresponding zone should be activated.

Select Scatter Point (5, 2)

Clicking this button allows the user to select a scatter point within the Working Area (see **Section 3.13**). The zone associated with the desired scatter point should be active prior to clicking this button, or else the user will not be able to select the desired scatter point. The cursor is set to 'select mode' (see **Section 3.16**), and the user may simply click on a scatter point (associated with the active zone) in the Working Area to select it. This is alternatively referred to as 'making the scatter point active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the AE window (see **Section** Error! Reference ource not found.).

If no zone is active (or none are yet defined in the model – in which case no scatter points can exist yet and therefore trying to select one does not make much sense) then attempting to select a scatter point elicits the 'Warning' window with the text 'You should select a zone first!'. In this case click 'OK', select the zone associated with the desired scatter point, and re-click the 'Select Scatter Point' button. Scatter Point implementation is discussed further in **Section 7.7**.

Add 3D Attribute Model (5, 3)

This feature allows the user to treat the entire model by adding recharge to the first active layer within each node of the simulation for which these boundaries are established.

For instance, it could be that the water table is not present until layer 3 in the model, and therefore it makes sense to only apply recharge to this layer versus the top layer for best results. The user outlines a portion of the active area (or the entire active area if so desired) to apply this feature, and once discretized the model will add active recharge/evaporation only to the active layers

Select 3D Attribute Model (5, 4)

Clicking this button allows the user to select any of the 3D attribute sub-models that were created in the above step for adding recharge to an active layer of the simulation. This feature is identical to selecting a zone within a model.

Set Simulation Time Parameters (6, 1)

Clicking this button allows the user to edit the time parameters associated with the model by opening the 'Simulation Time Parameters' window. The time parameters are discussed further in **Chapter 11**.

Set/Edit Default Parameters for the Active Model (6, 2)

Clicking this button allows the user to change the model parameters of the active modeling layer assigned to zones when they are created by opening the 'Default Attribute' window. Those parameters are discussed further in **Chapter 6**.

Define Cross-section (6, 3)

Clicking this button allows the user to create a cross-section by defining its extent (as a series of line segments) within the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**) and the user may simply click within the Working Area to define points that correspond to the desired cross-section line-segment endpoints. The cross-section creation process and other implementation information is discussed further in **Chapter 16**. This process is discussed further in **Section 16.2.**

Deep Discretization (7, 1)

Clicking this button allows the user to adjust the nodal grid by opening the 'Define Model Grid' window. More information concerning the nodal grid is available in **Chapter 12**.

Shallow Discretization (7, 2)

Clicking this button applies the changes made in a conceptual model onto the numerical model (also referred to as 'discretizing the changes'). More information concerning model discretization is presented in **Chapter 12.**

Create Submodel (7, 3)

Clicking this button allows the user to define a submodel (polygon) within the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**), and the user may simply click within the Working Area to define points that denote the outline of the desired submodel. Use of this button is discussed in **Section 15.1**. The submodel creation process and other implementation information is discussed further in **Chapter 15**.

Select/Edit Submodel (7, 4)

Clicking this button allows the user to select a submodel within the Working Area (see **Section 3.13**). The cursor is set to 'select mode' (see **Section 3.16**) and the user may simply click within a submodel in the Working Area to select it. This is alternatively referred to as 'making the submodel active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the AE window (see **Chapter 4**). This process is discussed further in **Section 15.7**.

Display Options (8, 1)

Clicking this button allows the user to adjust numerous display parameters by opening the 'Main Model -- Draw Option' window. This is the same window that appears when **Option…** from the **Display** menu is selected. This window and its functionality are discussed further in **Section 19.1**.

Refresh Screen (8, 2)

Clicking this button causes all IGW Version 5.0P screens and windows to be redrawn with any incorporated changes such as window resizing or a changing of color for a certain attribute. It is not always necessary to click the 'Refresh Screen' button, as the software automatically updates most changes. Details concerning changing the display are presented in **Section 19.7**. Clicking this button is the same as selecting 'Refresh' on the **Display** menu – see **Section 3.3.6**.

Zoom In (8, 3)

Clicking this button enlarges the Working Area and Working Area Attribute Display within the Model Screen (see **Section 3.13**). This is the same as selecting 'Zoom in' on the **Display** menu – see **Section 3.3.6**. The 'Zoom in' button is discussed further in **Section 19.8**.

Zoom Out (8, 4)

Clicking this button shrinks the Working Area and Working Area Attribute Display within the Model Screen (see **Section 3.13**). This is the same as selecting 'Zoom out' on the **Display** menu – see **Section 3.3.6**. The 'Zoom out' button is discussed further in **Section 19.8**

Numerical Solver Settings (9, 1)

Clicking this button allows the user to adjust flow, transport, and stochastic model solver settings by opening the 'Solver' window. See **Chapter 13** for more information concerning the solver engine settings.

Run Model Backward (9, 2)

Clicking this button causes the software to track particles in the opposite direction of the velocity vectors. Backward particle tracking is presented in **Section 10.3.2**. If the button is inactive it will appear grayed out as shown to the right.

Pause/Stop Model (9, 3)

Clicking this button causes the software to stop the present simulation at the current state. The software will finish its calculations and the model redraws for the present time step before appearing idle

Run Model Forward (9, 4)

Clicking this button causes the software to solve the numerical model. If the model is set to transient state, or there are transport calculations to be done in a steady-state model, then the software will continually update as it proceeds through the simulation. Options for running the model are discussed in **Chapter 14**. The solver options are presented in **Chapter 13**. The model must be discretized (see **Chapter 12**) before attempting to solve, or else an error message will appear.

Reset Flow Clock (10, 1)

Clicking this button resets the 'Flow Time' display in the Step Adjustment and Time Display Interface (see **Section 3.5**), and the flow component of the elapsed time displayed in the Working Area Attribute Display (see **Section 3.13**).

Reset Concentration Clock (10, 2)

Clicking this button resets the 'Plume Time' display in the 'Step Adjustment and Time Display Interface' (see **Section 3.5**), and the plume component of the elapsed time displayed in the Working Area Attribute Display (see **Section 3.13**).

Initialize Plume (10, 3)

Clicking this button returns all concentration plumes to their original locations and resets the concentration values for all cells in the model.

Reset Particle Clock (10, 4)

Clicking this button resets the 'Particle Time' display in the Step Adjustment and Time Display Interface (see **Section 3.5**), and the particle component of the elapsed time displayed in the Working Area Attribute Display (see **Section 3.13**).

Add Text (11, 1)

Clicking this button allows the user to add a text field in the Working Area (see **Section 3.13**). The cursor is set to 'draw mode' (see **Section 3.16**), and the user may simply click at a point in the Working Area to designate a text field.

> **i** The clicked point corresponds to the upper-left-hand corner of the text field. Also, the text field will not be visible until text is actually typed into it via the AE window (**Section** Error! Reference source not found.).

Select Text (11, 2)

Clicking this button allows the user the select a text field within the Working Area (see **Section 3.13**). The cursor is set to 'select mode' (see **Section 3.16**) and the user may simply click on a text field in the Working Area to select it. This is alternatively referred to as 'making the text field active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the Attribute Explore (AE) window (see **Section** Error! Reference source not ound.). Text fields are discussed further in **Section 19.3.2**.

Initialize Particle(s) (11, 3)

Clicking this button returns all particles to their original locations. Particle implementation is discussed further in **Chapter 10**.

Delete Particle(s) (11, 4)

Clicking this button deletes all particles from the conceptual model. Particle implementation is discussed further in **Chapter 10**.

No Capture (12, 1)

Clicking this button turns off all capture options. Screen capture options are discussed in **Section 23.1.2**.

External Calling Capture (12, 3)

Clicking this button activates the time-step save feature and allows the user to invoke the manual timer function. Screen capture options are discussed in **Section 23.1.2**.

Timing Capture (12, 3)

Clicking this button activates the automatic timing screen capture option. Screen capture options are discussed in **Section 23.1.2**.

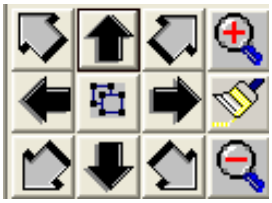
Set Capture Option (12, 4)

Clicking this button allows the user to edit screen capture options by opening the 'Automatic Capture' window. Screen capture options are discussed in **Section 23.1.2**.

3.5. Step-Adjustment and Time Display Interface

The 'Step Adjustment and Time Display Interface' (SATDI) is located on the left side of the main screen immediately below the Button Palette as highlighted in **Figure 3-1**. When model is running in transient mode, this area provides quick access to visualization for flow time, plume time and particle time. The area also provides interface to make changes in the length of time steps for the flow model, the plum and the particles. For more details on SATDI, please refer to **Section 11.2**.

3.6. Working Area Display Tools



There are 12 buttons in Working Area Display Tools. Clicking the arrow buttons in the area will move the entire Working Area and WAAD within the IGW main window. The buttons in the right most column are 'Zoom In', 'Refresh Screen', and 'Zoom Out'. They are exactly the same as those on the Buttons Palette (see **Section 3.4**).

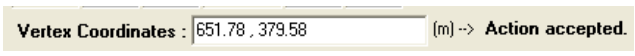
By clicking the button  in the center of all arrows brings the Working Area and WAAD back to default magnifications and location.

3.7. Layer Navigator

Layer navigator displays the number of current geological and computational layer in the model. Using the arrow buttons, one can navigate between geological and computational layers in the model.

> **i** The terms *geological layer* or *conceptual layer* convey exactly the same meaning in *IGW Version 5.0P* documentation. These terms also appear interchangeably in the *IGW Version 5.0P* interface and menus.

3.8. Vertex Coordinates Interface



'Vertex Coordinate Interface' (VCI) displays the coordinates of the last vertex defined, and allows the user to manually enter the coordinates of features when defining them in the Working Area. This provides for much greater accuracy compared to pointing and

clicking with the mouse. The user should enter the coordinates in simple Cartesian format by adding a ',end' for the final vertex. Note that only available unit for input in VCI is meter. The VCI text appears black (gray otherwise) only when the cursor is in 'draw mode' (see **Section 3.16**).

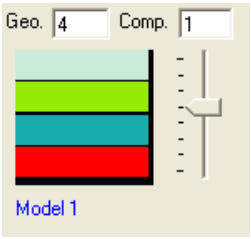
> **i** The 'end' command is not necessary when defining point features such as wells or single particles. Also, using the ',end' command to describe the feature before enough vertices [or points] have been defined, will result in the feature defining process being aborted. Certain feature-related status messages will also appear to the right of the VCI field (such as 'Action accepted' or 'Action accepted and ended'). Instead of using the 'end' command, the user may also double-click to finalize drawing a polyline.

3.9. Attributes Explorer Button

Clicking this button opens the Attributes Explore (AE) Window. AE is explained in detail in **Chapter 4**.

3.10. Layer Selector

The Layer Selector is another user interface to navigate though model layers. It also displays currently selected geological and computational layers in the top two boxes. The user can type in a desired computational and/or geological layer number in the fields provided at the top to switch between layers. The horizontal colored bars are graphical representation of geological or conceptual layers in the model. To the left of colored bars is a sliding button. Each tick mark along the sliding ruler represents a computation layer. The user can move to sliding button to move between the layers. Also see **Section 12.2**.



3.11. Grid Based Operations Button

Clicking this button opens the Grid Based Operations window. Grid Based Operations are explained in **Chapter 17**.

3.12. Cursor Activated Table (CAT)

The major portion of the right-hand side of the main window is occupied by what is known as the 'Cursor Activated Table' (CAT)[7] . The CAT displays the variable values in the model at any point where the cursor is located, for the current time step (graphic on right).

The left-hand side lists the name of the variable. The middle fields display the values and units associated with the variables (the unit field is user-changeable). The right-hand side contains a scroll bar that allows the user to view the desired variable if too many are displayed to be seen in the CAT simultaneously.

Clicking the 'Cell Attribute Viewer' button brings up the 'Choose Parameters at Cursor' window as shown in **Figure 3-7**. This window allows the user to select the parameters to be displayed in the CAT, and also generally defines

[7] Refer to **Chapter 5** of the ***IGW Version 5.0P Tutorials*** document for an interactive exploration of the CAT.

them (as the variable names displayed in the CAT are truncated). Refer to **Table 3-2** for expanded definitions of these variables and cross-references to other sections of this document.

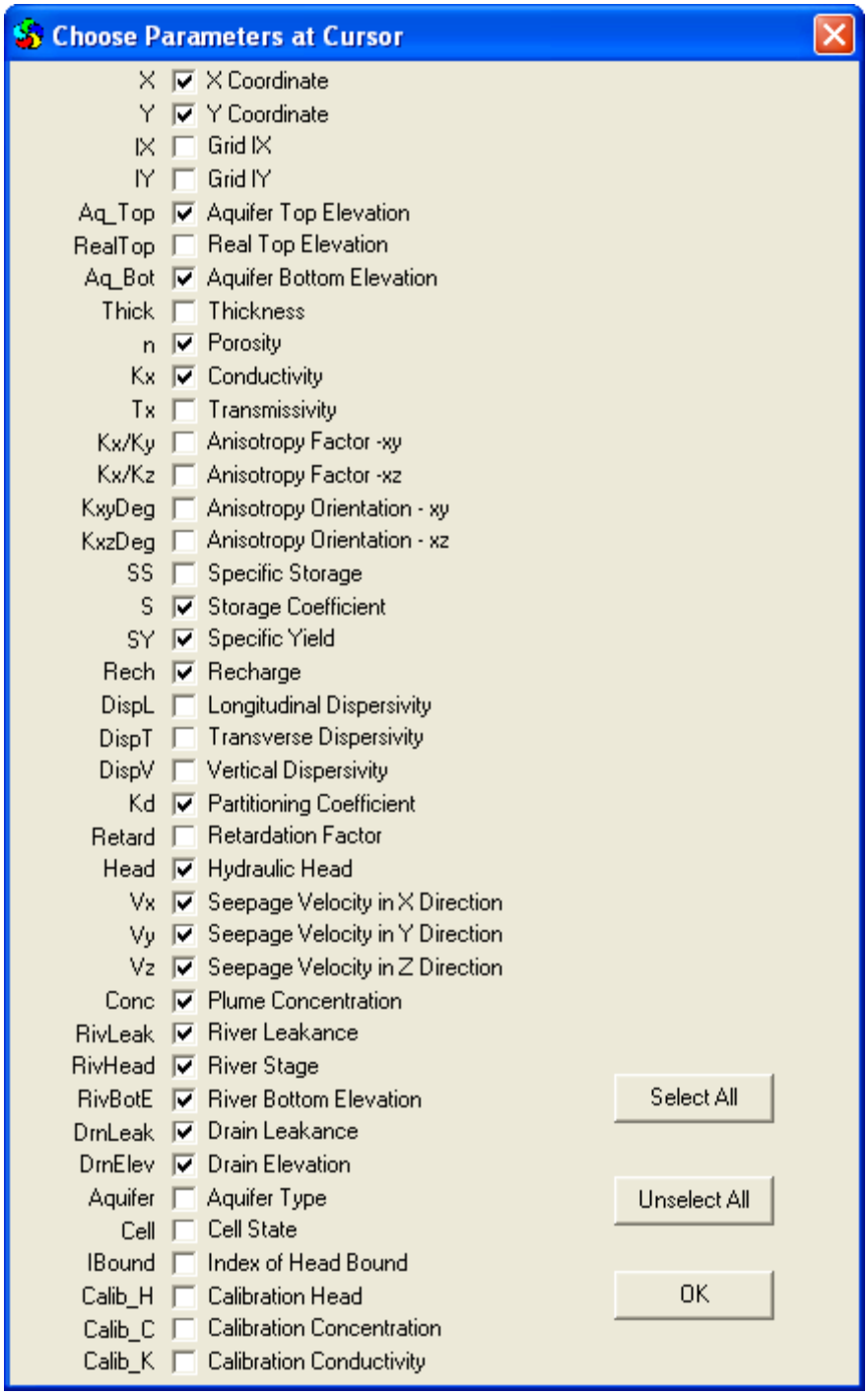


Figure 3-7 Parameters for Cell Attribute Viewer. Checked boxes are default selection.

Table 3-2 Expanded CAT Parameter Definitions

Para-meter	Association	Definition and References
X	X Coordinate	This indicates the x-coordinate of the cursor.
Y	Y Coordinate	This indicates the y-coordinate of the cursor.
IX	Grid IX	This indicates the column of the cell the cursor is located in.
IY	Grid IY	This indicates the row of the cell the cursor is located in.
Aq_Top	Aquifer Top Elevation	The elevation of the top of the aquifer. It is equivalent to 'Top Elevation'. See **Section 7.6.1.3**. Also see **Section 6.1** and **Section 6.2**.
RealTop	Real Top Elevation	The elevation of the ground surface. It is equivalent to 'Surface Elevation'. See **Section 7.6.1.3**. Also see **Section 6.1** and **Section 6.2**.
Aq_Bot	Aquifer Bottom Elevation	The elevation of the bottom of the aquifer. It is equivalent to 'Bottom Elevation'. See **Section 7.6.1.3**. Also see **Section 6.1** and **Section 6.2**.
Thick	Thickness	The thickness of the aquifer. It is equal to 'Aq_Top - Aq_Bot'.
n	Effective Porosity	The effective porosity of the aquifer material. It is equivalent to 'Effective Porosity'. See **Section 7.6.1.1**. Also see **Section 6.1** and **Section 6.2**.
Kx	Conductivity	The x-direction conductivity of the aquifer material. See **Section 7.6.1.1**. Also see **Section 6.1** and **Section 6.2**.
Tx	Transmissivity	The x-direction transmissivity of the aquifer material. It is equal to 'Kx * Thick'.
Kx/Ky and Kx/Kz	Anisotropy Factors	The ratio of the x-direction conductivity to the y-direction conductivity (or the z-direction conductivity). See **Section 7.6.1.1**. Also see **Section 6.1** and **Section 6.2**.
Kxy degree	Anisotropy Orientation	The angle between the x-axis and the Kx axis (in the x-y plane). It is equivalent to 'Orientation of AnisF in XY'. See **Section 7.6.1.1**. Also see **Section 6.1** and **Section 6.2**.
SS	Specific Storage	See **Section 7.6.1.1**. Also see **Section 6.1**.
S	Storage Coefficient	This is equal to 'Specific Storage * Thick'.
SY	Specific Yield	See **Section 7.6.1.1**. Also see **Section 6.1** and **Section 6.2**.
DispL	Longitudinal Dispersivity	See 'Local Dispersivity' in **Section 7.6.1.1**.
DispT	Transverse Dispersivity	See 'Local Dispersivity' in **Section 7.6.1.1**.
DispV	Vertical Dispersivity	See 'Local Dispersivity' in **Section 7.6.1.1**.
Kd	Partitioning Coefficient	See 'Partitioning - Kd' in **Section 7.6.1.2**.
Retard	Retardation Factor	This is equal to '1 + [[Bulk Density/Effective Porosity] * [Partitioning Coefficient]]'. See '**Soil Particle Density**' in **Section 7.6.1.1** for an explanation of 'Bulk Density'.
Head	Hydraulic Head	This is the hydraulic head in the aquifer.
Vx	Velocity in X Direction	Groundwater velocity in X direction.
Vy	Velocity in Y Direction	Groundwater velocity in Y direction.
Vz	Velocity in Y Direction	Groundwater velocity in Z direction.
Conc	Plume Concentration	This is the concentration of contaminants in the aquifer.
RivLeak	River leakance	This is the leakance factor associated with a head dependent flux component (here, river). Leakance is defined in the '**Drain**' subsection in **Section 7.6.1.2**. River features are discussed in the '**River**' subsection in **Section 7.6.1.2**.
RivHead	River stage	This is the stage assigned to a head dependent flux component (here, river). River features are discussed in the '**River**' subsection in **Section 7.6.1.2**.
RivBotE	River Bottom Elevation	This is the bottom elevation of a head dependent flux component (here, river). River features are discussed in the '**River**' subsection in **Section 7.6.1.2**.
DrnLeak	Drain Leakance	This is the leakance factor associated with a drain feature. Leakance is defined and drain features are discussed in the '**Drain**' subsection in **Section 7.6.1.2.**
DrnElev	Drain Elevation	This is the elevation assigned to a drain feature. Drain features are discussed in the '**Drain**' subsection in **Section 7.6.1.2**.
Aquifer	Aquifer Type	This indicates whether the aquifer is 'Confined' or 'Unconfined'. See **Appendix A-I**.
Cell	Cell State	This field indicates whether the cell is 'Wet', 'Dry', or 'Inactive'. See **Appendix A-II**.
Ibound	Index of Head Bound	This field indicates whether the cell is 'Active' (meaning the head is calculated), 'Inactive' (meaning no head is calculated for the cell), or 'ConstH' (meaning the cell has a prescribed head value). See **Appendix A-III**.
Calib_H	Calibration Head	This field indicates the value of the calibration data for head.
Calib_C	Calibration Conductivity	This field indicates the value of the calibration data for concentration.
Calib_K	Calibration Conductivity	This field indicates the value of the calibration data for conductivity.

A check mark should be placed in the boxes corresponding to the desired parameters. The user may also use the 'Select All' and 'Unselect All' buttons as desired. Clicking the 'OK' button in the 'Choose Parameters at Cursor' window makes the desired changes appear in the CAT.

> **i** Ibound, Calib_H, Calib_C, and Calib_K cannot be visualized in the Cell Attribute Viewer at this time.

In the CAT, 'Inactive' will be displayed in all but the 'X', 'Y', 'IX', and 'IY' fields when the cursor is placed in an inactive cell.

> **i** If a specific parameter is not assigned for a particular cell, then a value of N/A will be displayed for that cell in the CAT.

3.13. Working Area

The remainder of the Main Window is referred to as the 'Model Screen'. It is pictured in **Figure 3-8.**

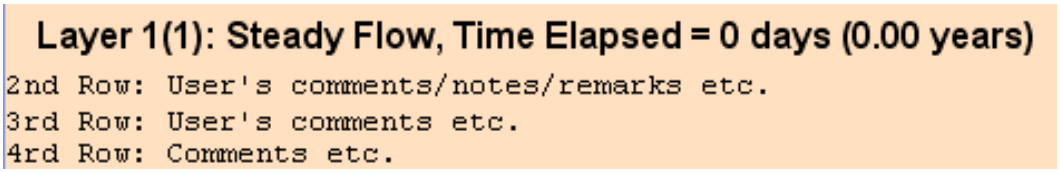
Figure 3-8 The Model Screen

The Model Screen exists mainly to provide a background against which the 'Working Area' (the large white rectangle) and the 'Working Area Attribute Display' WAAD (the peach rectangle) can be displayed.

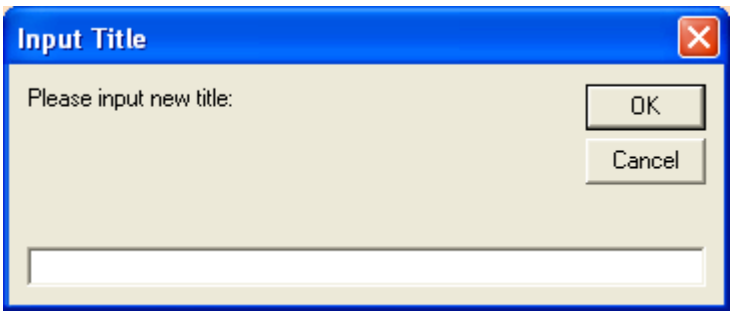
The Working Area is the region where the conceptual modeling is performed and subsequent solutions are obtained. It can be displayed anywhere within the Model Screen, and resized using the 'Zoom in' and 'Zoom out' buttons. It is not restricted by the size of the model screen: if it is larger than the model screen edges will appear to go behind the other software interface components (i.e. the CAT, Button Palette, SATDI, VCI, LMA / RMA, etc.), and even off the edges of the monitor.

3.14. Working Area Attributes Display

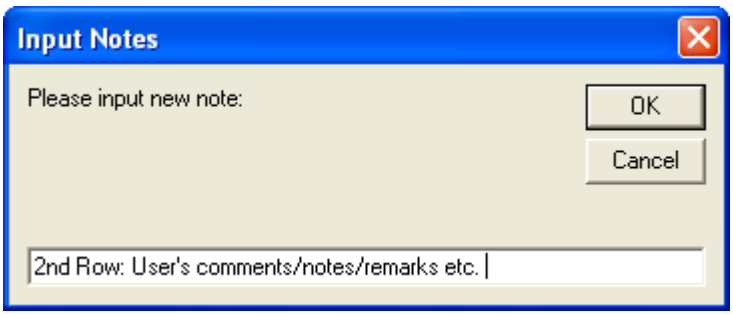
The Working Area Attributes Display or WAAD is attached to the bottom of the Working Area. The user can have up to four rows of text in WAAD as shown below.



The first row appears in a boldface font with default text as shown above. The default text contains information about the selected model layer [Geological (Computational)], type of flow regime (steady or transient) and the simulation time elapsed. User can also append text before the default text. However, the default text cannot be removed from the first row. The appended text appears in the same font as the default text. By clicking once anywhere in the first row the 'Input Title' window appears where user can add text up to 254 characters (including spaces). Any text added in this window is appended before the default text.



In the bottom three rows, user can add any desired text. Clicking once anywhere in the desired row opens the 'Input Notes' window. The user can add text up to 254 characters (including spaces) in any single row. The font style and size is fixed for these rows. The text does not wrap from one row to the other. The text in access to the row length (depending on the size of the WAAD) will simply not show, but can always be seen/accessed in the 'Input Notes' Window.



3.15. Status Bar

The Status bar is located at the bottom of IGW main window. It contains three slender gray boxes. Different messages will display in these boxes concerning the model solution status.

3.16. The Cursor

The cursor is important to using IGW Version 5.0P because of the software's graphical interface. The cursor has a number of modes (discussed in **Section 3.4)** that it may enter, depending on the current status of the software. These include:

- **Default mode**: The default cursor mode. The cursor always appears as its default shape. This mode is used for performing basic Windows™ functions and using the buttons and menus in the software.  In order to initialize the cursor, the default mode should be used.

- **Draw and add text mode**: This mode is used when defining features and adding text in the Working Area. The cursor appears as a crosshair when it is positioned in the Working Area.

- **Select mode**: This mode is used when selecting features in the Working Area. The cursor appears with a question mark next to it when positioned in the Working Area.

- **Node edit mode**: To modify a feature (polygon or polyline), select the feature and then right-click on the feature. A menu pops up. Select 'Edit Node' on the menu. All existing nodes on the feature turn into solid black boxes. A blue cross hair similar to a + sign in the middle of every two consecutive nodes is revealed, which also serves as a node. The cursor appears with a question mark next to it (when in the Working Area), but when the cursor is brought closer to either the black box or the + sign, its shape turns to  . Holding the left mouse button the node can be dragged to the new desired location. If a vertex is moved so that it lies directly between two other vertices, the software will automatically eliminate that vertex (as it is redundant and no longer necessary). See the sections on redefining features (**Section 7.3**) for zones and (**Section 8.3**) for polylines) for more information.

  Using the **Set Probe Sensitivity for Node Edit** option from the **Utilities** menu (**Section 3.3.5**), the user can delineate a 'region' around the node within which the cursor changes shape and dragging the mouse for editing becomes enabled. Clicking this option in **Utilities** menu opens the **Probing Setting for Node Edit** window shown in **Figure 3-9**. The default region for cursor activation for node edit is 1/5 of DeltaX of Mesh Grid. User can increase or decrease this from 1/100 to 100 DeltaX using the dropdown choices from Automatic setting. User can also assign a Customized distance for node edit.

R	When the model grid size is very fine then the default sensitivity for node edit may be only a few pixels and it may become difficult to activate the cursor into the edit mode  unless the mouse is very accurately pointing at the node. In such situations, one can increase the node sensitivity to make node editing easier.

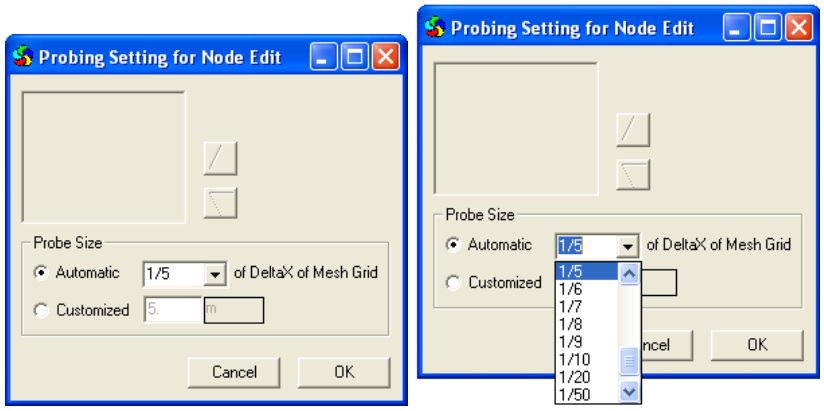
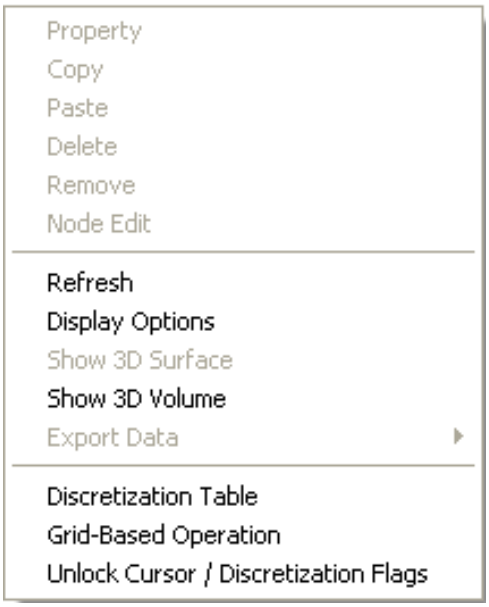


Figure 3-9 Probing Settings for Node Edit with dropdown choices

> **i** When the model is running, the cursor turns into an hourglass.

3.17. The Right-Click Menu

When the cursor is in the Working Area the user may right-click the mouse to access a special drop-down menu that has the following entries: 'Property', 'Copy', 'Paste', 'Delete', 'Remove', 'Node Edit', 'Refresh', 'Display Options', 'Show 3D Surface', 'Show 3D Volume', 'Export Data', 'Discretization Table', 'Grid-Based Operation", and 'Unlock Cursor / Discretization Flags'.



When no feature is selected, the 'Refresh', 'Display Options', 'Show 3D Volume', 'Discretization Table', 'Grid-Based Operation', and 'Unlock Cursor / Discretization Flags' entries are available.

Selecting 'Refresh' causes the software to update the Working Area and all windows to reflect any recent changes (see **Section 19.7**).

Selecting 'Display Options' opens the 'Main Model -- Draw Option' window (see **Section 19.1**).

Selecting 'Show 3D Volume' opens the model in a new window with a three-dimensional, editable format. More information on this feature is found in **Chapter 21.**

Selecting 'Discretization Table' opens the Discretization Table window. More information on this feature is found in **Section 12.4**.

Selecting 'Grid-Based Operation' opens this feature, in which the user has several options for model manipulation. More detailed explanation of this feature is found in **Chapter 17.**

Selecting 'Unlock Cursor / Discretization Flags' allows the user to see the values of the variables in the right-hand pane as the cursor is moved over them, in an active model.

When a feature is selected, the other entries also become available (see graphic below). Note: 'Paste, 'Delete', 'Remove', 'Show 3D Surface' and 'Export Data' are currently not implemented.

Selecting ‘Property’ opens up the AE window (see **Chapter 4**) with the selected feature attributes accessed.

Selecting ‘Copy’ allows one to copy a feature in the model and then place it either into a different layer of the model, or a different model altogether.

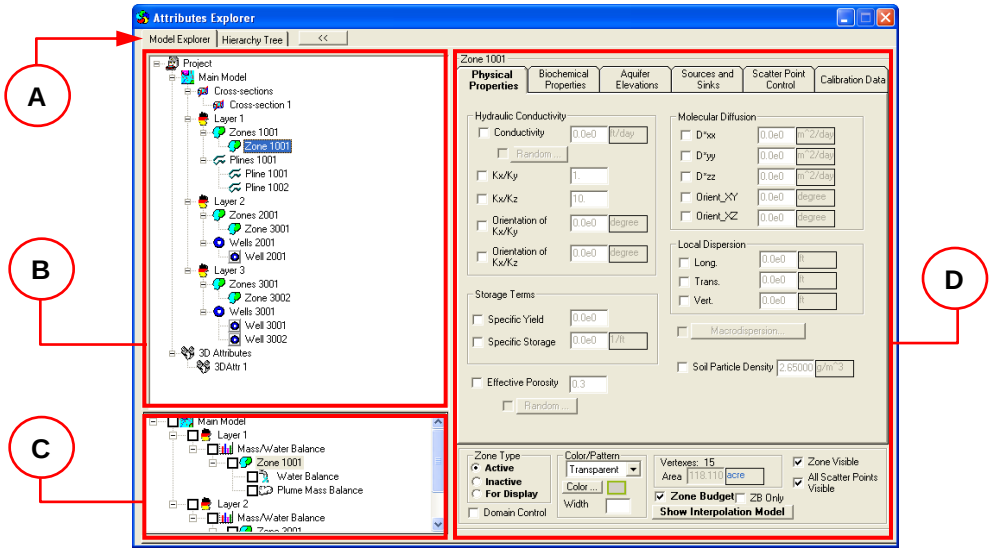
Selecting ‘Node Edit’ either puts the cursor in ‘Node Edit’ mode (see **Section 3.16**) or takes the cursor out of ‘Node Edit’ mode. The current state is indicated by the presence (or lack of) a check mark to the left of the ‘Node Edit’ entry.

!	In order to edit polylines and polygons, the user should click the related selection button or right-click on the polyline or polygon which will be edited, and select ‘Node Edit’.

i	When the user switches to node editing mode, a new vertex is automatically assigned to the middle point between two existing nodes. The user is free to change the location of that vertex in order to edit the nodes.

While the features of the Main Window provide access to most of the functions of IGW Version 5.0P, there are some operations that need to be performed through other interfaces. Attributes Explorer (AE)[8] is a secondary interface mechanism described in this chapter.

The AE is the IGW Version 5.0P interface that provides access to the attributes of all features defined in the model. It is pictured in **Figure 4-1**.



A – Layer tabs (Model Explorer and Hierarchy Tree)
B – Left Hand Pane (LHP)
C – Time Processor Selector (TPS)
D – Right Hand Pane (RHP)

Figure 4-1 Attributes Explorer (AE)

The AE opens when the software starts but is located out of the way with only a portion of the title bar visible in the lower right-hand corner of the user's monitor. It is a separate window, so it can be moved independently from the rest of the IGW Version 5.0P interface. The AE will need to be moved often as 1) it is used to define attributes for every model feature, and 2) it is always displayed in front of the Main Window, therefore obstructing its view.

To move the AE, one can click and drag the window to its desired location from the bottom right corner of the viewing screen, and then minimize it for easy access later on at any time.

As with any other window, the AE can be closed, but it will only remain closed until another feature is defined in the Working Area- at which point it will reopen in the center of the screen. If it is needed again prior to defining another feature, simply select the 'Attributes Explorer' button, located above the Layer Selector (see **Figure 3-1** and **Section 3.9**). 

The AE consists of two layers. The 'Model Explore' layer is visible by default (see **Figure 4-1**). Behind the Model Explore layer is the 'Hierarchy Tree' layer. Hierarchy Tree layer is not active in IGW Version 5.0P. Only Model Explorer Layer is discussed in the following subsections.

> **!** In IGW 2D, there is an 'Apply' button to introduce changes. Note there is not an 'Apply' button in IGW Version 5.0P. Values entered into the AE are automatically updated.

[8] Refer to **Chapter 6** of the ***IGW Version 4.7 Tutorials*** document for an interactive exploration of the AE.

4.1. The Model Explorer

The Model Explorer is where model features are selected and attributes defined. It is divided into three main portions viz. Left Hand Pane (LHP), Right Hand Pane (RHP) and Time Processor Selector (TPS). These three portions are highlighted in **Figure 4-1**. LHP displays a hierarchical representation of the model features for easy selection. RHP shows the present attributes for the feature selected (in the LHP) and allows for changing them. TPS shows all processes in the model which are time dependent (eg. water budget, concentrations etc.). Each of these are discussed in more details in the following sections

4.1.1. The Left-hand Pane (LHP)

Notice the hierarchy of model features. They can be selected by simply clicking on the desired feature. When a feature is selected, the RHP changes accordingly. Each entry group has a box next to it with either a (+) or (–), indicating that the group can be expanded to show either individual features / subgroups within it, or collapsed to hide them.

The first hierarchical level is called 'Project'. Here the user may enter information concerning the overall project.

The second hierarchical level is called 'Main Model'. The associated interface simply provides alternate access to some of the more commonly used buttons from the Button Palette. The submodel and cross-section groups also appear at this level.

The third hierarchical level is referred to as 'Layers'. This is where the user will find all computational layers of their model.

These first three levels are common to all models, and will appear as features is defined in the Working Area once the model is created. The associated RHP configurations are discussed in **Section 4.1.2**.

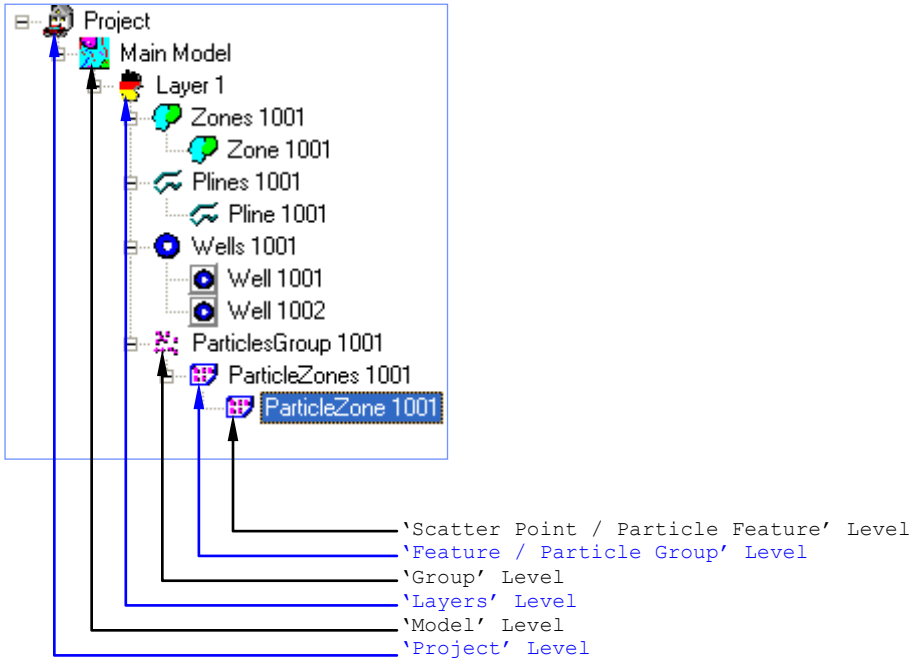


Figure 4-2 General view of LHP

Three more hierarchical levels that appear as certain features can be added to the model.

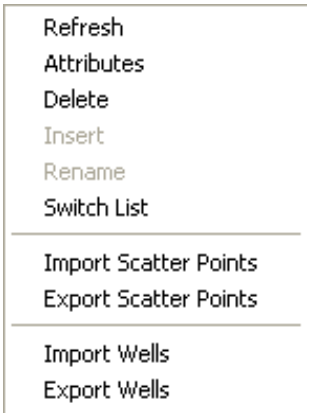
The fourth level is called 'Group'. All features of a certain type are grouped together under placeholders titled 'Zones', 'Plines', 'Wells', etc. The associated interface for these groups is blank.

The fifth level is called 'Feature/Particle Group'. Individual features are listed at this level along with group placeholders for the individual particle feature types. The feature RHPs are each discussed in the chapter dealing with that respective feature. The RHPs associated with the particle feature groups are blank.

The sixth level is called 'Scatter Point/Particle Feature'. Scatter points are listed under the zone they are associated with, and individual particle features are listed here in their respective groups. The RHPs are briefly discussed in **Section 4.1.2**. The specific chapter that deals with a feature contains more discussion on RHP for that feature.

The LHP also has an alternate 'Interpolation Model' view that is associated with scatter point functionality. See **Section 7.7.6.1**.

The user may also right-click on an items in the LHP to access a list of functions associated with that entry. Right click pop-up menu is shown at the right and its entries explained below:



- Refresh – refreshes LHP with latest changes
- 'Attributes' – shows the RHP associated with the feature
- 'Delete' – deletes the feature
- 'Insert' – inserts features
- 'Rename' – renames the feature
- 'Switch List' – toggles between the standard and alternate view of the LHP (see **Section 7.7.6.1**)
- 'Import Scatter Points' – allows the user to import a set of scatter point wells in the selected zone that were created either in another model or outside of the program.
- 'Export Scatter Points' - allows the user to save the scatter points from their model for future use.
- 'Import Wells' – the user can bring in well data from another IGW model that was previously created.
- 'Export Wells' - the user can save wells created in their model for future use.

In order to import/export scatter points/wells in IGW, the most common format is CSV ('Comma Separated Value)'. This file format is used to exchange data between different applications, and can be edited using a text/spreadsheet editor (e.g. Microsoft Excel®). File structure for importing/exporting scatter points in CSV format is given at **Table 4-1** and **Table 4-2**

I The user is free to import/export as many attributes as they want (or that IGW allows) for scatter points.

Table 4-1 Import/Export Structure for Scatter Points

Scatter Points Data file -- Generated by IGW					
Layer	1	Zone	Zone 1001	Attributes	4
Well#	X	Y	Cond	TopE	
Scatter point 1001	692.0230089	370.0657895	30.48	3.048	
Scatter point 1002	635.2795893	362.6644737	15.24	4.572	
Scatter point 1003	624.1776159	275.0822368	18.288	3.6576	
Scatter point 1004	658.7170887	331.8256579	21.336	3.9624	

I

Entries in the spreadsheet must match exactly with the desired zone in the model. For example, if attempting to read a scatter point file into **Zone 1001** in the model, the field in the scatter point file must read **Zone 1001**. If these are not consistent, the software will either:

1) Display a warning message indicating there is an inconsistency (if there is no zone with the specified title), or
2) Read the scatter points into the incorrect zone (if there is a zone with the specified title).

By default, exported data in CSV format has at least two attributes excluding scatter point ID (1001 to 1004). These are the X and Y coordinates of that particular scatter point (whose values change as the user changes them). In the AE, if any modeling parameters are introduced other than data point coordinates, the spreadsheet file will list those as well. Using **Table 4-1**, both the hydraulic conductivity values and the top elevations of the aquifer, from scatter points 1001 to 1004, are entered into the model. In this table, 'Layer' corresponds to the active modeling layer to which conceptual modeling components (i.e. zones and scatter points) are appended. 'Attributes' corresponds to the number of data types assigned to the scatter points. 'Zone 1001' indicates the active model area onto which scatter points are appended.

i

The coordinates of a scatter point do not necessarily have to fall inside of the associated zone, but a scatter point cannot be entered without being associated to a zone.

File structure for importing/exporting scatter points is almost the same as that for importing/exporting wells. The only difference will be the number of available data types, as wells do not have as many attributes as scatter points. In **Table 4-2,** the CSV spreadsheet structure for imported/exported wells can be seen. Here, excluding the well ID value (1001 to 1004), there are a minimum of two attributes, these being the X and Y coordinates (values change as the user edits them). As additional data are entered in the Attribute Explorer, they are appended to the spreadsheet. In this example, the new data are for the top of the screen interval, bottom of the screen interval and pumping rate for that particular well. The number of attributes are shown on the same spreadsheet.

Table 4-2 Import/export structure for wells

Wells data file -- Generated by IGW					
5					
Well #	X	Y	Screen_Top	Screen_Bot	Q
Well 1001	320.7236762	407.0723684	-23.333	-36.666	-2500
Well 1002	620.4769581	445.3125	-24.384	-38.1	-2500
Well 1003	571.1348541	319.4901316	-25.908	-37.1856	-2500
Well 1004	486.0197246	416.9407895	-24.9936	-35.052	-2500

! In order to import scatter points/wells to IGW, the data base structure should be the same as the spreadsheet exported from IGW. The user may choose the sequence to follow through columns of the spreadsheet, by simply exporting a dataset from IGW. The same structure can then easily be applied for the CSV file that will be imported.

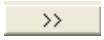
The 'Export scatter points/wells' command converts data to the metric system, regardless of the unit system the data are in presently. Before importing scatter points/wells to IGW, data should be entered in metric system units, for convenience.

i There are two other file formats, SPF and SWF. These formats are not currently used in the IGW importing/exporting procedure.

When an entry is selected in the LHP, it will be highlighted with a blue box. In addition, if a model feature is selected, its corresponding entry in the Working Area will be outlined in red. Features selected in the Working Area will automatically become highlighted in the AE.

When a feature entry is highlighted in LHP, clicking on it a second time (i.e. clicking on it again) will allow the user to edit its name.

If entries in the LHP do not fit within the allocated space, a scroll bar will appear at the bottom of the LHP to allow the user to pan over the extent of the LHP.

The expansion button  next to the layer tabs may also be used to enlarge the view of LHP.

4.1.2. The Right-hand Pane (RHP)

The contents and appearance of RHP is dependent on what is selected in the LHP. The RHP is where attributes are entered for the selected features of the model. This section describes some RHPs corresponding to some model features and provides reference to RHPs related to other features.

4.1.2.1. Project

The RHP for the 'Project' entry on the Project level is shown in **Figure 4-3**. The user may enter project identification data such as the name, manager, team, etc. to be stored in the file. The user may also select the 'Preferences' box and then define the directory path for either the project files or GIS files used in the model, as shown in **Figure 4-4**.

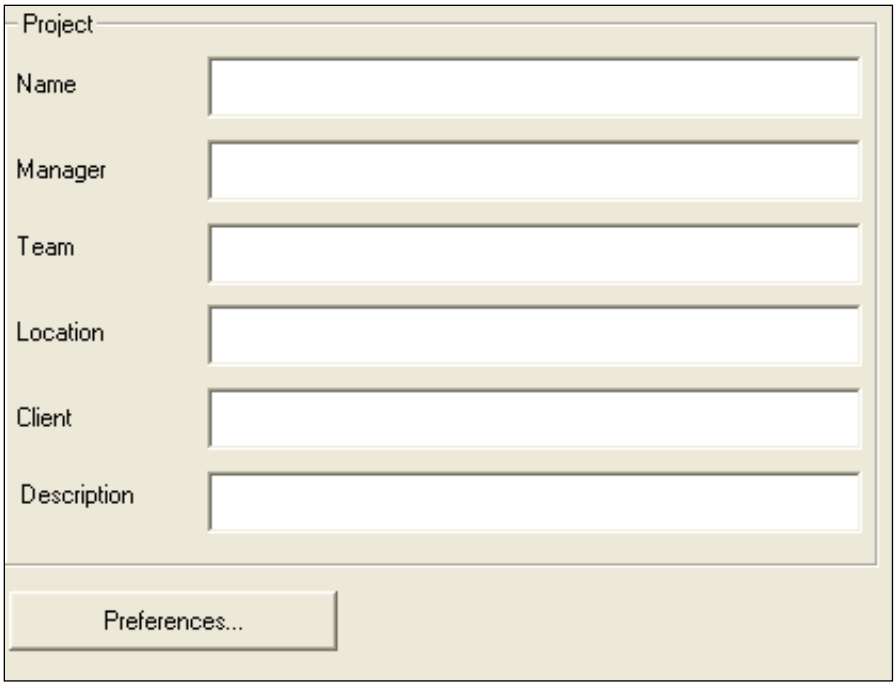


Figure 4-3 RHP for Project Identification

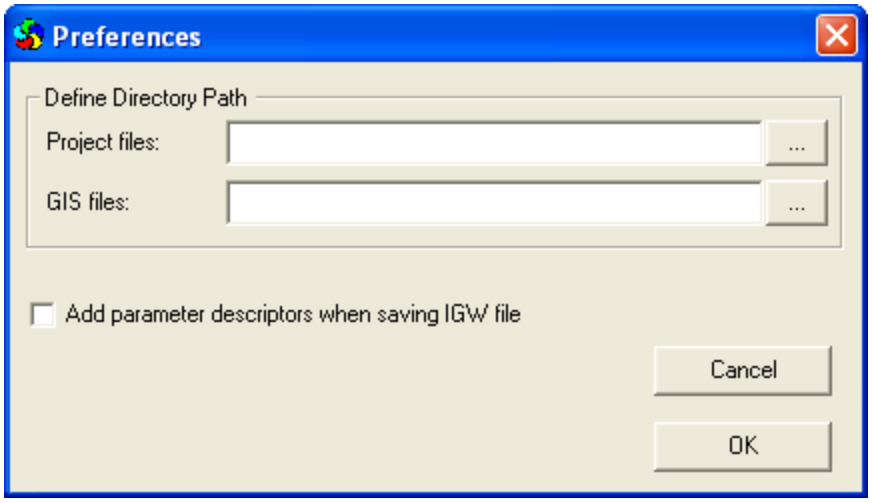


Figure 4-4 Preferences Box for Project Screen

4.1.2.2. Main Model

The RHP for the 'Model' entry on the Model level is shown in **Figure 4-5**. The buttons associated with the icons perform the same functions as the comparable buttons on the Button Palette in the Main Window. This RHP simply provides another access point.

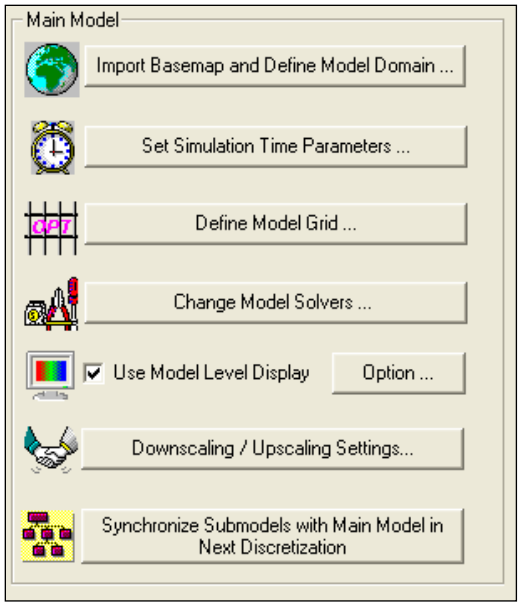


Figure 4-5 RHP for General Model Options

4.1.2.3. Layer

The RHP for a layer on the Layer / Submodel level is shown in **Figure 4-6**. This screen has default parameters for the entire layer such as elevation, thickness, recharge, etc. **Chapter 7** provides a detailed explanation of each of these features.

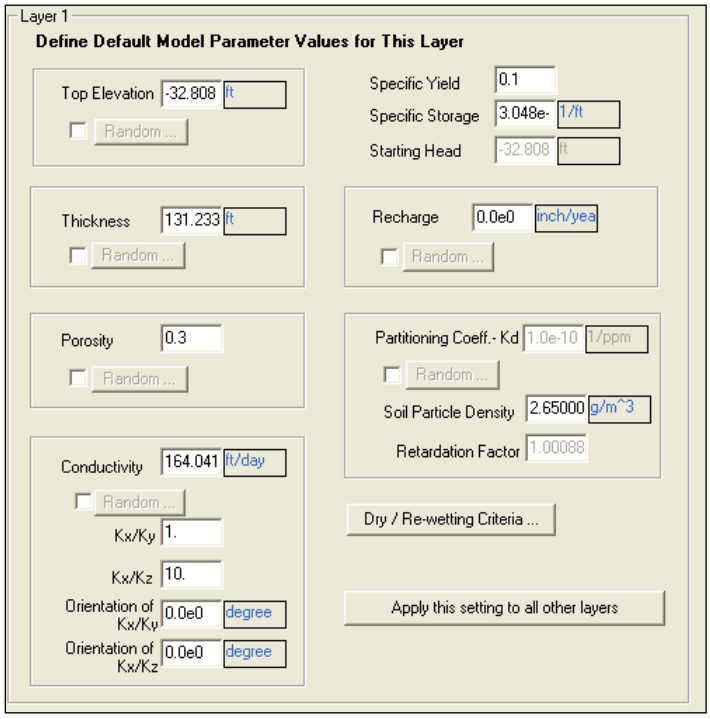


Figure 4-6 **RHP for Layers**

4.1.2.4. Zones

The RHP for the 'Zones' entry on the Group level provides a location to add **Multipliers for Sensitivity Analysis**. This RHP allows the user to manipulate the model consistently with multiplier factors, versus the more tedious task of having to hand-edit every single zone. There are three tabs in this window. There are three main tabs on this RHP.

Physical Properties tab has the multipliers for Hydraulic Conductivity, Retardation, Storage Terms, Molecular Diffusion, Local Dispersion, and First Order Decay. The user can enter different multipliers for different model layers or click '**Apply this setting to polygons in all other layers**' button to use the same multiplier(s) in all layers. Physical Properties tab is shown in **Figure 4-7**

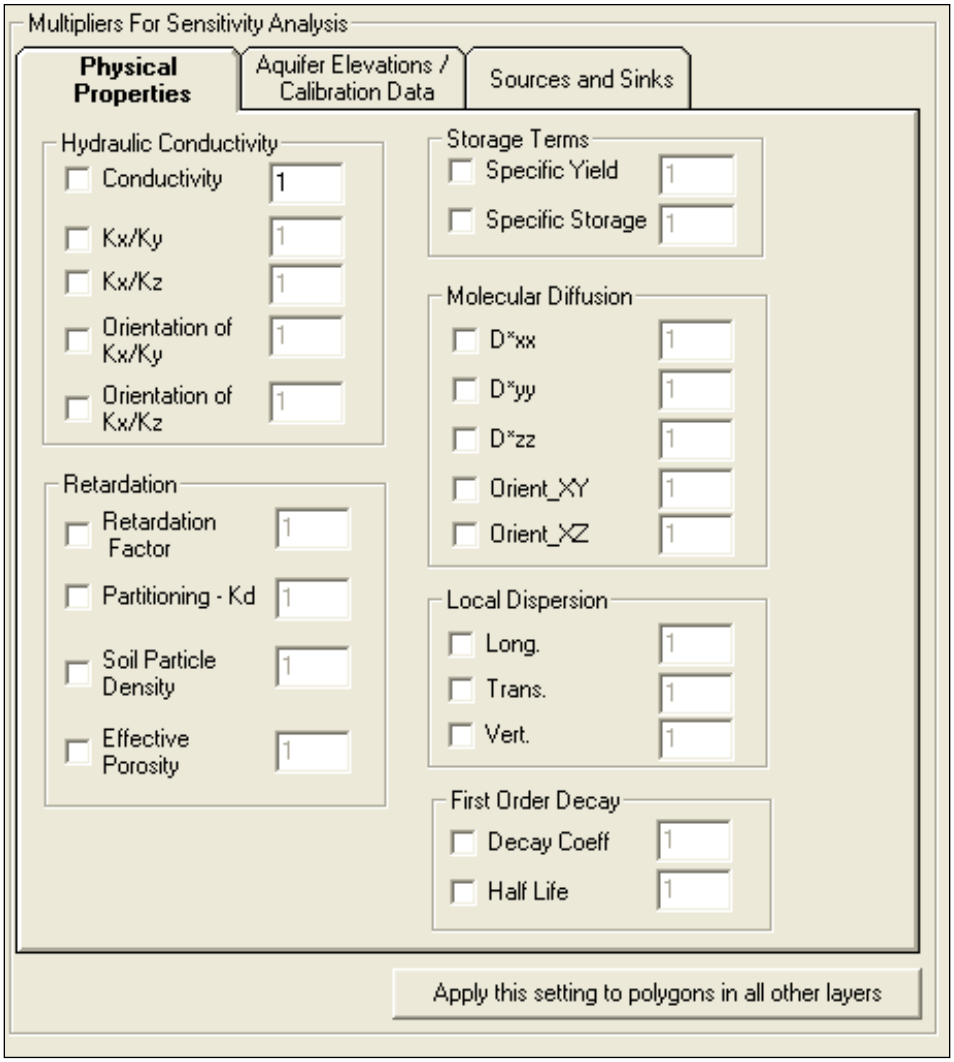


Figure 4-7 Multipliers for Sensitivity Analysis for Physical Properties

Aquifer Elevations/Calibration Data tab is empty in IGW Version 5.0P.

Sources and Sinks tab has the multipliers for Recharge, Source Concentration, River, Drain, Prescribed Head, General Head Dependent Leakage and Evapotranspiration. The user can enter different multipliers for different model layers or click '**Apply this setting to polygons in all other layers**' button to use the same multiplier(s) in all layers. Sources and Sinks tab is shown in **Figure 4-8.**

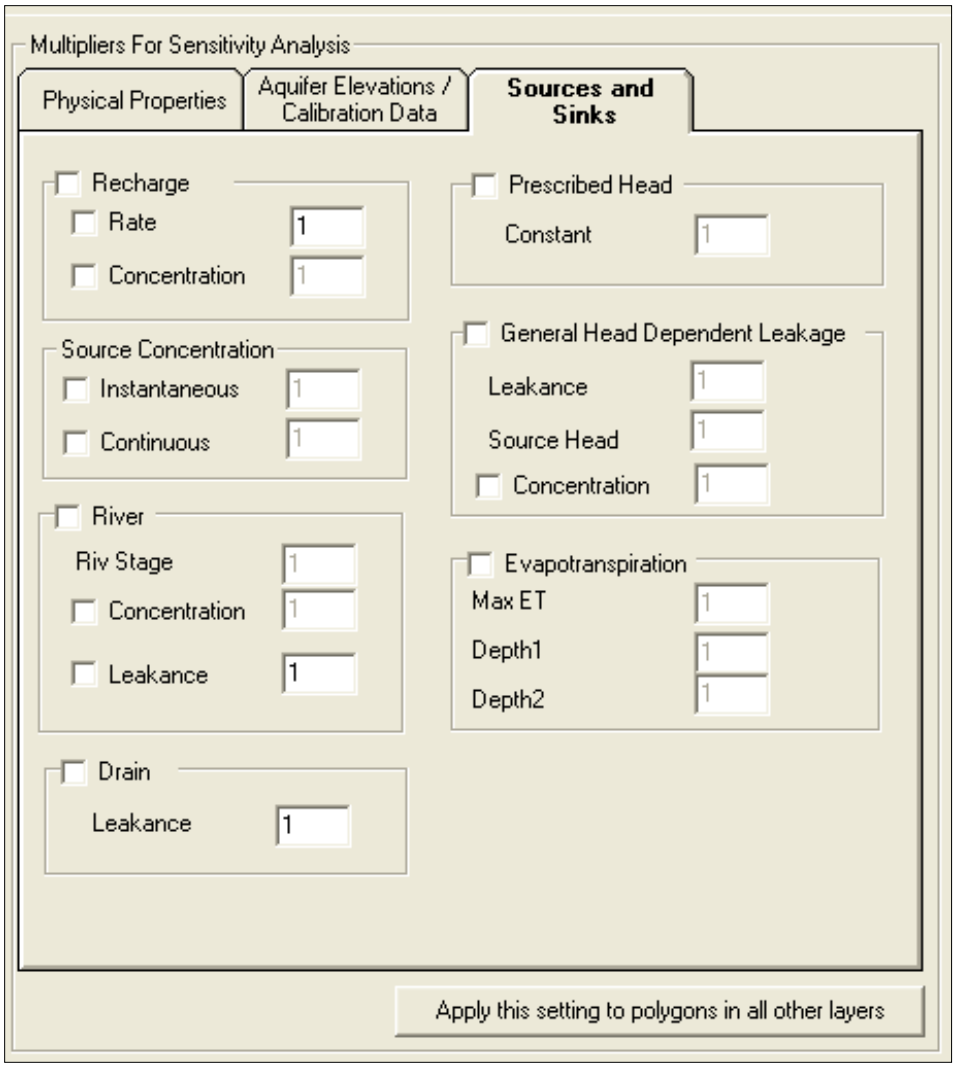


Figure 4-8 Multipliers for Sensitivity Analysis for Sources and Sinks

The individual zone entries, one level below in the LHP (i.e., Feature/Particle Group level, **Figure 4-2**) , are titled 'Zone XXXX' (where XXXX is a software-assigned number starting at 1001. RHP for this level is discussed in **Section 7.6**.

4.1.2.5. Polylines

The RHP for the 'Plines' entry on the Group level is blank as the entry acts simply as a placeholder for individual polyline entries. The individual polyline entries titled 'Pline XXXX' (where XXXX is a software-assigned number) on the Feature/Particle Group level are discussed in **Section 8.5**.

4.1.2.6. Wells

The RHP for the 'Wells' entry on the Group level is utilized in Version 4.7, in which the user has various general options such as turning off all of the wells in the group, applying a standard flow rate and/or concentration to these, and applying all values to sub-node pumping wells. The individual well entries titled 'Well XXXX' (where XXXX is a

software-assigned number) on the Feature / Particle Group level are discussed in **Section 9.4.2**.

Figure 4-9 shows the RHP view of the wells.

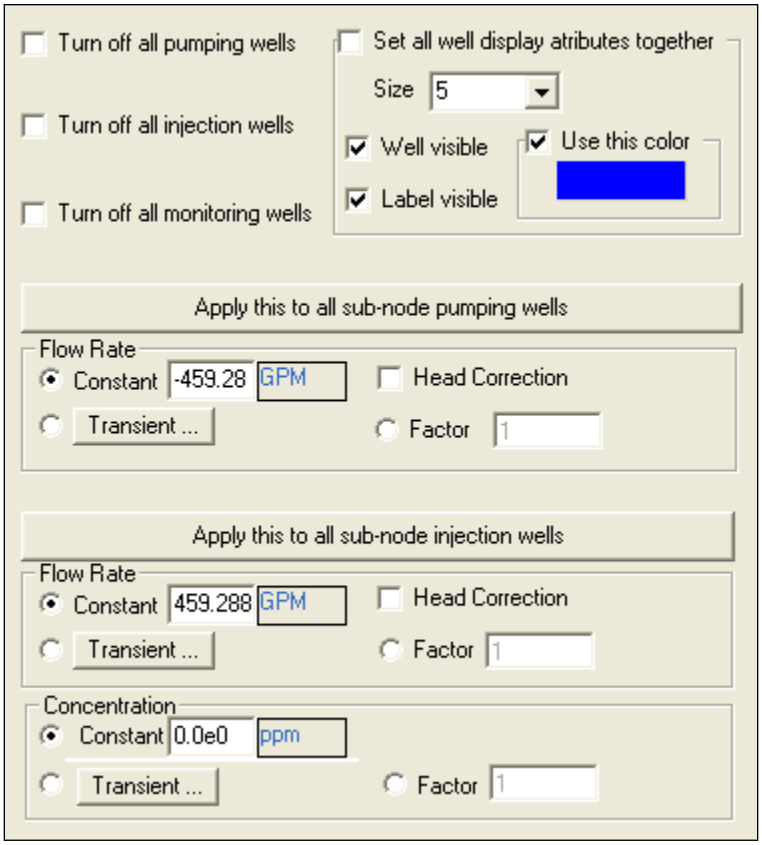


Figure 4-9 RHP View for Wells

4.1.2.7. Particles Group

The RHP for the 'ParticlesGroup' entry on the Group level is blank as the entry acts simply as a placeholder for individual particle group entries. These entries titled 'ParticlePoints', 'ParticleLines', 'ParticleWells', and 'ParticleZones' on the Feature / Particle Group level are discussed in **Section 10.2**.

4.1.2.8. Cross Sections

RHP for cross sections are explained in **Section 16.4**.

4.1.3. Time Process Selector (TPS)

The bottom left portion of the AE window is called the 'Time Process Selector' or TPS. The TPS allows the user to open a separate window for any previously defined time process (mass balance, mass flux, well head and concentration, etc.) and view the results as the model solution proceeds.

If no time processes are defined, the TPS appears blank in the AE window.

Following monitoring processes can be defined in the RHP of the AE window.

- Inside Zones: Mass balance for water and solute(s)

- At monitoring wells: Solute concentrations and heads
- Across Polylines: Mass fluxes for water and solute(s) [currently only functional for stochastic realizations]

In case of stochastic modeling, probability distributions for realizations can also be monitored for each of the above mention time processes. The display can show every single realization superimposed on the mean of all previous Monte Carlo simulations. The software also calculates probability distribution and other statistical parameters and the user can choose to display these in an interactive manner. More of this is explained in stochastic modeling (see **Chapter 18**).

When the model is discretized (see **Chapter 12**), all defined time processes appear in the TPS as shown in **Figure 4-10**. To view any desired time process, or probability plots, the user can check the box to the left of the process and the required time process/probability plot appears in a separate window. All monitoring processes are viewed in their own window. The user can open any number of time processor windows simultaneously.

The user can adjust the size of TPS (or LHP) by dragging the horizontal bar separating TPS and LHP. In **Figure 4-10**, the TPS area in the AE is expanded to show all processes in a given model.

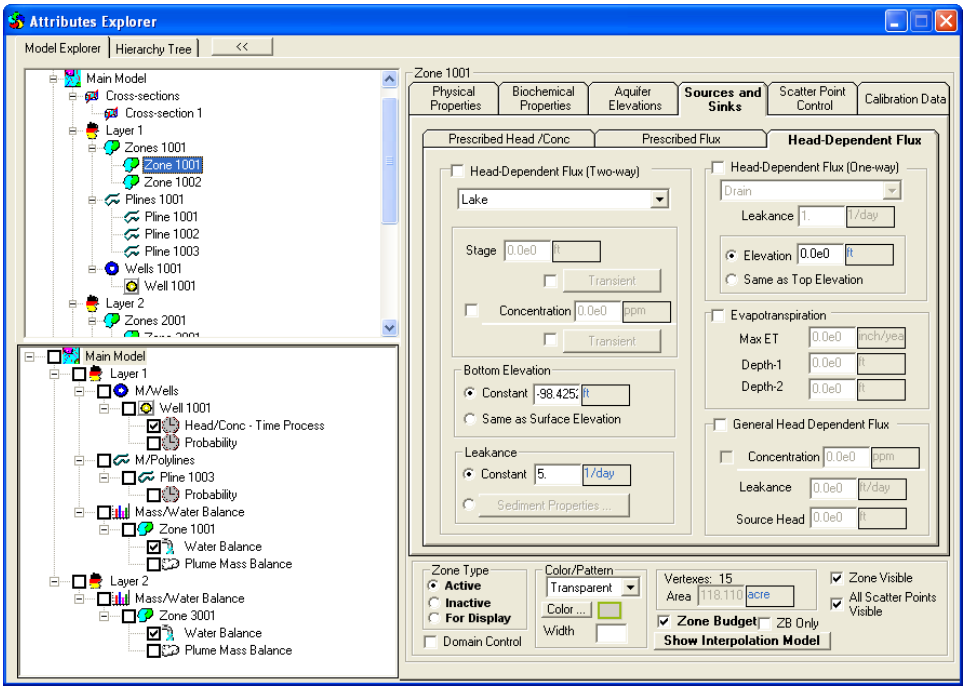


Figure 4-10 TPS showing different time processes

Information about defining time processes is given in **Section 7.6.2.4** for zone mass balances, in **Section 9.4** (see the '**Well Type Area**' subsection) for monitoring well implementation, and in **Section 8.5.4** for polylines.

All submodels created in the main model (**Chapter 15**) also show up in the TPS area. By checking the box next to a submodel will open that model in a separate window (see **Section 15.9**).

4.2. The Hierarchy Tree Layer

The **Hierarchy Tree** layer of the **AE** is currently not utilized in **IGW Version 5.0P**.

Chapter 5 BASEMAP

The basemap[9] feature allows the user to import a picture to use as a background image in the Working Area. The following sections explain the implementation and functionality of this feature.

5.1. The Model Scale and Basemap Window

The 'Model Scale and Basemap' window is the interface used to select a basemap file and set its associated real-world characteristics (scale, origin coordinates, etc). It is pictured in **Figure 5-1**.

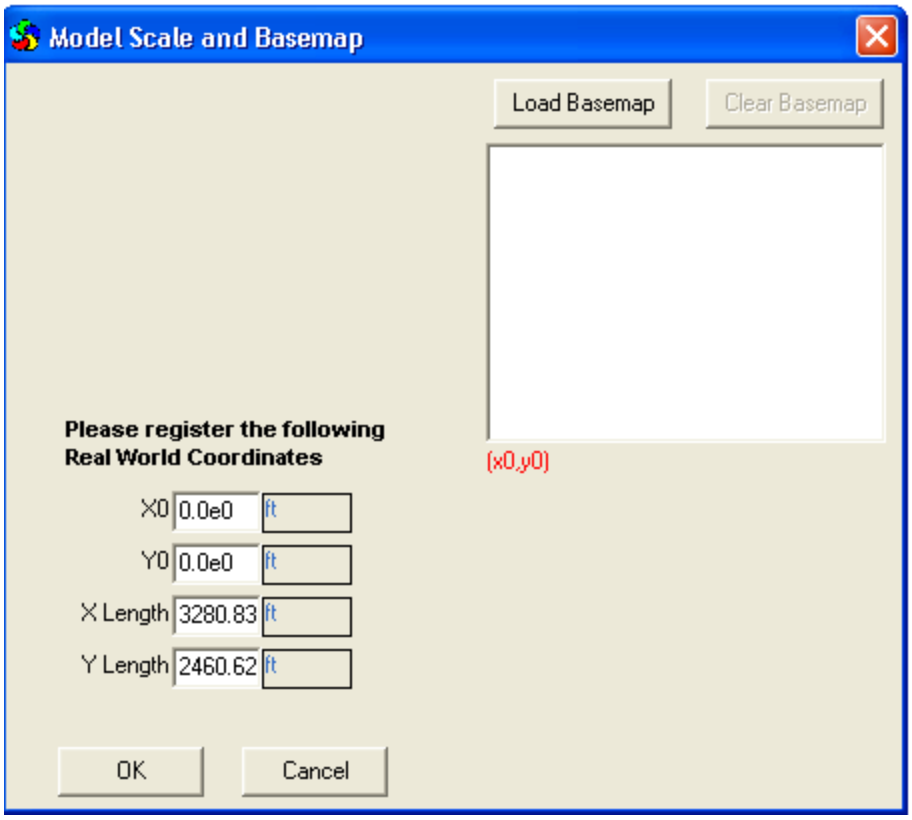


Figure 5-1 Model Scale and Basemap window

This window can be accessed by clicking the 'Set Basemap and Register a Basemap' button on the Button Palette (row 1, column 4), or by clicking the 'Import Basemap and Define Model Domain' button that appears in the 'Main Model' layer of the AE.

The white rectangular box displays a preview of the Working Area. The red coordinates indicate the location of the X0, Y0 point that can be defined by the user. (These coordinates only appear prior to a basemap being loaded, and for a preview in which the origins of both the basemap [physical origin] and Working Area coincide – even if the origin coordinates of the Working Area are modified.)

[9] Refer to **Section 3.1** of the ***IGW Version 5.0P Tutorials*** document for a step-by-step example of importing a basemap.

The section delineated by the 'Please Register the Following Real World Coordinates' heading is used to enter the origin coordinates (as referred to above) and to set the X and Y scale of the picture. **Table 5-1** defines these four variables.

Table 5-1 Definition of Real World Coordinate Parameters

VARIABLE	DEFINITION	DEFAULT VALUES	UNITS (Bold = Default)
X0	The x-direction coordinate value of the displayed Working Area origin.	0	m, cm, **ft**, inch
Y0	The y-direction coordinate value of the displayed Working Area origin.	0	m, cm, **ft**, inch
X Length	The distance represented by the x-direction extent of the basemap image.	1000 (m)	m, cm, **ft**, inch
Y Length	The distance represented by the y-direction extent of the basemap image.	750 (m)	m, cm, **ft**, inch

The user simply selects the desired unit then enters the value in the appropriate field. These values can be defined even if no basemap picture is to be imported (the values simply associate with the Working Area as a whole). However, if a basemap is to be used, these numbers should only be set after the 'Load Basemap' procedure has been completed (see **Section 5.2**).

Any changes will take effect once the 'OK' button is clicked to close the 'Model Scale and Basemap' window.

5.2. Loading a Basemap

Clicking the 'Define Model Domain/Import Basemap' button opens the 'Model Scale and Basemap' window shown in **Figure 5-2**. Clicking the 'Load Basemap' button open the 'Open' window. The user selects the type of file to open, surfs to its location, and clicks the 'Open' button. This opens the basemap in the 'Vectorization of Raster Pictures' window (**Figure 5-3**) with the basemap visible in a preview pane. The map is vectorized (as explained in **Section 5.2.1**) and imported into the model.

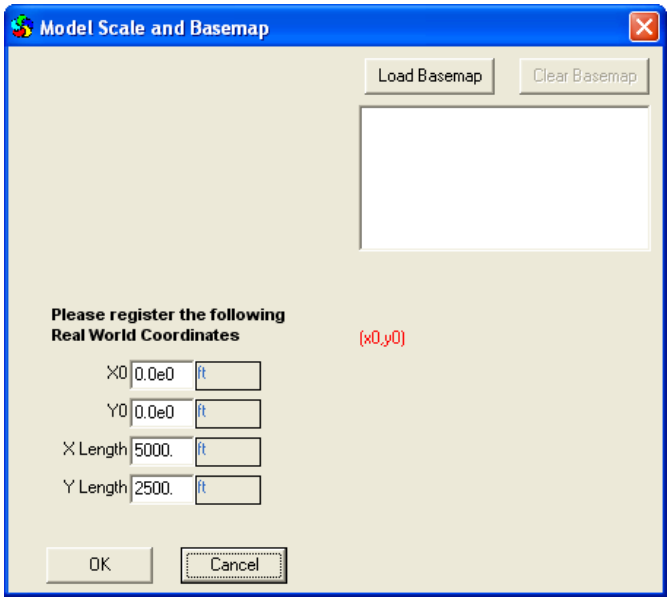


Figure 5-2 Model Scale and Basemap window

Currently, IGW Version 5.0P provides support for BMP (BitMaP), GIF (Graphic Interchange Format), JPG (Joint Progressive Experts Group), SHP (SHaPefile), and DXF (AutoCAD Drawing Interchange Format) file types. If a raster type file is selected (BMP, GIF, or JPG) it must be vectorized through the process outlined in **Section 5.2.1**. Vector type pictures do not need to be vectorized, therefore the import process is simpler. It is described in **Section 5.2.2.**

5.2.1. Vectorization of raster pictures

The vectorization process begins with the appearance of the 'Vectorization of Raster Pictures' window, as shown in **Figure 5-3**.

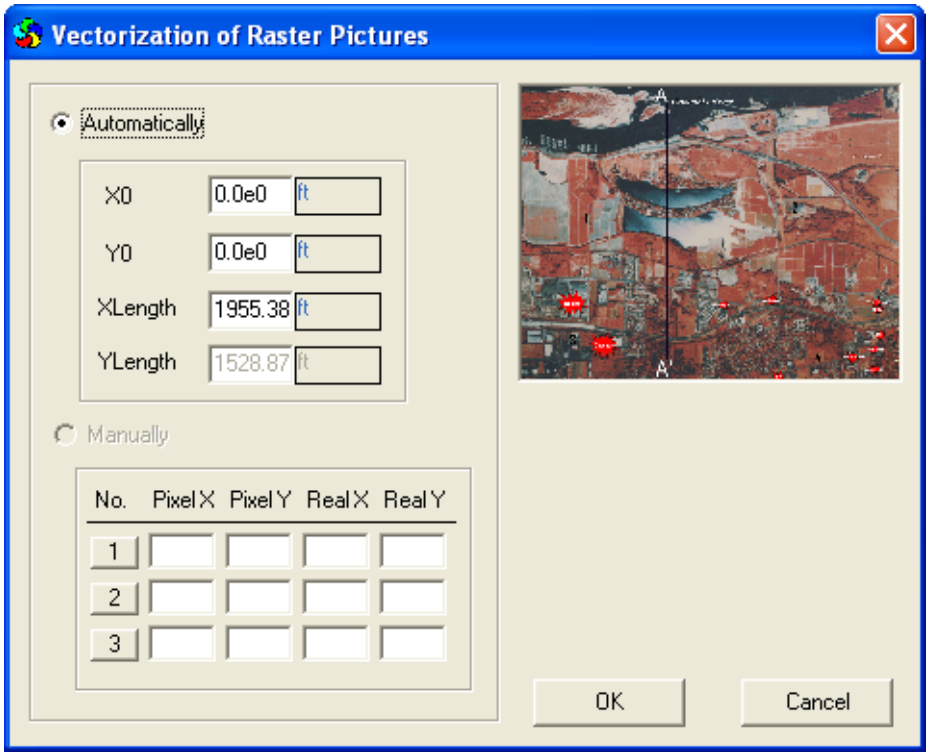


Figure 5-3 Vectorization of Raster Pictures

The window will appear with a preview of the selected picture in the white rectangular preview pane, with 'Automatically' selected by default. The X0, Y0, XLength, and YLength fields associated with the automatic vectorization process will display the software default values. The XLength and YLength fields correspond to the same variables in the 'Model Scale and Basemap' window, and shown in **Table 5-1**. The X0 and Y0 fields in this case are used to enter the Working Area coordinates of the basemap origin. At this point, the user can only specify X0, Y0, and XLength. The YLength field is calculated automatically based on the pixel dimensions of the picture.

The 'Manually' setting may be selected if the user desires more control over the vectorization process.

Once the user has entered the desired settings, clicking the 'OK' button returns the user to the 'Model Scale and Basemap' window (**Figure 5-1**). The window will have the desired picture in the preview pane, the desired coordinate and scale values entered, and additional information concerning the file location and other attributes displayed to the left of the preview pane.

> **i** The software assumes that the information contained in shapefiles (*.shp) is given with meters as the unit of measure. If this is not the case, the user must convert the file information before importing it into IGW.

The user now has a couple of options at this point. They may click the 'Load Basemap' button to combine another picture with the current one (see **Section 5.3**). The user may click the 'Clear Basemap' button to clear all basemap information and start from scratch, or the 'Cancel' button to cancel the entire basemap importing process. The user may change the assigned dimensions for the basemap (see **Section 5.2.3**). The user may also click the 'OK' button to accept the basemap configuration and end the process. Clicking the 'OK' button sets the changes in the software, closes the window, and updates the Working Area to show the desired basemap.

5.2.2. Vector Type Images

Clicking the 'Open' button in the 'Open' window (after selecting the desired vector-type file) brings the user to the 'Model Scale and Basemap' window, with the desired picture in the preview pane, the file-specified coordinate and scale values entered, and additional information concerning the file location and other attributes displayed to the left of the preview pane.

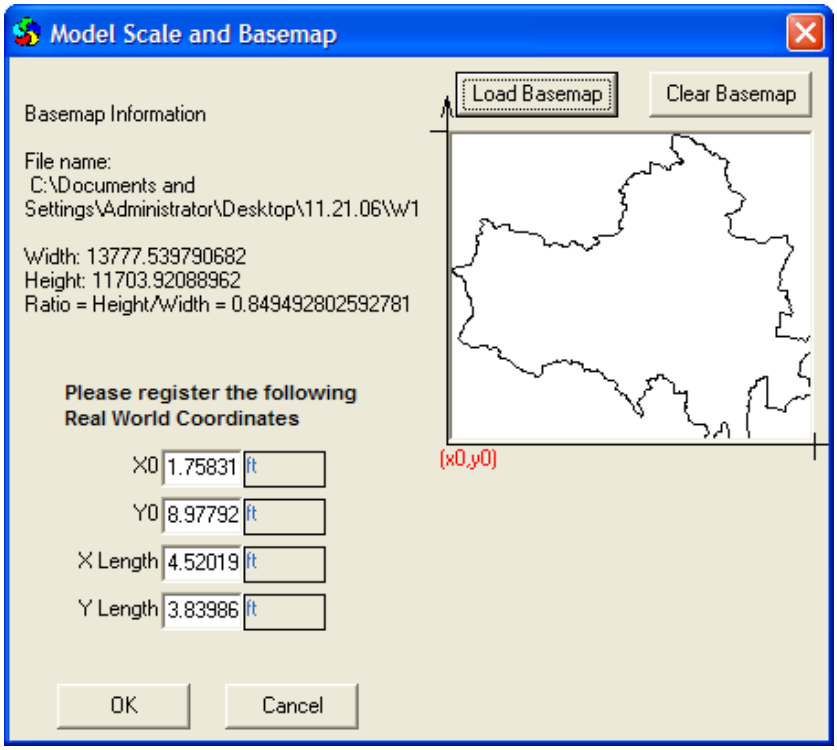


Figure 5-4 Loading a Shape File (Vector Type Image)

The user now has a couple of options to continue. They may click the 'Load Basemap' button to combine another picture with the current one (see **Section 5.3**). The user may click the 'Clear Basemap' button to clear all basemap information and start from scratch. The user may click the 'Cancel' button to cancel the entire basemap importing process. The user may change the assigned dimensions for the Working Area (see **Section 5.2.3**). Or the user may click the 'OK' button to accept the basemap configuration and end the process. Clicking the 'OK' button sets the changes in the software, closes the window, and updates the Working Area to show the desired basemap.

5.2.3. Changing Working Area Dimensions

Even if a basemap is being used, the Working Area is not locked to the size of the basemap (or combinations thereof, if multiple basemaps are used). Simply enter the desired dimensions and the 'X Length' and 'Y Length' in 'Model Scale and Basemap' window (**Figure 5-2**) and the Working Area becomes that size. If it is set larger than the basemap, the Working Area will show extra white space. If it is set smaller, the Working Area will display only a portion of the basemap.

It should be noted that the software references the images from the lower left-hand corner of the Working Area, versus the center of the screen. Therefore, any additional white space will show at the right-hand side and top of the image, and not be evenly spaced. Similarly, when displaying only a portion of the basemap, it will appear in the vicinity of the lower left-hand corner.

5.3. Loading Multiple Basemap Images

Additional basemaps may be added at any time by accessing the 'Model Scale and Basemap' window and clicking the 'Load Basemap' button (see **Section 5.2**).

The procedure is the same as described in the previous sections, however the preview pane in the 'Model Scale and Basemap' window will now show a preview of the combined images. The process may be repeated as many times as desired.

IGW Version 5.0P also allows the user to bring in multiple files of mixed formats.

! It should be noted that all combined basemap images are treated as one image in the software. Consequently, if an incorrect picture is imported or assigned the wrong scale, the entire merged basemap set will have to be cleared and the process repeated.

5.4. Clearing the Basemap

At any time, the user may open the 'Model Scale and Basemap' window (**Figure 5-2**) and click the 'Clear Basemap' button. After confirming the action (in a separate window), the preview pane shows a blank white screen. Click 'OK' to apply the change. Click 'Cancel' to close the 'Model Scale and Basemap' window and abort the clearing of the basemap (after confirming the action in a separate window).

I When a basemap is added to an IGW file, the software remembers the location of the file and opens it from that place. Therefore, if the file is opened on a different machine or the location of the basemap changes, the new location of the basemap file must be specified. This can be done by:

1) Editing the IGW file to reflect the new location of the basemap, or
2) Opening the file and browsing to the location of the file when prompted (the changed location will be updated in the IGW file).

If the user chooses to skip loading the basemap file at the prompt, the original location information for the basemap in the IGW file will be preserved.

Chapter 6 MODEL LAYER PARAMETERS

There are certain software parameters that must have numerical values at all times in order to prevent software calculation errors. Therefore, default values exist for these parameters that are assigned whenever a new zone is created. The following sections detail the parameters, the default values, and the process of changing them.

6.1. Default Model Parameters

In *IGW Version 5.0P*, the 'Default Model Parameters' window is not a separate entity as it was in IGW 3. The interface used to specify the values for the default parameters is now found in the AE, under the 'Layers' name. It is pictured in **Figure 6-1**.

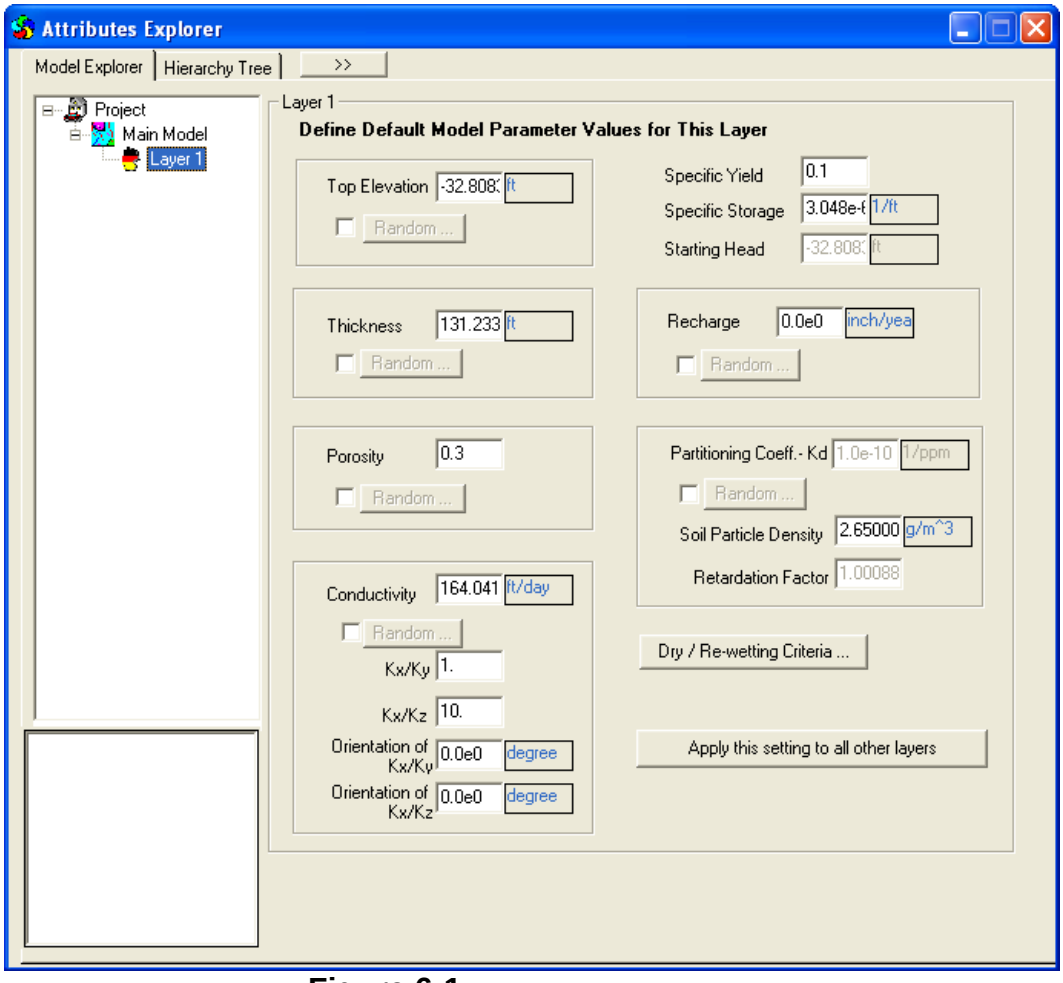


Figure 6-1 Default Model Parameters

The user simply specifies the desired values and units for the parameters of interest. The software will apply the new default values to all subsequently created zones, but only after the 'Apply this setting to all other layers' button is selected.

The individual fields and nested button functions are described in the following section.
The parameters and buttons displayed in the 'Default Attribute' window are addressed in this section. **Table 6-1** lists the parameters, their basic definitions/functions, the default values, and references to other parts of this document where they are explained in greater detail.

Table 6-1 Default Values for Attributes Explorer Parameters

PARAMETER	DEFINITION/ FUNCTION	DEFAULT VALUE	DEFAULT UNIT	REFERENCE SECTION
Top Elevation	Top elevation of aquifer	-32.808	ft	**7.6.1.3**#
Thickness	Thickness of the aquifer	-32.808	ft	**7.6.1.3**#
Specific Yield*	Specific yield of aquifer material	0.1	*dimensionless*	**7.6.1.1**#
Specific Storage*	Specific storage of aquifer material	3.048e-6	1/ft	**7.6.1.1**#
Starting Head	Initial head in aquifer	Software-assigned% / -10	ft	**7.6.1.1**#
Porosity	Effective porosity of aquifer material	0.3	*dimensionless*	**7.6.1.1**#
Random	Clicking this button activates random effective porosity field parameters, containing several options	-	-	-
Option	Clicking this button opens the 'Random Parameters' window	-	-	**B-II**
Recharge	Recharge into the aquifer	0.0e0	in/year	**7.6.1.2**#
Random	Checking this activates random recharge field	-	-	-
Option	Clicking this button opens the 'Option of Unconditional Random Field (Attr.)' window	-	-	**B-II**
Conductivity	Conductivity of the aquifer material	164.041	ft/day	**7.6.1.1**#
Random	Checking this activates random conductivity field	.	-	-
Option	Clicking this button opens the 'Option of Unconditional Random Field (Attr.)' window	.	-	**B-I**
Kx/Ky	x vs. y direction isotropy ratio	1	*dimensionless*	**7.6.1.1**#
Kx/Kz	x vs. z direction isotropy ratio	10	*dimensionless*	**7.6.1.1**#
Orientation of Kx/Ky	Orientation of anisotropy in the x-y plane	0.0e0	degrees	**7.6.1.1**#
Orientation of Kx/Kz	Orientation of anisotropy in the x-z plane	0.0e0	degrees	**7.6.1.1**#
Partitioning Coeff. – Kd	Partition coefficient of the aquifer material	1.0e-10	1/ppm	**7.6.1.1**#
Random	Checking this activates random partitioning coefficient field	-	-	-
Option	Clicking this button opens the 'Random Parameters' window	-	-	**B-II**
Soil Particle Density	The soil particle density of the aquifer material	2.65e+6	g/m^3	**7.6.1.1**#
Retardation Factor	Retardation factor	1.00088	*dimensionless*	None^
Dry / Re-wetting Criteria…	Clicking this button opens the 'Default Desaturation/Re-wetting Cell Criteria' window	-	-	**6.3**

* if unchecked, the default value is zero
^ not explicitly definable in IGW, it is a function of Kd and the bulk density (ignore the 1.00000 in the field)
# see subsection with parameter name within the listed section
% see 'Starting Head' in Section 7.6.1.1 for information details of default starting head values

6.2. Attribute Priority Protocol

IGW will assign zone attributes to every cell in which the center node lies inside of the defined zone area. However, if the node lies inside the defined zone area at two different points in the same zone – due to overlapping the boundary lines – that cell will not be assigned to those particular zone attributes.

When scatter points are defined in a zone, their explicitly defined parameters take priority over the zone attributes. For example, if a scatter point is placed in a zone and the conductivity for the

scatter point is specified, the conductivity specified for the zone no longer has any effect in the software.

If two zones overlap, the attributes of smaller zones will take precedence over larger zones.

6.3. Dry/Re-Wetting Criteria

The 'Dry/Re-wetting Criteria…' button in the Default Model Parameters window (**Figure 6-1**) opens the 'Default Desaturation/Re-wetting Cell Criteria' window shown in **Figure 6-2**.

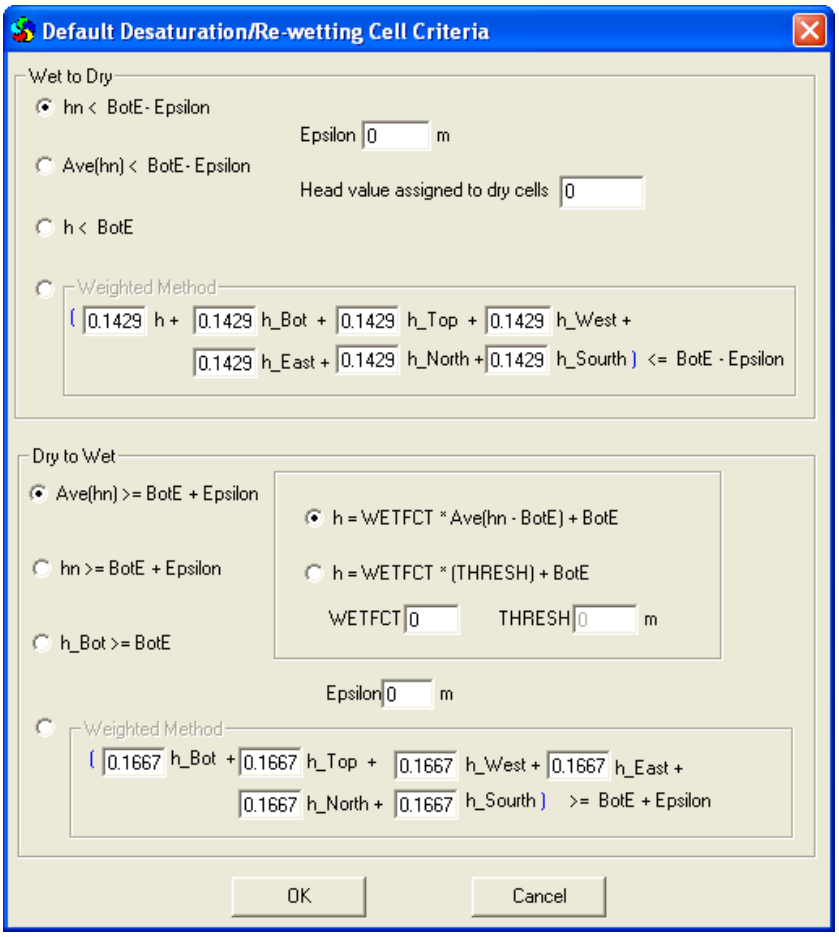


Figure 6-2 Default Options for Desaturation/Re-wetting

This interface allows the user to specify criteria for deactivating cells when they become dry, and also reactivating dry cells when appropriate.

The 'Default Desaturation/Re-Wetting Cell Criteria' window is divided into two areas that are discussed in the following subsections.

6.3.1. Making Wet Cells Dry

As the name implies, the user specifies the criteria for making cells dry (and hence inactive) in this area.

There are four choices:

1) hn < BotE – Epsilon
2) Ave(hn) < BotE – Epsilon
3) h < BotE
4) Weighted Method

The first reads 'one head value in the surrounding cells (hn) is less than the bottom elevation of the aquifer (BotE) minus epsilon (Epsilon)'.

The second reads 'the average of the head values in the surrounding cells (Ave(hn)) is less than the bottom elevation of the aquifer (BotE) minus epsilon (Epsilon)'.

The third reads 'the head in the cell (h) is less than the bottom elevation of the aquifer (BotE)'. This is the default selection.

The fourth is an expansion of the second, allowing different values for h in the weighted equation.

The user can specify a value for epsilon in the 'Epsilon' field (it must be in meters). This epsilon is independent of the epsilon in the 'Dry to Wet' area of this window. The default value is 0 meters.

The other field dubbed 'Head Value Assigned To Dry Cells' is used to enter a numerical value that the software assigns to the head values in dry cells to flag them for the user (the software internally assigns a zero value for transimissivity). The default value is 0 (with meters as the unit of measure). This value is not considered in the model solution as the software recognizes these cells to be temporarily inactive (and hence not involved in the solution because they are dry).

If simulations are run that contain heads in the 0 meter (or specified value) range, the model will function properly. It will appear that the dry cells actually have head values (due to the software assigning 0 or the specified value to them), and therefore the user may interpret the dry cells to be wet and have the displayed head value. To avoid this kind of confusion, either simply change this number to a value that will not be encountered during the simulation, or examine the 'Cell' and 'IBound' values (**Appendix A-II** and **Appendix A-III**, respectively) that are displayed in the CAT (**Section 3.6**) to determine the actual state of the cell in question.

6.3.2. Making Dry Cells Wet

The user specifies the re-wetting criteria in this area.

The user has four options concerning the re-wetting of cells:

1) Ave(hn) >= BotE + Epsilon
2) hn >= BotE + Epsilon
3) h_bot >= BotE
4) Weighted Method

The first reads 'the average of the head values in the surrounding cells (Ave(hn)) is greater than or equal to the bottom elevation of the aquifer (BotE) plus epsilon (Epsilon)'.

The second reads 'one head value in the surrounding cells (hn) is greater than or equal to the bottom elevation of the aquifer (BotE) plus epsilon (Epsilon)'. This is the default selection.

The third reads 'the head in the cell (h) is less than the bottom elevation of the aquifer (BotE)'.

The fourth is again an expansion of the second, allowing different values for h in the weighted equation.

The user can specify a value for epsilon in the 'Epsilon' field (it must be in meters). This epsilon is independent of the epsilon in the 'Wet to Dry' area of this window. The default value is 0.2 meters.

The user can also specify what value the head should take when re-wetting occurs. Again there are two options:

1) h = WETFCT * Ave(hn-BotE) + BotE
2) h = WETFCT * (THRESH) + BotE

The first reads 'the head in the cell (h) equals the wetting factor (WETFCT) times the average of the differences of the surrounding cell head values and the bottom elevation of the aquifer (Ave(hn-BotE)) all plus the bottom elevation of the aquifer (BotE)'. This is the default selection.

The second reads 'the head in the cell (h) equals the wetting factor (WETFCT) times the threshold value (THRESH) all plus the bottom elevation of the aquifer (BotE)'.

The wetting factor can be specified in the WETFCT field (it is dimensionless). The default value is 0.5.

The threshold can be specified in the THRESH field (it must be in meters). This field becomes active only after the second head specification option is selected. The default value is 0.2 meters.

Chapter 7 ZONES

Zones are used to define areas in the Working Area that represent specific features or areas of interest. The following sections describe the implementation and functionality of zone features.

7.1. Defining Zones

The first step in defining a zone[10] is clicking the **Create Zone/Assign Property** button (Button Palette-row 2, column 2). This puts the cursor in drawing mode. The user may now define a zone in the Working Area by simply clicking the mouse at points around the edge of the desired area (as explained below).

First, the user selects a beginning point (arbitrary) somewhere on the edge of the zone and clicks the mouse.

Next, the user moves the mouse cursor to another point on the edge of the area. A line will extend from the initial point to the current location of the cursor indicating the 'proposed' edge of the zone. The user should adjust the location of the mouse cursor to make the 'proposed' edge coincide with the user's desired edge. Once this is done, clicking the mouse sets the point.

Next, the user should move the mouse cursor to another point on the edge of the area. One line will extend from the last defined point. Again, the user should adjust the location of the mouse cursor to make the line from the last defined point coincide with the desired edge of the zone. Clicking the mouse will set the point. Another line will extend from the initial point indicating the 'proposed' polygon shape. The edge defining process should be repeated until the desired zone shape has been achieved.

Double-clicking the mouse ends the process and sets the zone in the software.

> **i** A rectangle is easily defined by holding down the **[SHIFT]** key before clicking the mouse on the first point, and then simply moving the cursor to the opposite end of the rectangle and clicking the mouse again.

Alternately, the user may type in the coordinates for each vertex instead of clicking the mouse at the desired location using VCI (see **Section 3.8**). This method is not limited by the resolution of the screen/mouse relationship and therefore allows for a more precise development of zone features.

When a zone is defined in the software it becomes an active feature.

At this point, the cursor is still in 'draw mode' and the user may continue to add more zones as desired.

A zone drawn in the model area is shown in **Figure 7-1**.

[10] Refer to **Section 3.42** of the ***IGW Version 5.0P Tutorials*** document for step-by-step examples of defining a zone.

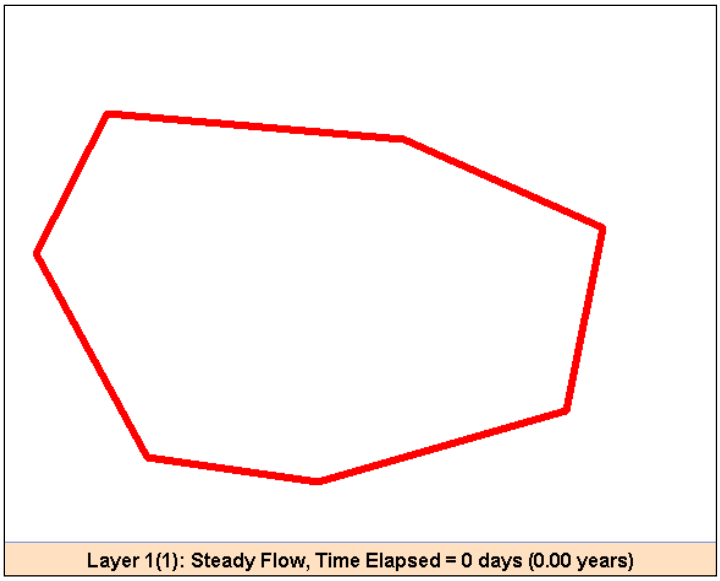


Figure 7-1 An Example Zone in the Working Area

R A good first place to start in building a model is to define the zone over which the solution will apply. The user can define such a zone as 'Domain Control Zone' by checking the appropriate box in the RHP of AE when the zone is 'selected'. 'Domain Control Zone' check box is located at the bottom right corner of the RHP (see **Figure 7-3**)

7.2. Selecting Zones

To select a zone in the **Working Area,** first click the **Select a Zone** button (Button Palette-row 3, column 2) and then click the cursor within the desired zone. The zone becomes outlined in red, therefore indicating that is currently selected.

In the case of zone overlapping, the software will select the zone that is taking precedence (according to the software rules – see **Section 6.2**) at the specific location the mouse was clicked.

Alternately, the desired zone may be selected in the AE (see **Section 4.1.1**).

7.3. Redefining Zones

A zone that has been defined in the model can be redefined by placing the cursor in 'Node Edit' mode (see **Section 3.16** for instructions on placing the cursor in this mode; also see "*set probe sensitivity for node edit"* in **Section 3.3.5** to see how Node Edit is used). As soon as you are in mode edit mode, the selected zone appears as shown in **Figure 7-2**. The black squares represent existing nodes and the blue crosshair between the existing nodes represent additional nodes created by the software. The user may change the shape of the zone in the Working Area either by:

1) Moving the existing vertices; and/or
2) Creating new vertices.

To move an existing vertex, click and hold the mouse above the black square that corresponds to the desired vertex endpoint, drag the cursor to the desired location and release the mouse button.

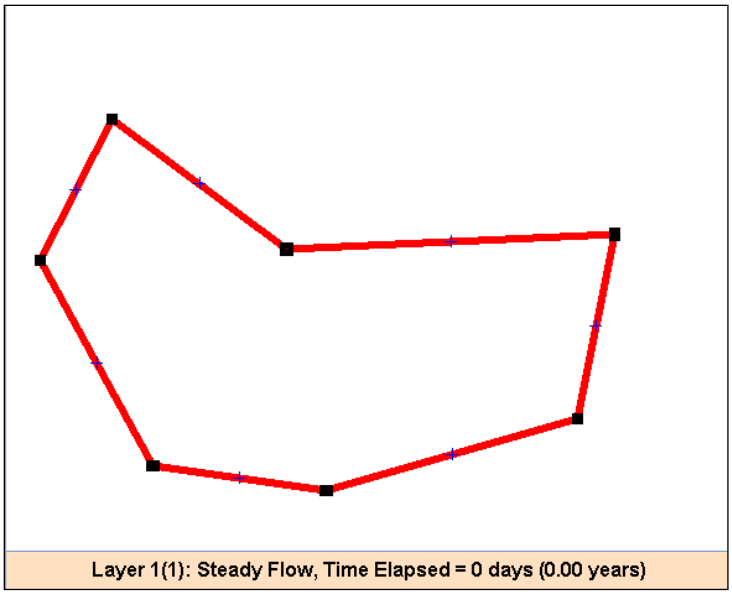


Figure 7-2 Zone with nodal points, denoted as blue crosses

To create a new vertex, click and hold the mouse above the blue crosshair symbol nearest to the desired location of the new vertex. Drag the cursor to the desired vertex location and release the mouse button.

These steps may be repeated as many times as necessary until the desired zone shape is achieved.

If the newly desired zone shape is quite different from the existing zone shape it may be more efficient to simply replace the zone. This process is described in **Section 7.5**.

7.4. Moving Zones

To move zones in the Working Area, hold down the **[CTRL]** key while clicking and holding the cursor down on top of a selected zone. Then, drag the zone into the desired position simply by moving the mouse to that position, and releasing the cursor.

7.5. Replacing Zones

Sometimes the user may want to replace an existing zone with a new one, without losing the associated attributes and scatter points.

To do this, first select the zone to be replaced (same methodology as **Section 7.2**). Next, click the **Modify Existing Zone** button and verify the request in the warning window that appears.

Draw the new shape for the zone (same methodology as **Section 7.1**).

When finished, the old zone will disappear and the new zone will have all the associations of the old one (it will also become the active feature).

This is similar to redefining the zone (**Section 7.3**), except this method replaces the zone in one step instead of using incremental adjustments. Therefore, this replacement method is most useful if the newly desired zone is quite different in shape than the one to be replaced. If only a minor adjustment to the zone shape is required, it would be simpler to use the method presented in **Section 7.3**.

7.6. Setting Zone Attributes

Zone attributes[11] are set in the AE (see **Section** Error! Reference source not found.). After ccessing the AE, the first step is to select the desired zone in the LHP, i.e. Zone 1001 (see **Section 4.1.1**). Doing this brings up the 'Zone' RHP (see **Section 4.1.2**). A sample of the RHP for 'Zone 101' is shown in **Figure 7-3**.

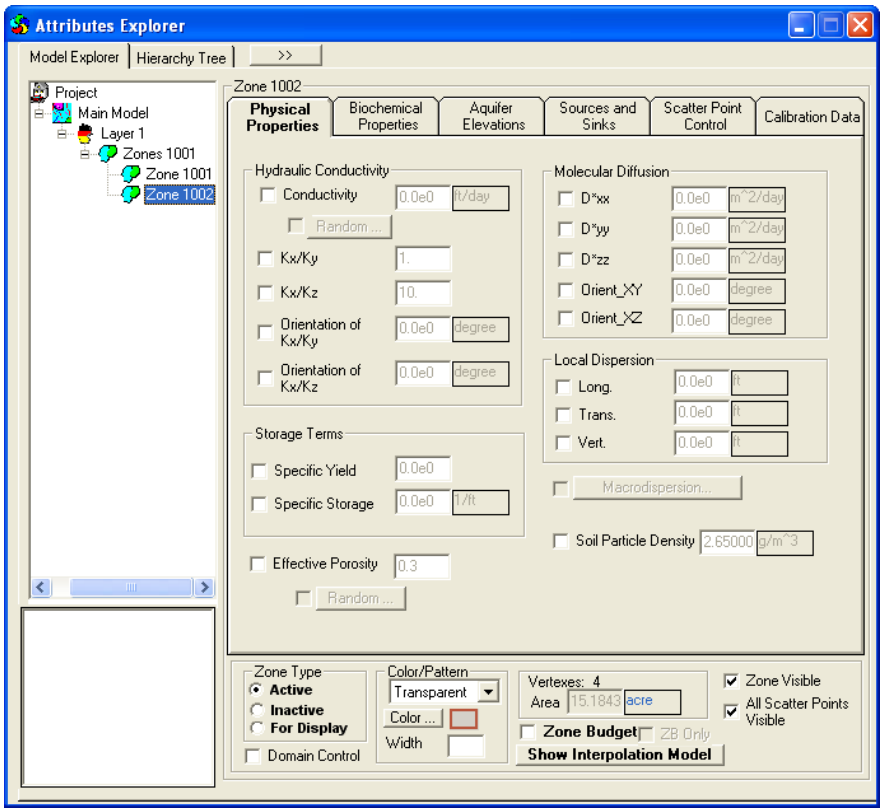


Figure 7-3 RHP for a Model Zone

The **Zone RHP** is divided into an upper and lower section. The upper section is referred to as the **Attribute Entry Area (AEA)**. The lower is referred to as the **Visualization / Option Area (VOA)**. The following subsections discuss them further.

7.6.1. Attribute Entry Area (AEA)

The **AEA** is divided into 6 layers:

1.) **Physical Properties** layer,
2.) **Biochemical Properties** layer,
3.) **Aquifer Elevations** layer,
4.) **Sources and Sinks** layer,
5.) **Scatterpoint Control** layer, and the
6.) **Calibration Data** layer.

[11] Refer to **Section 3.2 , Section 3.53** and **Section 3.4** of the ***IGW Version 5.0P Tutorials*** document for examples of defining zone attributes.

These layers allow for the input of the type of data inferred by their title. They are discussed in the following subsections.

7.6.1.1. Physical Properties

This layer is where the physical properties of the aquifer material are defined. The layer can be seen highlighted with blue background in LHP as shown in **Figure 7-3.**

The individual parameters are discussed in the following subsections.

HYDRAULIC CONDUCTIVITY

Also referred to as the Coefficient of Permeability, the hydraulic conductivity, K, is a measure of groundwater's ability to move through porous media. It has dimensions of [L/T].

Checking the box next to **Conductivity** allows the user to specify the conductivity value in the appropriate field. The user may check the box next to the deactivated **Random** button to activate the random K distribution (with default settings). Clicking the **Random** button opens the **Option of Unconditional Random Field** window (**Appendix B-I**), therefore allowing the user to change the random distribution settings.

The default conductivity value is 164.041 ft/day (set in the **Default Attribute** window , **Chapter 6**)**.** The default entry is 0.0e0. The default unit is m/day with cm/sec and ft/day also available.

KX/KY

The K_x / K_y value quantifies the isotropic character of the aquifer material in the form of a ratio. This ratio relates the conductivity in the x-direction to the conductivity in the y-direction.

The default value is 1 (set in the **Default Attribute** window, **Chapter 6**), the default entry is 1, and the ratio is dimensionless.

KX/KZ

The K_x / K_z value quantifies the isotropic character of the aquifer material in the form of a ratio. This ratio relates the conductivity in the x-direction to the conductivity in the z-direction.

The default value is 10 (set in the **Default Attribute** window, **Chapter 6**), the default entry is 1, and the ratio is dimensionless.

ORIENTATION OF ANISOTROPY IN XY

This parameter is the angle between the x-axis and the Kx axis (in the x-y plane). Refer to **Figure 7-4** for a visual aid.

The default value is 0.0e0, and default entry is 0.0e0. The default unit is degrees, with radians also available.

ORIENTATION OF ANISOTROPY IN XZ

This parameter is the angle between the z-axis and the Kz axis (in the x-z plane). This value has no effect in IGW Version 5.0P calculations as the software incorporates only one layer.

The default value is 0.0e0, and default entry is 0.0e0. The default unit is degrees, with radians also available.

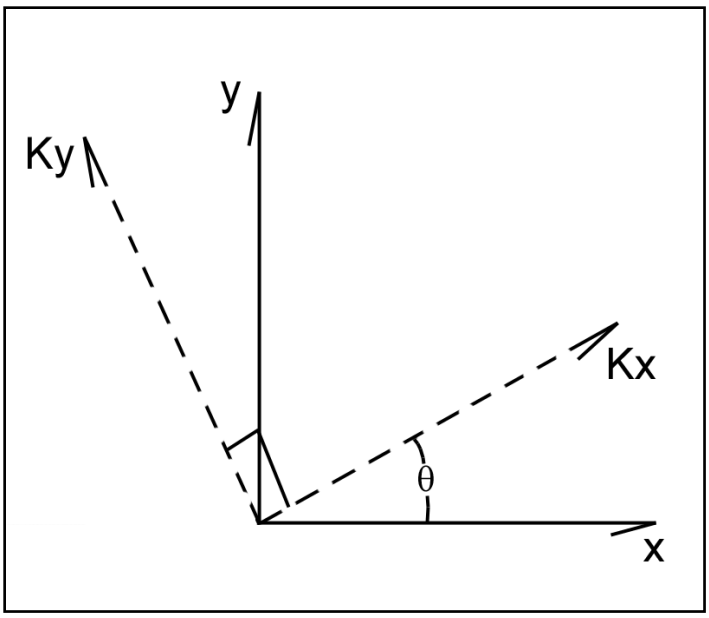


Figure 7-4 Orientation of Anisotropy

SPECIFIC YIELD

This is the storage term for unconfined aquifers. Denoted S_y, it is defined as the volume of water that an unconfined aquifer releases from storage, per unit surface area of aquifer, per unit decline in the water table (see **Figure 7-5**).

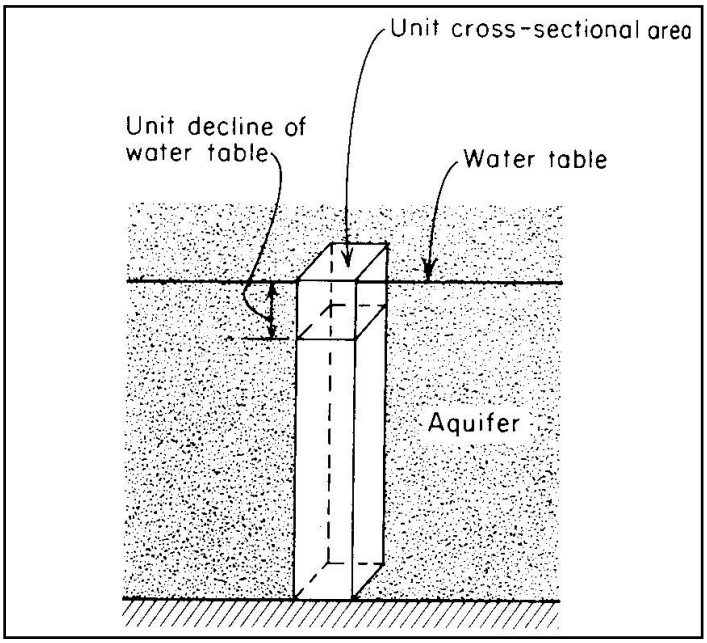


Figure 7-5 Parameters Defining Specific Yield

The default value in layer window is 0.1. The default entry for a model polygon is zero, and the parameter is dimensionless.

SPECIFIC STORAGE

The specific storage, S_s, is the amount of water per unit volume of saturated aquifer material that can be stored or released, based on the water and aquifer material

compressibility per unit change in head. S_s has dimensions of [L^{-1}] and is defined mathematically as:

$$S_s = \gamma(\alpha + \eta_e \beta) \qquad \textbf{7.6.1.1.2}$$

where:

γ	=	density of water [ML^{-3}]
α	=	compressibility of aquifer skeleton [LT^2M^{-1}]
β	=	effective porosity [1]
η_e	=	compressibility of water [LT^2M^{-1}] $(4.6\times10^{-6} m^2/N)$

Selecting the box next to **Specific Storage** allows the user to specify a value.

The default value in the layer window is 3.048e-6 1/ft.. The default entry for a model polygon is zero. The available units of measure are 1/m, 1/cm, 1/ft, and 1/inch.

EFFECTIVE POROSITY

The effective porosity, η_e, is the ratio of the actual pore space within the aquifer materials, in which water can travel in x, y, and z directions. The value of effective porosity becomes extremely important when modeling contaminant transport. As clay material has a high porosity or ability to store water, the effective porosity is orders of magnitude lower since the water is essentially 'trapped' within the pore space, unable to migrate. The volumetric relationship of effective porosity is the ratio of the volume of voids to the total volume of material.
The default random (η_e) distribution is activated by clicking the box next to the deactivated **Random** button. Clicking the random button allows the user to change the settings for the random n_e distribution by opening the **Random Parameters** window (see **Appendix B-II**). Selecting the box next to **Effective Porosity** allows the user to specify the n_e value and specify any desired randomness in the n_e field.

The default value (set in the **Default Attribute** window, see **Chapter 6**) is 0.3. The default entry for a model polygon 0.0e0. The porosity is dimensionless.

LOCAL DISPERSION[12]

Dispersivity is a measure of generalized variance in flow direction due to mechanical dispersion (mixing) within the soil matrix. It is defined as a unit of length.

Selecting the box next to **Local Dispersivity** allows the user to specify the longitudinal (Long.), transverse (Trans.), and vertical (Vert.) dispersivity values.

The default value and entry for each is 0.0e0. The default unit is feet (ft) with centimeters (cm), meters (m), and inches (inch) also available.

MACRODISPERSION

Dispersion occurring at the field scale is called macrodispersion.

SOIL PARTICLE DENSITY

The soil particle density, ρ_s, is a measure of the density of the individual soil particles. It is used, along with the effective porosity, to determine the bulk density (ρ_b).

$$\rho_b = (1 - n_e) * \rho_s \qquad \textbf{7.6.1.1.3}$$

The default value entry is $2.65x10^6$ g/m^3, as defined in the **Layer** window.

[12] Refer to **Section 13.2** of the ***IGW Version 5.0P Tutorials*** document for additional information.

7.6.1.2. Biochemical Properties

This layer is where chemical properties for the zone are defined. It is shown in **Figure 7-6**.

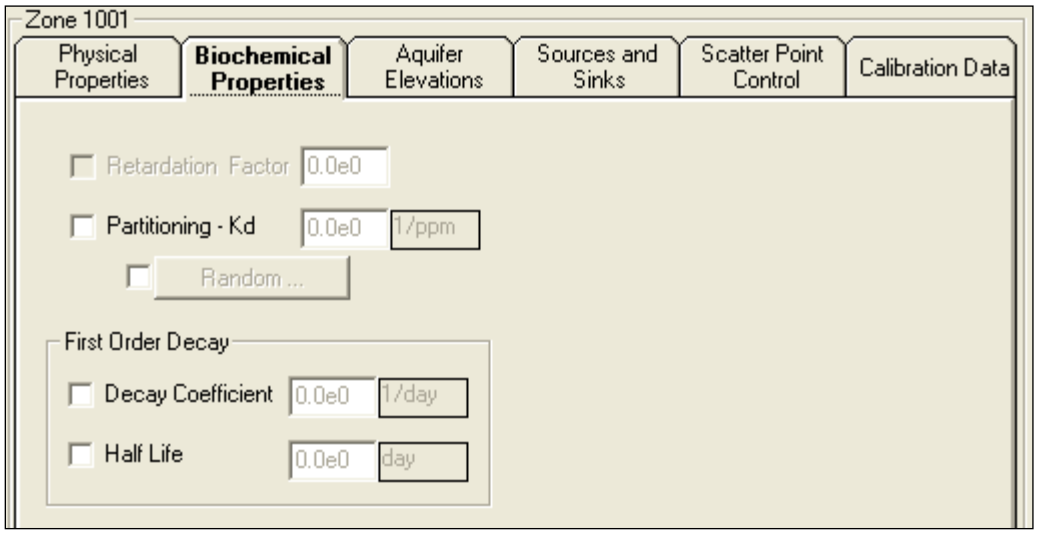


Figure 7-6 Biochemical Properties

RETARDATION FACTOR

Retardation factor is a dimensionless parameter characterizing the retarding effect of adsorption on solute transport. Mathematically, the retardation factor, *R*, is defined by **Equation 7.6.1.1.1**

$$R = 1 + \rho \frac{K_d}{\theta} \qquad \textbf{7.6.1.1.1}$$

where K_d = Partition coefficiet [$L^{-3}M$]
ρ = bulk density [ML^{-3}]
θ = volumetric water content [1]

Checking the box next to **Retardation Factor** allows the user to specify the *R* value. By default *R* = 1.

Partition coefficient is explained in detail under the next heading. Since *R* is related to partition coefficient K_d through **Equation 7.6.1.1.1**, the user can enter the value for either of the two parameters and the other one would adjust automatically.

PARTITIONING – Kd

The partition (or distribution) coefficient (Kd) describes a substance's affinity for sorbing to solid particles. It relates the amount of solute (i.e. water) sorbed on the soil to the amount of chemical that is dissolved in the water, and is defined by **Equation 7.6.1.1.2**:

$$K_d = \frac{C_s}{C_w} \qquad \textbf{7.6.1.1.2}$$

where:

C_s = concentration of substance sorbed to aquifer media [mole M^{-1}]
C_w = concentration of substance dissolved in aquifer water [mole L^{-3}]

For organic contaminants, it is also defined as the product of the *organic carbon content* (f_{oc}) and the *organic carbon / water partition coefficient* (K_{oc}).

Checking the box next to **Partitioning - Kd** allows the user to specify the Kd value and any desired randomness.

The default value (set in the **Default Attribute** window) is 0, which is equivalent to 'no sorption'. The default entry is zero. The available units of measure are m^3/g (default), 1/ppb, 1/ppm, and m^3/kg.

The default random Kd distribution is activated by clicking the box next to the deactivated **Random** button. Clicking the newly activated **Random** button allows the user to change the settings for the Random Kd distribution by opening the **Random Parameters** window (see **Appendix B-II**).

DECAY COEFFICIENT

The decay coefficient is used to describe a first-order, irreversible process of the form:

$$\frac{\partial C}{\partial t} = -\lambda C \qquad \textbf{7.6.1.1.4}$$

where C = concentration [ML^{-3}]
t = time [T]
λ = decay coefficient [T^{-1}]

HALF LIFE

The half life describes the time it takes half of the given material to decay.

7.6.1.3. Aquifer Elevations

This layer is where elevations for the zone are defined. It is shown in **Figure 7-7.** The individual parameters are discussed further in the following subsections.

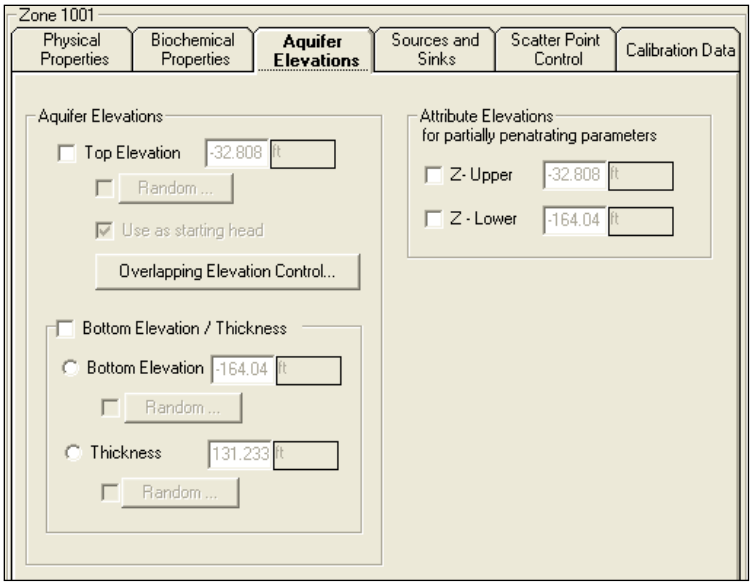


Figure 7-7 RHP for Aquifer Type / Elevations

TOP ELEVATION

Top Elevation is the elevation of the top of the aquifer material. Clicking the box next to **Top Elevation** allows the user to enter a desired value in the associated field. The default value is –32.808 feet (set in the **Layer** window). The default entry is also –32.808 feet, with available units of meters (m), centimeters (cm), feet (ft) and inches (in).

The **Random** button can also be employed to randomly assign a top elevation to the model, based on its layer parameters.

For multi-layered models, the **Overlapping Elevation Control** button will allow overlapping layers to be decided by either the current layer's top elevation, or the upper layer's bottom elevation. If there is no upper confining layer (unconfined aquifers), the user should set the top elevation equal to the surface elevation. 'Option for Elevation Overlapping/Separating' window is shown in **Figure 7-8**

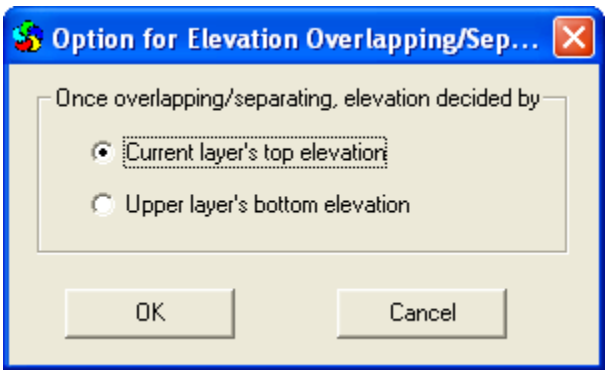


Figure 7-8 Elevation Overlapping Screen

BOTTOM ELEVATION/THICKNESS

Bottom Elevation is the elevation of the bottom of the aquifer material. Clicking the box next to **Bottom Elevation** allows the user to enter a desired value in the associated field. The default value is –164 feet (set in the **Layer** window). The default entry is also –164 feet, with available units of measure being feet (ft by default), centimeters (cm), meters (m), and inches (in).

IGW Version 5.0P allows the user the option to set a **Thickness** for each layer, versus having to always designate a top and bottom elevation to define depth.

The **Random** button can be employed to assign random thickness values in the model. See Appendix **B-II** for more information on how to use this feature.

ATTRIBUTE ELEVATIONS FOR PARTIALLY PENETRATING PARAMETERS
Top and bottom elevations of partially penetrating parameters can be entered here.

7.6.1.4. Sources and Sinks

This layer is where the water body and contamination characteristics for the zone are defined. The layer is subdivided into three separate tags:

- **Prescribed Head / Concentration**
- **Prescribed Flux**
- **Head-Dependent Flux**

7.6.1.4.1. Prescribed Head / Concentration

This layer is where the water body and contamination characteristics for a model zone are defined **(Figure 7-9**).

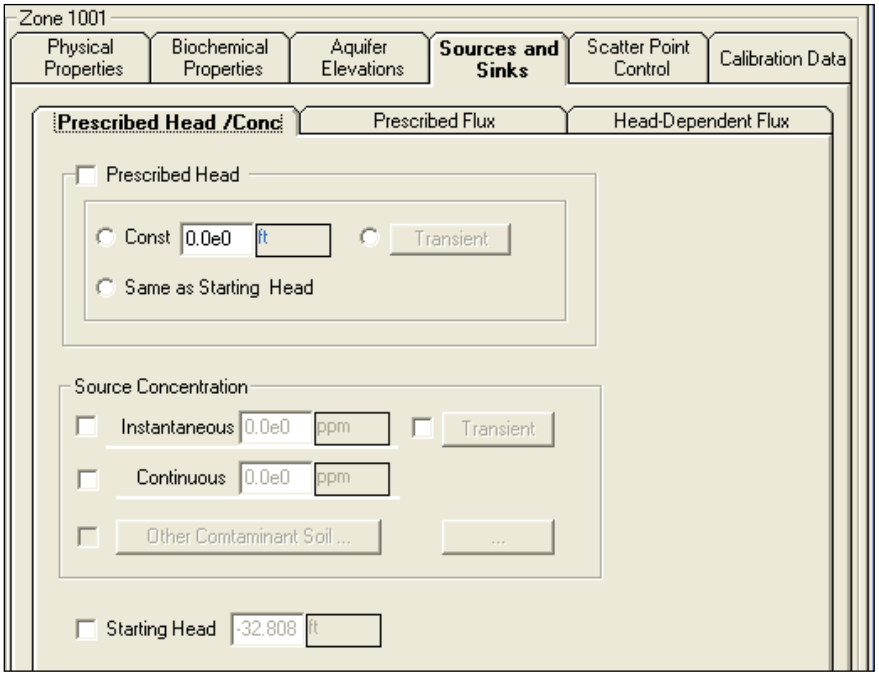


Figure 7-9 Prescribed Head / Concentration

PRESCRIBED HEAD AREA
Selecting **Constant Head**, or the **Transient** button in the **Prescribed Head** area, allows the user to set a prescribed hydraulic head for a zone.

If **Constant Head** is selected, then the user may select **Constant Value** and enter a value in the field. Zero is default value; default units are meters (m), with feet (ft), centimeters (cm) and inches (inch) also available. Or, select **Same as Starting Head**

to set the constant head value to the same as that entered in the **Starting Head** field (see the **Starting Head** subsection).

If the **Transient** button is selected, then the user may activate the default transient settings for the constant head. Clicking the activated **Transient** button allows the user to edit the settings by opening the **Transient Settings** window. Please refer to **Appendix C-I** for a discussion of this window.

If **None** is chosen (default), then there is no constant head setting for the zone.

SOURCE CONCENTRATION AREA[13]

In the **Source Concentration** area, the user can select one of two options to define the zone as a contaminant area.

Checking the box next to **Instantaneous Concentration** defines the model zone as having an instantaneous concentration with the desired concentration entered in the associated field (default is 0.0e0) and available units of ppm (default), ppb, g/m^3, and kg/m^3.

Checking the box next to **Continuous** defines the zone as having a constant concentration with the desired concentration entered in the associated field (default 0.0e0) and available units of ppm (default), ppb, g/m^3, and kg/m^3.

These two are mutually exclusive, therefore only one can be selected for a single zone.

Clicking on  button allows the user to define random distribution properties of source concentration for both instantaneous and continous sources. This buttons opens the 'Concentration Distribution Model' window (**Figure 7-10**) where user can define the Gaussian properties of the source. The default setting in this window is 'Constant'.

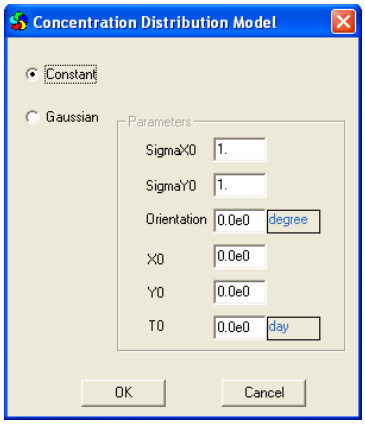


Figure 7-10 Concentration Distribution Model window

A concentration plume will appear in the zone after the model is discretized (see **Chapter 12**).

The solver settings for contaminant plumes are presented in **Section 13.3**.

'**Other Contaminant Soil …**" button is not active in IGW Version 5.0P.

STARTING HEAD

[13] Refer to **Section 14.1** of the ***IGW Version 5.0P Tutorials*** document for an example of defining a zone as a concentration source.

Checking this box allows the user to edit the initial head value that the software will use in its solution calculations. If the box is not checked, either here or in the **Layer** window, then the software will assign the starting head to be:

1) Constant head equal to the specified head of the feature where a prescribed head feature is defined, or
2) The top elevation of the aquifer where no prescribed head feature is defined.

The default entry in the field is –32.808 (this value becomes active and can be changed if the box is checked). The default unit of measure is feet (ft), with meters (m), centimeters (cm), and inches (in) also available.

7.6.1.4.2. Prescribed Flux

This layer is where the water body and contamination characteristics for the zone are defined (**Figure 7-11**)

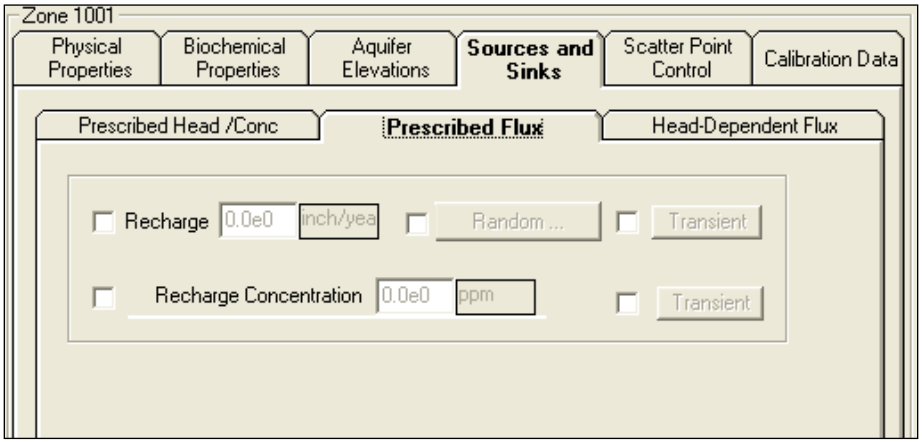


Figure 7-11 Prescribed Flux

RECHARGE

Checking the box next to **Recharge** defines the zone as one having a constant level of recharge. The user can subsequently enter the desired value in the appropriate field (default entry is 0.0e0) and select the desired unit (m/day is default; cm/day, cm/year, ft/year, and inch/year are also available).

RANDOM BUTTON

Checking the box next to **Random** allows the user to assign arbitrary random values for a given parameter in their model. The user can apply this feature only to recharge in the **Prescribed Flux** region.

TRANSIENT BUTTON

Checking the box next to the **Transient** button sets the zone as one having a transient level of recharge. The user may edit the transient recharge settings by clicking the activated Transient button and therefore opening the **Transient Settings** window. Please refer to **Appendix C-I** for a discussion of this window. Note that the default values discussed in the appendix are different than those applicable to this section.

RECHARGE CONCENTRATION

Checking the box next to this feature allows a stated level of contamination to be injected into the model at a given layer. This feature can be used to simulate contaminant transport in flow regimes within a model.

7.6.1.4.3. Head Dependent Flux

This layer is where the water body and contamination characteristics for the zone are defined (**Figure 7-12**). This layer has the following four areas:

- Head-Dependent Flux (Two-Way)
- Head-Dependent Flux (One-Way)
- Evapotranspiration
- General Head Dependent Flux

Each of these areas is explained below.

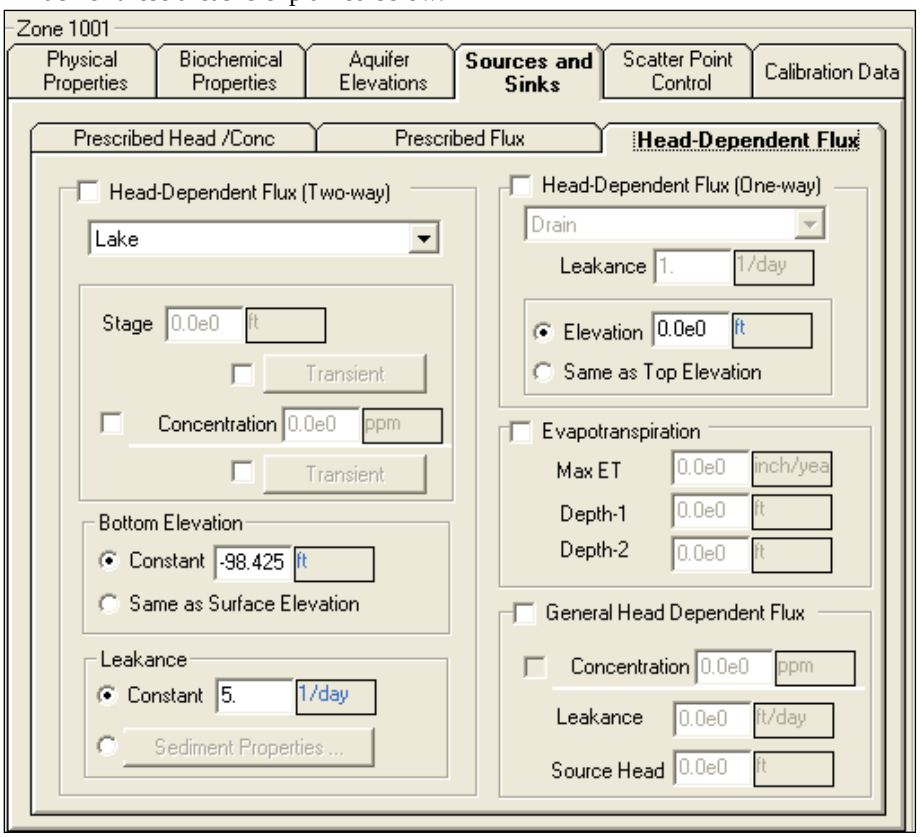


Figure 7-12 Head-Dependent Flux

HEAD-DEPENDENT FLUX (TWO-WAY) AREA

Two-way flux allows for water to infiltrate into and/or escape from the system, accounting for such processes as baseflow fluxes between the aquifer and a water body, and effects of lowering or rising of the water table on baseflow etc. Checking the box next to **'Head-Dependent Flux (Two-Way)'** defines the zone as an area of aquifer discharge/recharge (depending on the relative local head). This area can be used to model lakes, streams, etc.

The first field in the 'Head-Dependent Flux (Two-way)' area is where user can choose a name for the type of water body. Several choices are available from a drop-down menu which includes **Lake, Wetland, Swamp, Pond, Stream, Creek, Slough, Ditch, Seeps and Drains**. Choosing the name is only arbitrary and does not have any impact on model results. The chosen name can help refine the water budget and mass balance estimates (please refer to **Section 14.2** for details on water budget and mass balance)

There are three sub areas within 'Head-Dependent Flux (Two-Way)' area. These are: associated with river zones and each has its own area within the larger **River** area (they are active only when the **River** box is checked):

- **Stage**
- **Bottom Elevation**
- **Leakance**

Stage Area:
The **Stage** is simply the water level. It can be set as a constant value by selecting **Constant** (default) and entering a value in the field (the default entry is zero). Available units are meter (m) – default, centimeters (cm), feet (ft), and inches (inch).

The stage can alternately be set to transient conditions by selecting the **Transient** button (deactivated by default). Checking the box before the button activates it. The user can then click the button which opens the **Transient Settings** window. Refer to **Appendix C-I** for more information.

The user can also specify constant or transient concentrations in the water body by using the appropriate fields in this area.

Bottom Elevation Area:
The river bottom elevation is specified by selecting **Constant** (default) and entering a value in the field (-98.425 feet is the default value/entry). The available units of measure are feet (ft – the default), centimeters (cm), meters (m), and inches (in).

The river bottom may also be set to the same as the surface elevation by selecting **Same as Surface Elevation**. This allows the user to take advantage of topographical data (i.e. placing a river in a ground surface depression, etc.)

> **!** The software automatically turns off the river function in cells where the stage becomes less than the ground elevation.

Leakance Area:
The leakance, L, with a dimension of [T^{-1}], is defined as the discharge per unit area per unit head difference. It is assigned to the zone (or the water body) to establish the relative hydraulic connection between the feature and the aquifer. A higher leakance value indicates a stronger connection.

The flux q_r from the zone to the aquifer is calculated in the software as:

$$q_r = \begin{cases} L(h_r - h_a) & if \quad h_a > rivbot \\ L(h_r - rivbot) & if \quad h_a \le rivbot \end{cases} \qquad \textbf{(7.6.1.2.2)}$$

where:

q_r = specific discharge out of the zone [LT^{-1}]
h_r = head (stage) in the zone [L]
h_a = head in the aquifer [L]
rivbot = bottom elevation of the zone [L]
L = leakance [T^{-1}]

The user may specify the leakance directly by selecting **Constant** and entering a value in the field (the default entry is 5). Available units of measure include day^{-1} (default), hour^{-1}, sec^{-1}, month^{-1}, and year^{-1}.

The leakance may also be specified according to **Equation 7.6.1.2.3**:

$$L = \frac{K_r}{d} \qquad \textbf{7.6.1.2.3}$$

where:

L = leakance [T^{-1}]
K_r = vertical hydraulic conductivity of the bed sediments [LT^{-1}]
d = thickness of bed sediments [L]

!	If a situation is defined where d is equal to zero for any nodes, then the software automatically assigns a very small number to prevent **Equation 7.6.1.2.3** from having a zero in the denominator. This situation will yield an extremely high leakance that shows the connection between the river and the aquifer as having no resistance.

The zone bed sediment properties are edited by clicking the **Sediment Properties** button that subsequently opens the **River Bottom Sediment Properties (RBSP)** window pictured in **Figure 7-13**. In this window, the user can enter a value in the **Sediment Cond K** field to set the K_r value. The default value/entry is 0.1, and default unit of measure is m/day, with cm/sec and ft/day also available.

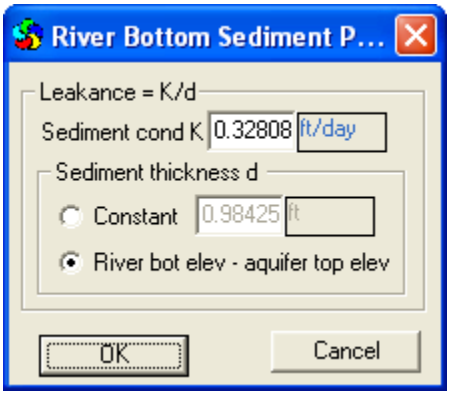


Figure 7-13 River Bottom Sediment Properties

In the **'Sediment thickness d'** area in the window, the user can select **Constant** and subsequently enter a value to set the thickness of the river bottom sediments. The default value/entry is 0.98425 ft, with centimeters (cm), meters (m), and inches (in) also available.

Alternately, the user may set the thickness to be equal to **River bot elev – aquifer top elev** or, in other words, 'the difference between the bottom of the river and the top elevation of the aquifer'.

!	

If unconfined situations are expected or develop during the simulation, it is recommended to use the **Constant** setting to define the sediment thickness. The problem arises because the top of the aquifer and water table no longer coincide in an unconfined aquifer, and saturated thickness may change.

HEAD-DEPENDENT FLUX (ONE-WAY) AREA

Checking the box before '**Head-Dependent Flux (One-Way)**' defines the zone as an area of direct outward seepage flux, relative to the aquifer (such as a wetland, drain, trench, or quarry). A drain does not supply any aquifer recharge. The relative local head controls the flux between the aquifer and the water body. Outward flux from the aquifer into zone exists when the local head in the aquifer exceeds the elevation of the zone. The equation controlling the flow of water into the zone is:

$$q_d = \begin{cases} L(h_a - d) & if \quad h_a > d \\ 0 & if \quad h_a \le d \end{cases} \qquad \textbf{(7.6.1.2.1)}$$

where:

q_d = specific discharge into the zone [LT^{-1}]
h_a = head in the aquifer [L]
d = elevation of the zone [L]
L = leakance [T^{-1}]

The user may specify the leakance for the zone in the **Leakance** field.

In the **Elevation** field, the user may enter a value for the elevation of the zone. The default value/entry is 0.0e0. The default units are meters (m), with centimeters (cm), feet (ft), and inches (inch) all available.

The user may also choose to set the zone elevation equal to the top elevation of the aquifer by selecting **Same as Top Elevation**.

EVAPOTRANSPIRATION AREA

The user can specify the head-dependent evapotranspiration (ET) flux [LT^{-1}] that will occur in a zone. The three fields in the ET area are explained below:

Max ET is the maximum ET flux which takes place when head in the aquifer is equal to or very close to surface elevation.

Depth-1 is difference between the surface elevation and the head in the aquifer for which ET will be maximum.

Depth-2 is 'extinction depth' below which no ET will take place. It is given by the difference between the surface elevation and the head in the aquifer at or below which ET flux will be zero.

When the head in the aquifer is above Depth-1, the ET flux from the zone is equal to Max ET. When head in the aquifer is between Depth-1 and Depth-2, The model calculates ET flux based on linear interpolation from Max ET at Depth-1 to zero at Depth-2. When head in the aquifer is below Depth-2, ET flux in the zone is zero.

Additionally, the user can set ET rates to specific levels, to test variation throughout the vertical extent of their model.

GENERAL HEAD DEPENDENT FLUX AREA

In this area, the user can specify values for **Concentration** (ppm), **Leakance** (ft/day), and **Source Head** (ft).

7.6.1.5. Scatter Point Control

Scatter point control tab is shown in **Figure 7-14.** This is where the user can manipulate scatter points (discussed later in this chapter) in any of the following ways:

> **!** In order to access the 'Scatter Point Control' tab, the user should first select the model zone in which the scatter points are located.

- **Turn Starting Head to Prescribed Head**
  After importing or interactively assigning a starting head in IGW, clicking on this button will convert starting head data to a prescribed head boundary.

- **Turn Prescribed Head to Starting Head**
  This option will convert prescribed head to starting head.

- **Toggle "Use as Starting Head" Flag**
  Clicking this button will display a flag sign attached to the scatter points belonging to the zone that is currently displayed.

- **Copy Starting Head to Calibration Head**

  Starting head in the model can be assigned as calibration head. After running the model, the user can use this calibration head data to evaluate the fit of their calibration by comparing it to the head distribution from the model.

- **Delete All Scatter Points in This Zone**
  If the user selects this option, then all scatter points in the given zone will be deleted, after a warning message is displayed.

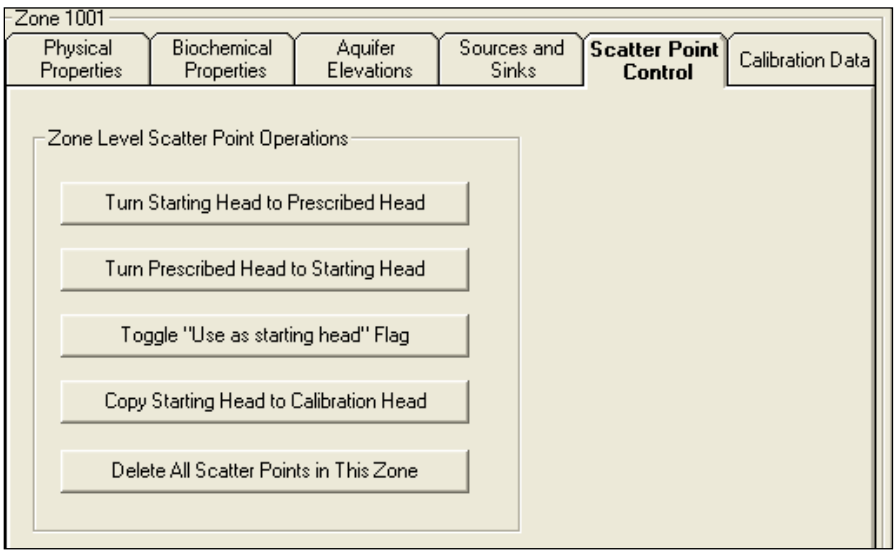


Figure 7-14 Scatter Point Control

7.6.1.6. Calibration Data

In order to calibrate existing data to the results from a model, the user can simply create a scatter point(s) in the corresponding model zone and then assign hydraulic head with **Head** (ft by default), concentration data with **Concentration** (ppm by default), and hydraulic conductivity with **Conductivity** (ft/day by default) to that particular scatter point. Calibration Data tab is shown in **Figure 7-15.**

After running the model, the calibration data points will show as flags of different colors in the model area. Red represents maximum difference between the point value and model prediction while the blue denotes the minimum difference. In between red and blue, a band of rainbow colors is used.

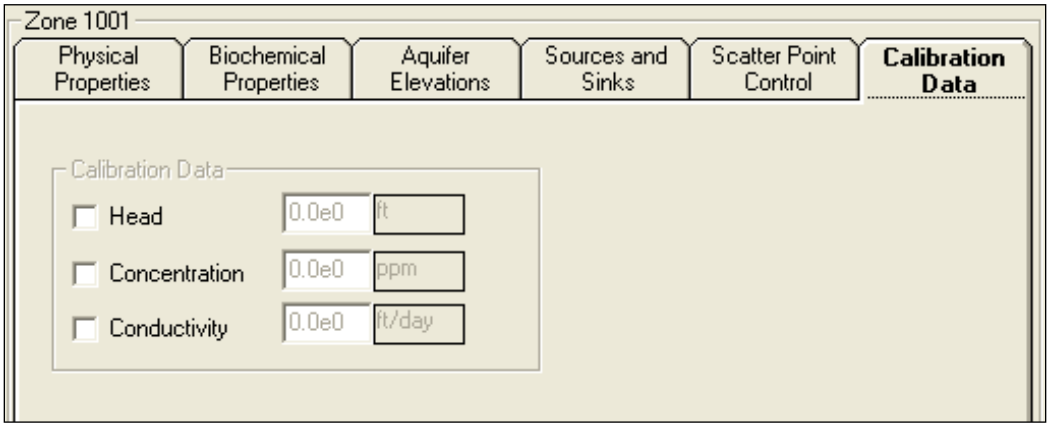


Figure 7-15 Calibration Data

7.6.2. Visualization and Domain Control

This interface is divided into the **Zone Type (ZTA)**, **Zone Color/Pattern (ZCA)**, one untitled area referred to as the **Geometrical Information Area (GIA)**, a check box, **Perform Mass Balance**, and a button, **Interpolation Model** (**Figure 7-16**). These features are discussed in the following subsections.

Figure 7-16 Visualization/Domain Control Area

7.6.2.1. Zone Type

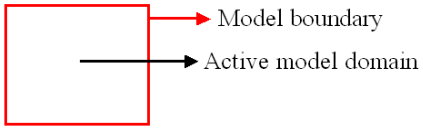
The **Zone Type** offers the user 3 selections for implementing the zone in the software:

- **Active,**
- **Inactive,** and
- **For Display**.

The default setting is **Active**. This setting includes the zone in software model calculations.

Selecting **Inactive** sets the zone as inactive in the Working Area. Its transimissivity is set to zero in software calculations.

Active model cells are defined based on the model boundaries. Cells within that boundary are called active cells, while the outer cells are inactive cells.



Cells of the Working Area that are not associated with a zone are termed **Inactive Cells**. The software internally assigns these a transmissivity value of zero.

Selecting **For Display** sets the zone as a drawing tool only, though the area encompassed by the zone retains its previous attributes.

7.6.2.2. Zone Color / Pattern

The **Zone Color/Pattern area** offers the user options for visualization. The user can select a pattern from the drop-down menu, and select a color by clicking the **Color** button (or the color outline area) and selecting the desired color from the **Color** window that appears. User can also choose the line width for the zone from the drop down list between 1.0 and 10.0 in the **Width** field.

7.6.2.3. Geometrical Information

This **GIA** shows two pieces of information: **Vertexes** and **Area**. **Vertexes** indicates the number of vertexes in the zone. **Area** indicates the conceptual area of the zone in the **Working Area**, in m^2. This area is the exact area of the zone as drawn, and not the area used in model calculations. The area used in model calculations is derived by approximating the zone shape with finite difference cells).

7.6.2.4. Zone Budget Check Box

Checking the box next to **Zone Budget**[14] allows for the zone to have its water and plume mass balances calculated and displayed. Please refer to **Section 14.2** for details on how to perform budget analysis.

When the model is discretized, an entry for both **Water Balance** and **Plume Mass Balance** will appear in the **TPS** (**Section 4.1.3**), under a place holder for the selected

[14] Refer to **Chapter 16** of the ***IGW Version 5.0P Tutorials*** document for an example of utilizing this feature and viewing the results of the simulation.

zone that resides in the **Mass/Water Balance** level. Clicking these will open the respective windows. These displays and settings are discussed in **Appendix D-I**.

7.6.2.5. Interpolation Model

The **Interpolation Model** button changes the view of the **LHP** to show the scatter point parameters that are being interpolated, and the number of points in each set. The button title changes to **Scatter Point Attribute** when viewing the alternate **LHP**.

The functionality of scatter points is discussed in **Section 7.7.**

7.6.2.6. Zone and Scatter Point Visibility

Checking the box next to **Zone Visible** allows the zone displayed in the **AE** to be visible in the model. Un-checking this box will hide this layer from view.

In the same manner, checking the box next to **Scatter Points Visible** allows the user-specified scatter points to appear in the **Working Area**.

7.6.2.7. Domain Control

By default, the model will run for the domain of every polygon drawn in the **Working Area**. After drawing a single model zone, if the user combines this with another polygon, then **IGW** will solve the model for all of the existing polygons (**Figure 7-17**).

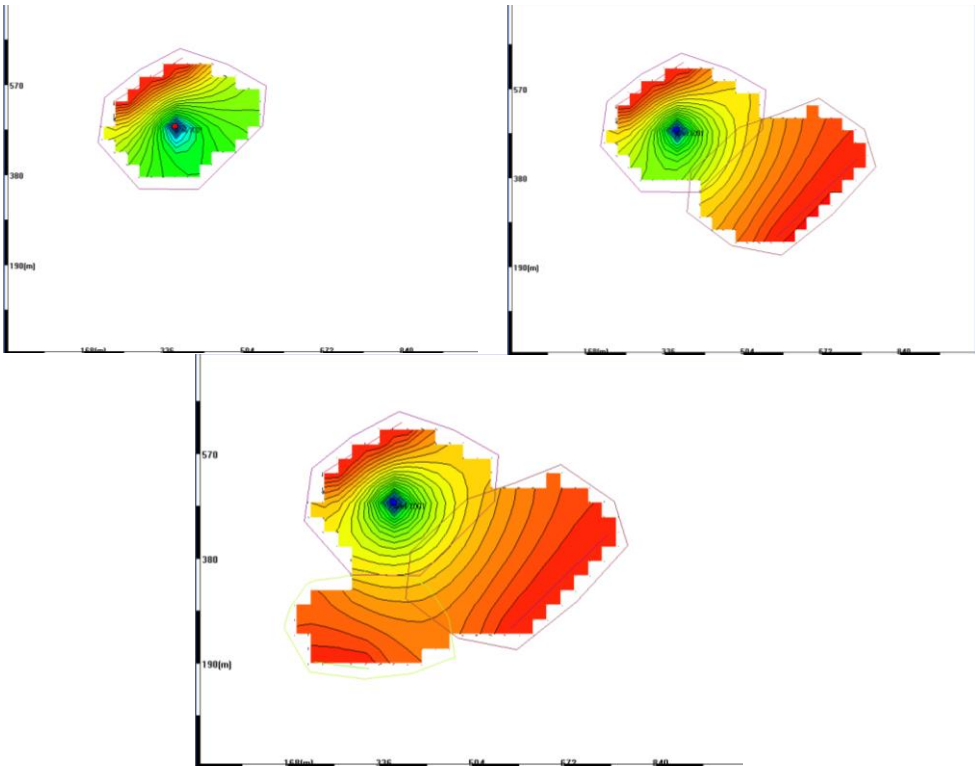


Figure 7-17 Examples on active model area concept

i	By default, **IGW** assigns no-flow conditions to the boundary of any polygons created in **Working Area**.

If the user assigns one or more polygons as domain control, then **IGW** will solve the model for **ONLY** those polygons. One can assign a domain control by:

- Selecting a polygon in the **Attributes Explorer LHP** (or in the **Working Area**), and then
- Checking the **Domain Control** box.

In **Figure 7-18,** model is solved for two polygons which are defined as domain control.

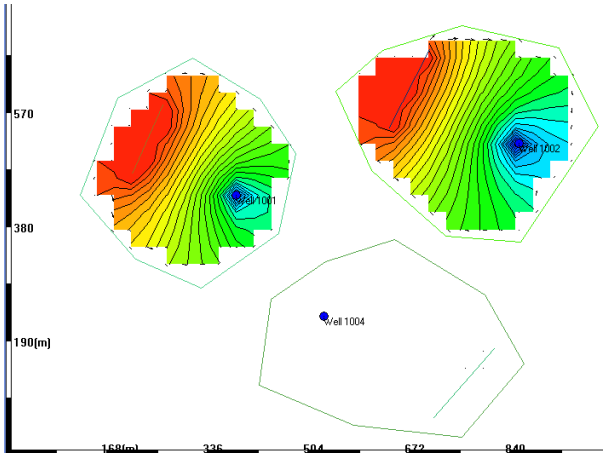


Figure 7-18 Assigning Domain Control

One of the advantages of using the **Domain Control** option is illustrated in **Figure 7-19**. The user may want to overlap other regular or irregularly-shaped polygons onto an existing model zone (e.g. adding recharge polygons). Since **IGW** will solve the model for every active cell in the **Working Area**, the user should assign the real model polygon (here, watershed) as the domain control. **Domain Control** will therefore make it easier to define the model boundary.

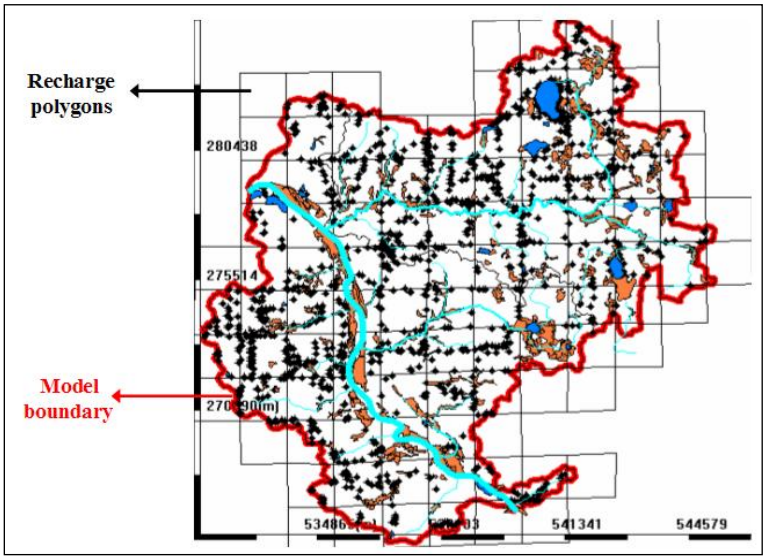


Figure 7-19 Selecting Watershed Boundary as Domain Control

7.7. Scatter Points

Scatter points are discrete points that may be associated with zones to achieve greater resolution when data (such as bore hole logs) are available at one or more points in the model domain. IGW Version 5.0P can interpolate values at scatter points for spatial attributes in the associated zones (see **Section 7.7.6**). The user can add scatter points in model zones in any (or all) of the following ways:

- manually add scatter points (see **Section 7.7.1**);
- import from a file (see **Section 7.7.4**); and,
- import from GIS data (see **Section 7.7.5**).

The following subsections describe the implementation and functionality of scatter points.

7.7.1. Defining Scatter Points

Scatter points must be associated with a zone in the **Working Area**. Therefore, the first step in defining scatter points is to make the desired zone active (see **Section 7.2**). If no zone is active when attempting to add a scatter point, a **Message** window will appear with the text 'You should define at least one zone first!'. Clicking the **OK** button closes the window.

After the appropriate zone has been made active, the next step is to click the **Add Scatter Point** button. This puts the cursor in **draw mode**. The user may now define a scatter point in the **Working Area** by simply clicking the mouse at the desired location.

Alternately, the user may type in the coordinates for each scatter point (in the **VCI – Section 3.13**) instead of clicking the mouse at each location. This method is not limited by the resolution of the screen/mouse relationship and allows for more accurate placement of scatter points within the **Working Area**.

When a scatter point is defined in the **Working Area**, it becomes the active feature. See **Figure 7-20.**

At this point, the cursor is still in **draw mode** and the user may continue to add scatter points as desired.

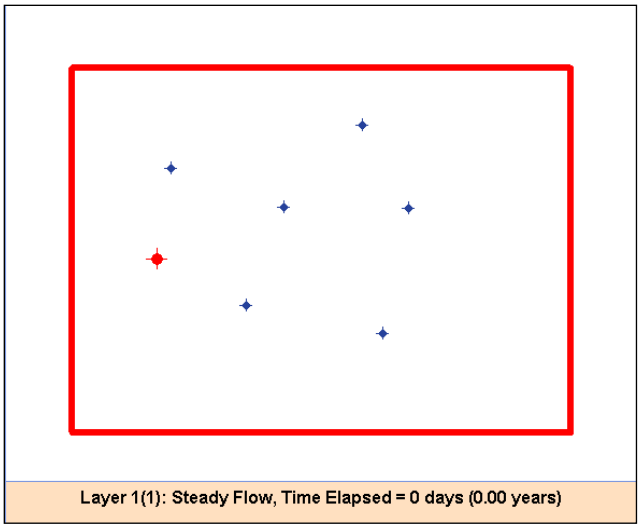


Figure 7-20 Scatter Points in the Working Area

I Scatter points may be defined anywhere within the **Working Area**, regardless of the size of the selected zone. Scatter points defined outside of the zone will still contribute to the interpolation/simulation scheme for the selected zone (in the same fashion as those defined within the zone), and still affect only that zone. Therefore, the user should ensure that the correct zone is selected before defining scatter points.

7.7.2. Selecting Scatter Points

To select a scatter point in the **Working Area**, first click the **Select Scatter Point** button and then click on the desired scatter point. The scatter point turns bold and red, indicating that is currently selected.

If multiple scatter points exist at the same location, then this method will only allow the user to select the first scatter point defined. To select the others (or to select the first with an alternate method), access them in the **AE** (**Section 4.1.1**).

7.7.3. Assigning Attribute Values to Scatter Points

The scatter points appear in the **LHP** of **AE** under their parent zone. The user can select a point and then assign it spatial attributes and/or calibration values using appropriate fields in the **RHP**. A sample of the **AE** for scatter points is shown in **Figure 7-21**. The tabs in the **RHP** for scatter points are identical to those of the zones, accept that **Scatter Point Control** tab is inactive.

In the bottom portion of **RHP**, the user can change the point style (size and color), can type in the exact coordinates of the point, and can 'check' or 'uncheck' the box to choose whether or not the point be visible in the **Working Area**.

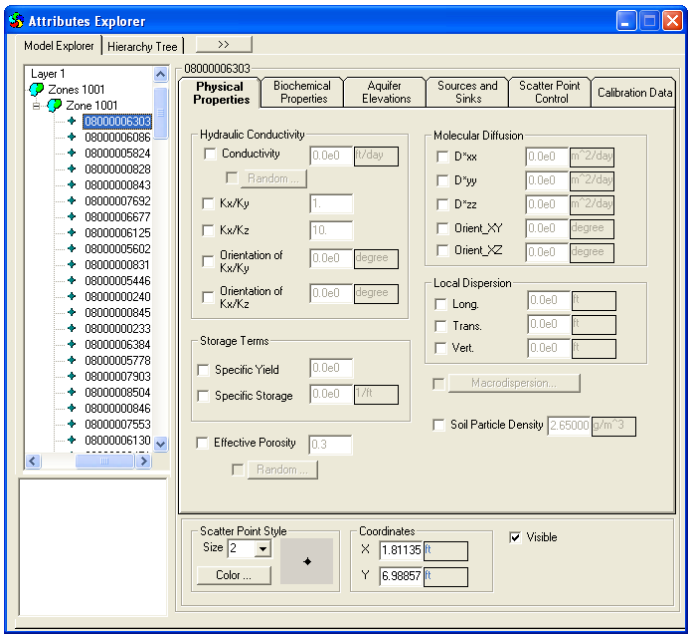


Figure 7-21 RHP for Scatter Points

After the scatter points have been defined in a zone, the user can assign the following spatial attribute values to these points in the **AE**.

- hydraulic conductivity;
- top elevation;
- bottom elevation; and,
- starting head.

Hydraulic conductivity value can be assigned to a point by selecting the 'Physical Properties' tab. User can check the box before 'Conductivity' and then enter the value for hydraulic conductivity in the required field. Aquifer top and bottom elevations can be assigned to the scatter points by selecting the 'Aquifer Elevations' tab and entering the values in the required fields. To assign starting head, the user can select 'Sources and Sinks' tab, then 'Prescribed Head/Conc' sub-tab. In the 'Prescribed Head/Conc' sub-tab, check the box before 'Prescribed Head', then check the radio button before 'Const' and enter the value in the field.

The user can also assign following calibration values to the scatter points:

- head;
- concentration; and,
- conductivity.

To assign the above calibration values to the scatter points, the user can select 'Calibration Data' tab. The user can check the required boxes and enter values in the fields.

7.7.4. Importing Scatter Points from a file

If there is large number of scatter points in a zone, it may become tedious to individually define every point and assign its attributes as explained in **Section 7.7.1** and **Section 7.7.3**. The scatter point information (location coordinates as well as attributes) for any number of scatter points can be arranged in a *.csv (comma separated value) file and directly imported into a zone selected in the model. The text file has to be of a specific format as explained below.

A text editor window in **Figure 7-22** shows a *.csv file for entering scatter points in a zone. Its format is explained as follows:

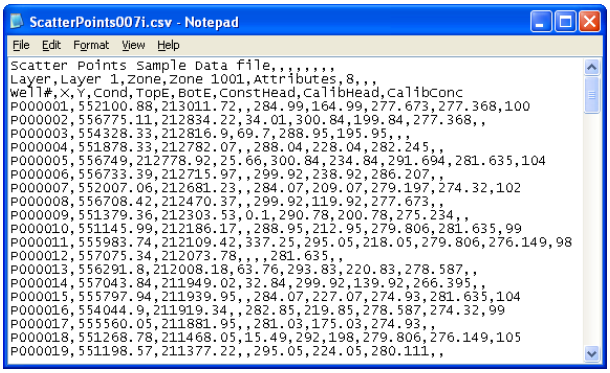


Figure 7-22 A sample file for scatter points

First line in the file is a text entry containing information about the file.

Second line has various comma separated fields. The first field is a keyword 'Layer' followed by the layer name in the model such as 'Layer 1'. The next entry is a keyword 'Zone' followed by the zone name such as 'Sample zone'. The next entry is a keyword 'Attribute' followed by the numeric entry for the maximum number of attributes that will be attached to a point in the zone.

The third line contains a list of keywords. Based on the keyword in this line, IGW will assign the corresponding attributes to the scatter points. The key words in the file stand for:

Well #	-	Scatter point name or ID (text entry)
X	-	X coordinate [m] of point location
Y	-	Y coordinate [m] of point location
Cond	-	Hydraulic conductivity value [m/day] at the point
TopE	-	Aquifer top elevation [m] at the point
BotE	-	Aquifer bottom elevation [m] at the point
ConstHead	-	Starting head value [m] at the point
CalibHead	-	Calibration head value [m] at the point
CalibConc	-	Calibration concentration value [ppm] at the point.

Please note the units associated with each of the above keywords. The values in the file must be in these units. Key words are part of IGW code, therefore the user cannot change them. A miss-spelled keyword will cause problem in reading file by IGW.

From fourth line onwards, the data should be entered corresponding to the keywords in third line. Data for Well #, X and Y is mandatory for every point. Other entries are optional. For more clarity, the *.csv file shown in a text editor in **Figure 7-22** is also shown in a spreadsheet view (**Figure 7-23**).

	A	B	C	D	E	F	G	H	I
1	Scatter Points Sample Data file								
2	Layer	Layer 1	Zone	Zone 1001	Attributes	8			
3	Well#	X	Y	Cond	TopE	BotE	ConstHead	CalibHead	CalibConc
4	P000001	552100.9	213011.7		284.99	164.99	277.673	277.368	100
5	P000002	556775.1	212834.2	34.01	300.84	199.84	277.368		
6	P000003	554328.3	212816.9	69.7	288.95	195.95			
7	P000004	551878.3	212782.1		288.04	228.04	282.245		
8	P000005	556749	212778.9	25.66	300.84	234.84	291.694	281.635	104
9	P000006	556733.4	212716		299.92	238.92	286.207		
10	P000007	552007.1	212681.2		284.07	209.07	279.197	274.32	102
11	P000008	556708.4	212470.4		299.92	119.92	277.673		
12	P000009	551379.4	212303.5	0.1	290.78	200.78	275.234		
13	P000010	551146	212186.2		288.95	212.95	279.806	281.635	99
14	P000011	555983.7	212109.4	337.25	295.05	218.05	279.806	276.149	98
15	P000012	557075.3	212073.8				281.635		
16	P000013	556291.8	212008.2	63.76	293.83	220.83	278.587		
17	P000014	557043.8	211949	32.84	299.92	139.92	266.395		
18	P000015	555797.9	211940		284.07	227.07	274.93	281.635	104
19	P000016	554044.9	211919.3		282.85	219.85	278.587	274.32	99
20	P000017	555560.1	211882		281.03	175.03	274.93		
21	P000018	551268.8	211468.1	15.49	292	198	279.806	276.149	105
22	P000019	551198.6	211377.2		295.05	224.05	280.111		
23	P000020	556791	211294.4		299.92	139.92			
24	P000021	557675.3	211289.3	38.12	284.07	239.07	280.416		

Figure 7-23 Scatter point file in spreadsheet view

The keywords shown above only pertain to the example file in **Figure 7-22.** A complete list of attribute keywords used in IGW is given in **Table 7-1**.

To import scatter points from a file into a zone, right-click on the zone in the **LHP** of **AE** and click on 'Import Scatter Points' from the drop down menu. The 'Open' window appears. Browse to the file location, select the file containing scatter point information and click 'Open'. The scatter points appear in the zone right away with all the attributes assigned from the information in the file. The user can select a point and see its assigned attributes in the **RHP** of **AE**.

Table 7-1 Attribute Keywords

ATTRIBUTE	KEYWORD	REFERENCE SECTION
x-coordinate	x	-
y-coordinate	y	-
z-coordinate	z	-
partition coefficient	kd	**7.6.1.1– 'Partitioning – Kd'**
top elevation	TopE	**7.6.1.3#**
bottom elevation	BotE	**7.6.1.3#**
starting head	starthead	**7.6.1.4.1#**
prescribed head	consthead	**7.6.1.4.1– 'Prescribed Head Area'**
calibration data head	calibhead	**7.6.1.6 – 'Calibration Data Layer'**
calibration data concentration	calibconc	**7.6.1.6 – 'Calibration Data Layer'**
calibration conductivity	calibcond	**7.6.1.6 – 'Calibration Data Layer'**
surface elevation	surface	**7.6.1.3#**
instantaneous concentration	instconc	**7.6.1.4.1 – 'Source Concentration Area'**
constant concentration	constconc	**7.6.1.4.1 – 'Source Concentration Area'**
effective porosity	poro	**7.6.1.1#**
recharge	rech	**7.6.1.4.2 – 'Const Rech' and 'Transient Rech… Button'**
dispersivity – longitudinal	displ	**7.6.1.1 – 'Local Dispersivity'**
dispersivity – transverse	dispt	**7.6.1.1 – 'Local Dispersivity'**
drain elevation	drnelev	**7.6.1.4.3 – 'Drain'**
drain leakance	drnleak	**7.6.1.4.3 – 'Drain'**
river bed elevation	rvbedelev	**7.6.1.4.3 – 'River'**
river stage	rivstage	**7.6.1.4.3 – 'River'**
river leakance	rivleak	**7.6.1.4.3 – 'River'**
specific storage	ss	**7.6.1.1#**
specific yield	sy	**7.6.1.1#**
Kx/Ky	AnisF	**7.6.1.1#**
Kx/Kz	AnisF_XZ	**7.6.1.1#**
Orientation of AnisF in XY	anisorient	**7.6.1.1#**
Orientation of AnisF in XZ	anisorient_XZ	**7.6.1.1#**
Decay Coefficient	decay coeff	**7.6.1.2#**
# see attribute named subsection within listed section		

7.7.5. Importing Scatter Points from GIS Data Sets

IGW Version 5.0P has a sophisticated GIS-interface to directly import GIS data into the **Working Area**. See **Chapter 22** for detailed features and functionality of GIS-interface. Refer to **Sections 22.5**, **22.6**, **22.15**, **22.16** and **22.17** for importing GIS based scatter point data in IGW.

7.7.6. Statistical Interpretation of Scatter Point Data

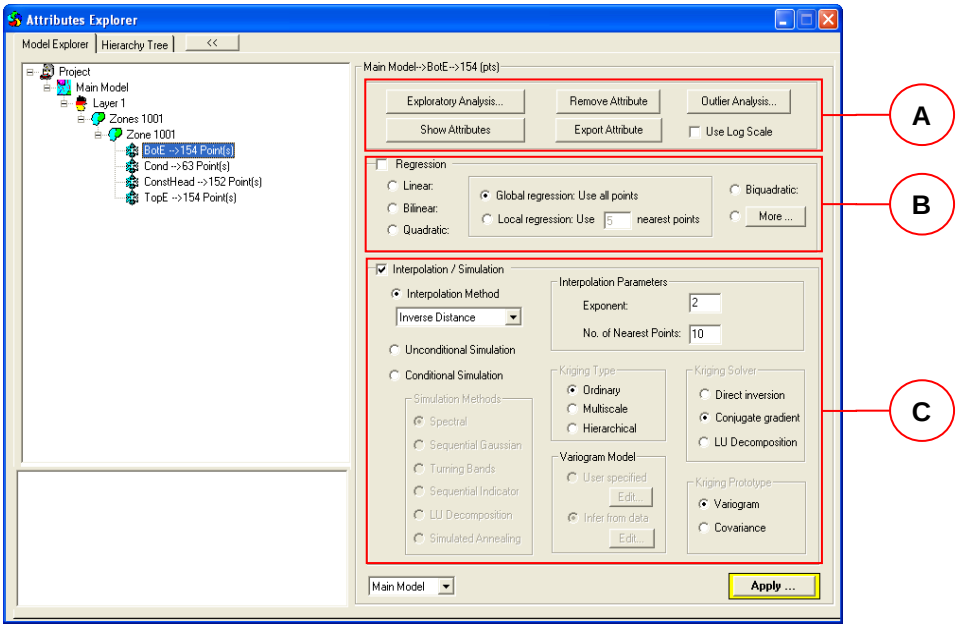
IGW Version 5.0P has a sophisticated statistical/geostatistical interface which can be used to analyze and interpret data attributes associated with scatter points. The **AE** window can be changed to alternate **LHP** and **RHP** views where user can access the statistical tools to analyze and interpret data at scatter points. Please see **Chapter 20** below

7.7.6.1. Alternate LHP and RHP

The alternate **LHP** and **RHP** can be accessed by clicking the **Show Interpolation Model** button in the zone **RHP** (see **Section 7.6.2.5**), or by right-clicking on any zone in the **LHP** and selecting **Switch List** from the drop-down menu. The alternate LHP and RHP in the AE window is shown in **Figure 7-24**.

> **i** The alternate view changes the appearance of the scatter points in the LHP. Individual scatter points under a zone are replaced by entries for each parameter that have been specified by the scatter points. These entries also indicate the number of scatter points that have this particular parameter specified.

Each entry in **LHP** has its own **RHP** in which the user may specify the conditions for statistical analysis of the group of scatter points. The format for all of the alternate **RHP**s is the same.



A – Top area for exploratory data analysis

B – Regression area

C – Interpolation/simulation area

Figure 7-24 Alternate LHP and RHP for Scatter Points

As seen in **Figure 7-24**, the "BotE --> 154 Point(s)" is highlighted in the LHP. The title in the **RHP** reads the same. This means that there are 154 scatter points in this zone that have explicitly specified aquifer bottom elevation values. These points can be statistically analyzed and interpreted.

7.7.6.2. Statistical Tools in RHP

The user can analyze the data on scatter points for their statistical parameters in a variety of ways. Spatial parameters in a model zone, such as aquifer elevations, hydraulic conductivities etc. can be interpolated within the zones. RHP provides the user to perform these analyses and interpretations. Please refer to **Chapter 20** for more detailed account of statistical tools for data processing.

There are three main areas in the alternate **RHP** as shown in **Figure 7-24**.

1) Top area with five buttons and a check box
2) Regression
3) Interpolation / Simulation

The top area deals more with descriptive statistics of the data while **Regression** and **Interpolation/Simulation** deal with geostatistical analysis of data. To set up a particular analysis the user may simply place a check mark in the appropriate box (Interpolation/Simulation is selected by default).

> **i** Checking both boxes sets up the **Combination Analysis** in which the data is first regressed, and then the residuals are analyzed through the desired interpolation/simulation scheme.

If there are not enough data to perform a certain analysis, the software will prompt the user (in a separate window) with a message indicating the specific problem. The more detailed settings are discussed further in the following subsections.

THE TOP AREA

Clicking the '**Exploratory Analysis**' button will open the 'Exploratory Data Analysis' window as shown in **Figure 7-25**. The window has a table on the right side and 4 tabs on the left. The table on the right shows descriptive statistics of the selected data.

The four tabs to the left include 'Histogram', 'PDF', 'CDF' and 'h-Scatter plot'. Below the table there are two sliding bars to set parameters for h-scatter. Clicking on any of these tabs will show a graph of the required statistic. The lag distance and tolerance for the h-scatter plot can be set using the sliding bars given at the bottom-right corner of the window. Whenever lag distance or tolerance is altered, the user should click the 'Draw' button to update the scatter plot. Similarly, number of intervals for histogram, CDF and PDF can be set in the bottom right area of this window.

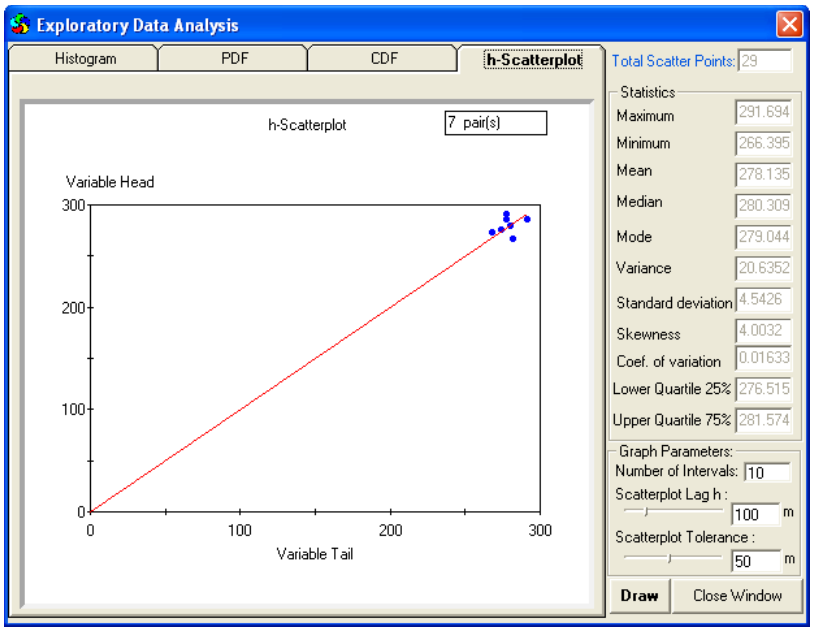


Figure 7-25 Exploratory Data Analysis window

Clicking the '**Show Attributes**' button will revert the alternate (**LHP** and **RHP**) view to the normal view.

Clicking the '**Remove Attributes**' button will remove the selected attributes from all points in the zone.

'**Export Attributes**' button is not active in IGW Version 5.0P

Clicking the '**Outlier Analysis**' button will open the 'Outlier Analysis' window as shown in **Figure 7-26**. In this window the user can choose the number of standard deviations from mean value a given data value should be taken as outlier. The user can also chose, with a check box, whether or not the outliers be global or based on deviation from the local trend. After choosing the number of standard deviations, the user can click on 'Detect Outliers' button. All the detected outliers will show in the widow at the left. In order to remove the detected outliers from the scatter point data set, click the 'Remove Outliers' button.

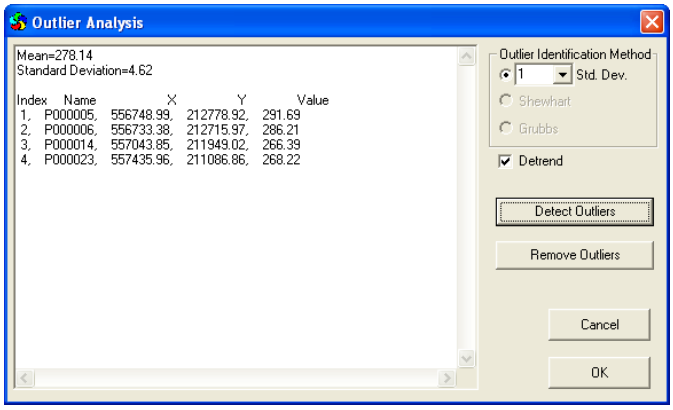


Figure 7-26 Outlier Analysis window

has five buttons and a check box for They are titled **Regression** and **Interpolation / Simulation**. The user is therefore presented with three main options for analyzing the scatter point data:

Finally, there is check box '**Use Log Scale**' in this area. By checking this box, the selected data will be analyzed and interpreted at log scale. The box is uncheck by default for all parameters. However, for hydraulic conductivity data, the box is checked by default.

REGRESSION

Regression area in RHP of AE is shown in **Figure 7-27**. The user may choose from four preset regression types: **Linear**, **Bilinear**, **Quadratic**, and **Biquadratic**. The user may also choose a custom regression format by selecting the **More** button. Clicking the **More** button opens the **Regression** window pictured in **Figure 7-28**.

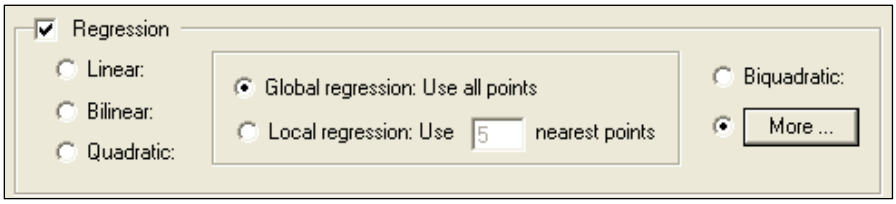


Figure 7-27 Attributes Explorer Regression Interface

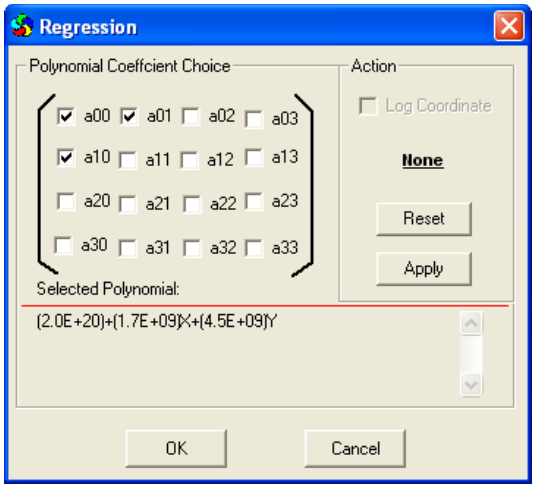


Figure 7-28 Regression Parameters

In this window, the user selectively chooses the parameters by placing a check mark in the boxes associated with the desired coefficients. The polynomial form is presented in the lower portion of the window. Clicking the **Apply** button shows the numerical form of the polynomial (the black **None** will change to red **Done**) and clicking the **Reset** button shows the algebraic form (changes the **Done** message to **None**). Clicking the **OK** button closes the window and sets the changes in the software. Clicking the **Cancel** button closes the window and discards the changes. The user should note that **Log Coordinate** is always selected and therefore the interface is disabled.

!	The software will only allow the user to implement choices for which there are enough data to complete the analysis.

i	Using high-order regressions can produce unrealistically high values, especially for large polygons. The user should invoke these with discretion.

INTERPOLATION / SIMULATION

The user may choose from three options in this area: **Interpolation Method** (default), **Unconditional Simulation**, and **Conditional Simulation**.

For **Interpolation Method**, the user may choose (from the associated drop-down list) either **Inverse Distance** (default) or **Kriging Method**.

INVERSE DISTANCE

The **Inverse Distance** method determines parameter values based more heavily on closer points than distant ones. Refer to the ***IGW Version 5.0P Reference Manual*** for more details.

There are two parameters in the **Deterministic Parameters** area that need to be set when using the **Inverse Distance** interpolation method. The exponent is set in the **Inverse Distance Exponent** field and the number of nearest scatter points to use is set in the **No. of Nearest Points** field. The default values are 2 and the number of scatter points, respectively.

KRIGING METHOD

The **Kriging Method** is based on the assumption that points that are near each other have a certain degree of spatial correlation, whereas points that are widely spread are statistically independent. Kriging is a set of linear regression routines that minimize estimation variance from a predefined covariance model. Refer to the ***IGW Version 4.7 Reference Manual*** for more details.

From user can choose from the number of points to be used in kriging. The user also has choices for kriging type, solver, prototype and variogram model.

There are two options in the variogram model i.e. **User specified** and **Infer from data.** Clinking on the 'Edit' button for user specified choice opens the **Variogram Model Parameters** window shown in **Figure 7-29**. This window allows the user to explicitly define kriging parameters.

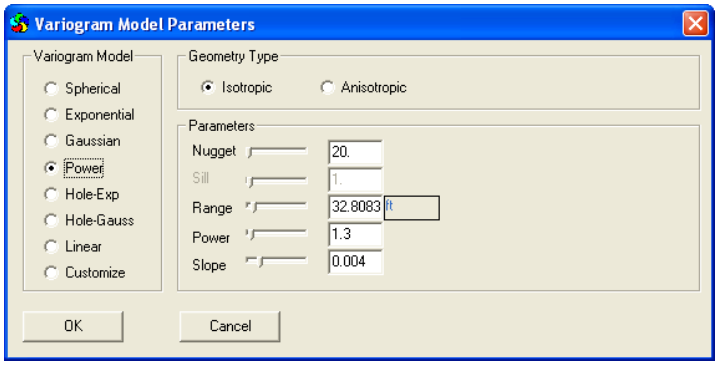


Figure 7-29 Variogram Model Parameters

Choosing **Infer from Data** opens the **Variogram** window as shown in **Figure 7-30**. This window provides visualization of the statistical analysis and allows the user to construct a variogram model based on scatter point data. The user can choose between automatic and manual settings for building the variogram model. When in manual mode, the user can choose influence radius, number of lags, mathematical function for variogram model and related parameters of the function. The display area continuously updates the shape of variogram data and fitted model whenever user makes any change in the available options. When

satisfied with the variogram model, the user can click the OK button and the final model will be applied in kriging interpolation.

This window provides visualization of the statistical analysis and allows the user to tweak some of the automatic settings.

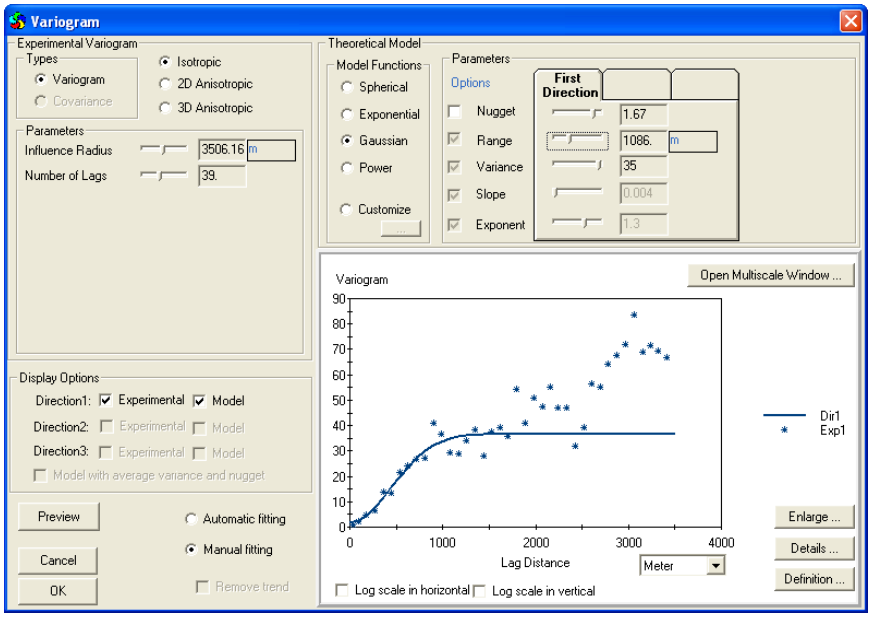


Figure 7-30 Variogram modeling window

For **Simulation**, the user may choose between **Unconditional Simulation** and **Conditional Simulation**.

UNCONDITIONAL SIMULATION

The **Unconditional Simulation** procedure generates a spatially correlated random field based on sample statistical parameters. This procedure will generate new values for the locations corresponding to the measured values.

There are six options available in the **Simulation Methods** area when this procedure is selected:

1) **Spectral Algorithm** (the default),
2) **Sequential Gaussian Simulation**,
3) **Turning Bands Algorithm**,
4) **Sequential Indicator**,
5) **LU Decomposition**, and
6) **Simulated Annealing**. **Appendices B-I-I**, **B-I-II**, and **B-I-III** discuss these methods (some of the writing in the referenced appendices is not directly applicable as the appendices were written with respect to the **Option of Unconditional Random Field** window – simply disregard those portions).

There are two options that appear in the **Spatial Statistics Parameters** area when **Unconditional Simulation** is chosen. Choosing **User Specified** allows the user to explicitly define simulation parameters by accessing the **Random Field Options** window (see **Appendix F-III**), clicking the appropriate **Option** button. Choosing **Infer from Data** instructs the software to automatically determine the parameters and apply them. Clicking the associated **Edit** button opens the **Variogram** window (see **Appendix F-II**). This window provides

visualization of the statistical analysis and allows the user to tweak some of the automatic settings.

Refer to the ***IGW Version 5.0P Reference Manual*** for mathematical details.

CONDITIONAL SIMULATION

The **Conditional Simulation** procedure generates a spatially correlated random field based on sample statistical parameters. This procedure is similar to the **Unconditional Simulation** method except the values for the locations corresponding to the measured values are held equal to those values when the field is generated.

There are two options available in the **Simulation Methods** area when this procedure is selected:

1) **Spectral Algorithm** (the default), and
2) **Sequential Gaussian** Simulation. **Appendices B-I-I**, **B-I-II**, and **B-I-III** discuss these methods (some of the writing in the referenced appendices is not directly applicable as the appendices were written with respect to the **Option of Unconditional Random Field** window – simply disregard those portions).

There are two options that appear in the **Spatial Statistics Parameters** area when **Conditional Simulation** is chosen. Choosing **User Specified** allows the user to explicitly define simulation parameters by accessing the **Random Field Options** window (see **Appendix F-III**), clicking the appropriate **Option** button. Choosing **Infer from Data** instructs the software to automatically determine the parameters and apply them. Clicking the associated **Edit** button opens the **Variogram** window (see **Appendix F-II**). This window provides visualization of the statistical analysis and allows the user to tweak some of the automatic settings.

Refer to the ***IGW Version 4.7 Reference Manual*** for mathematical details.

Chapter 8 POLYLINES

A polyline is basically a series of line segments. Polylines are used to define source/sink areas or flux monitoring boundaries in the Working Area. Polylines are typically used for 'thin' features that may not receive full model representation (due to the grid size) if defined as a zone. The following sections describe the implementation and functionality of polyline features.

8.1. Defining Polylines

The first step in defining a polyline is clicking the **Create a New Polyline and Assign Property** button, located at row 2, column 3. This puts the cursor in drawing mode. The user may now define a polyline in the Working Area by simply clicking the mouse at points along the desired series of line segments. **Figure 8-1** shows a polyline in the Working Area.

First, the user selects a beginning point at one end of the desired polyline and clicks the mouse.

The user moves the mouse cursor to another point along the desired polyline. A line will extend from the initial point to the current location of the cursor indicating the 'proposed' segment of the polyline. The user should adjust the location of the mouse cursor to make the 'proposed' edge coincide with the user's desired edge. Once this is done, clicking the mouse sets the point.

Next, the user should then move the mouse cursor to another point along the desired polyline. Again, a line will extend from the initial point to the current location of the cursor indicating the 'proposed' segment of the polyline. The user should adjust the location of the mouse cursor to make the 'proposed' segment coincide with the user's desired segment. Clicking the mouse will set the point. This process should be repeated until the desired polyline shape has been achieved.

Double-clicking the mouse ends the process and sets the polyline in the software. If the desired polyline were simply one line segment, then the user should double click at the second point instead of continuing the process.

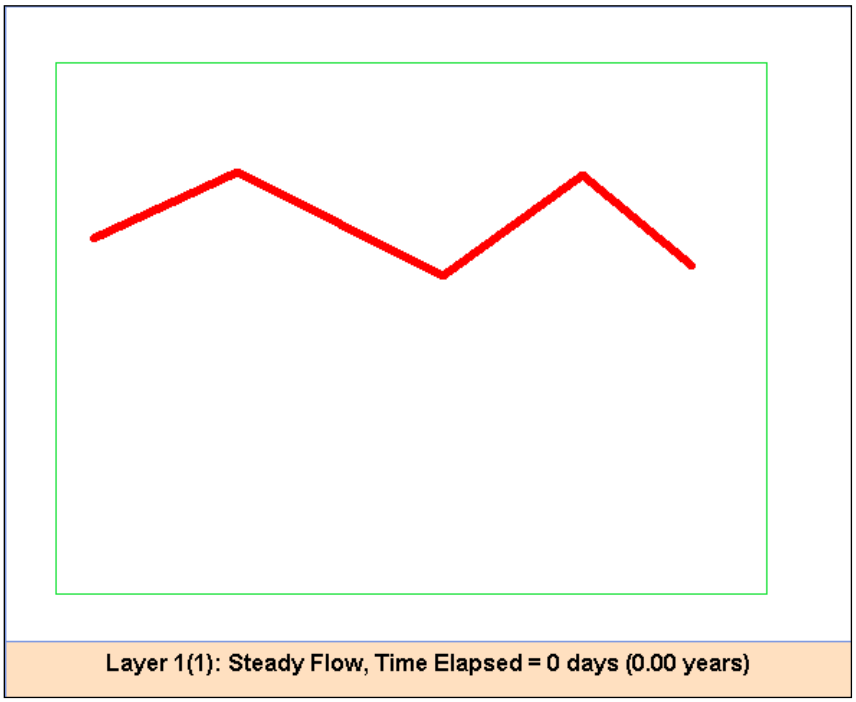


Figure 8-1 An Example Polyline in the Working Area

Alternately, the user may type in the coordinates for each segment endpoint (in the VCI – see **Section 3.13**) instead of clicking the mouse at the desired location. This method is not limited by the resolution of the screen/mouse relationship, and allows for a more precise development of polyline features.

When a polyline is defined in the software, it becomes the active feature.

At this point, the cursor is still in 'draw mode' and the user may continue to add polylines as desired.

8.2. Selecting Polylines

To select a polyline in the Working Area, first click the **Select a Polyline and Edit It** button (row 3, column 3), and then click the cursor at a point on the desired polyline. The polyline becomes outlined in red therefore indicating that is currently selected.

Alternately, the desired polyline may be selected in the **AE** (see **Section 4.1.1**).

8.3. Redefining Polylines

A polyline that has been defined in model can be redefined by placing the cursor in 'Node Edit' mode (see **Node Edit Mode** in **Section 3.16**). The user may change the shape of the polyline in either the Working Area or any submodel windows by:

1) Moving the existing vertices, and/or
2) Creating new vertices.

To move an existing vertex (line segment endpoint), click and hold the mouse above the black square that corresponds to the desired vertex, drag the cursor to the desired vertex location, and release the mouse button.

To create a new vertex (make one line segment into two), click and hold the mouse above the blue crosshair symbol (one exists between each vertex) nearest to the desired location of the new vertex. Drag the cursor to the desired vertex location and release the mouse button.

These steps may be repeated as many times as necessary until the desired polyline shape is achieved.

8.4. Moving Polylines

Polylines can be moved in the Working Area by selecting the **[CTRL]** key and clicking on the Polyline at the same time. While holding down the **[CTRL]** key, use the cursor to move the polyline into the user's desired position, and then release the cursor and **[CTRL]** key.

8.5. Setting Polyline Attributes

Polyline attributes are set in the AE (see **Section** Error! Reference source not found.). After ccessing the AE, the first step is to select the desired polyline in the LHP (see **Section 4.1.1**). Doing this brings up the 'Pline' RHP (see **Section 4.1.2**). A sample of the RHP for a polyline is shown in **Figure 8-2**.

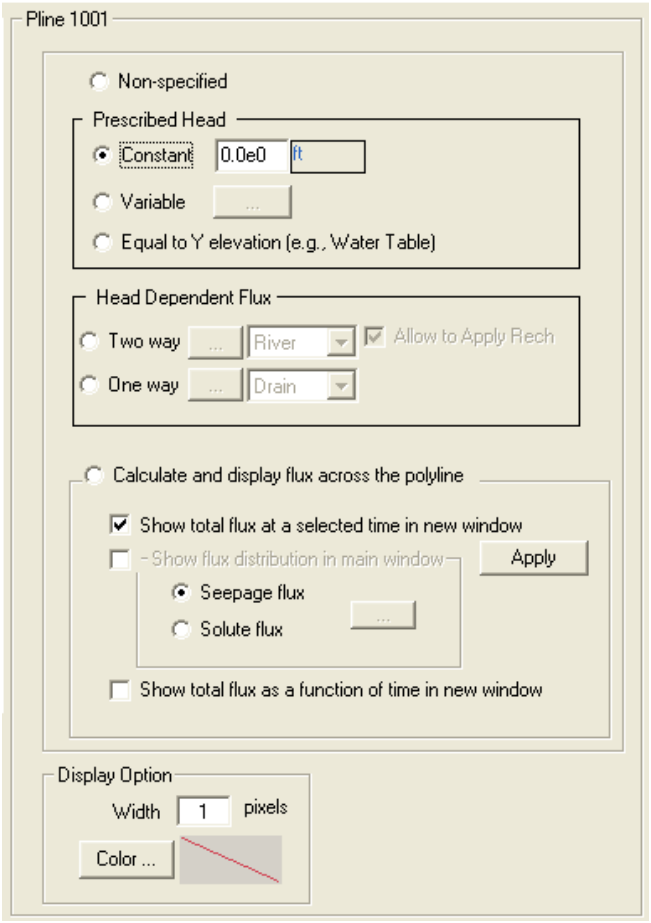


Figure 8-2 RHP for Polylines

The polyline attributes displayed in the AE are discussed in the following subsections.

8.5.1. Non-Specified

Selecting **Non-Specified** in the **Polyline Type** area makes the polyline non-functional in the model solution. It is in effect for drawing purposes only. This is the default setting.

8.5.2. Prescribed Head

This section allows the user to manipulate the polyline with a designated head type. The head choices are **Constant**, **Variable**, and **Equal to Y elevation** as shown in **Figure 8-3.**

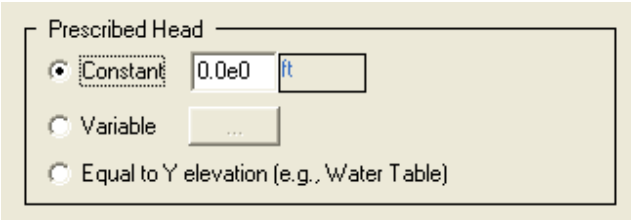


Figure 8-3 Prescribed Head area

CONSTANT HEAD

Selecting 'Constant' in the 'Prescribed Head' area sets the polyline at a constant head value. The desired value is entered in the field that appears to the right of the 'Constant Head' entry after it is selected. The default entry is 0.0e0. The default unit is meters (m) with centimeters (cm), feet (ft), and inches (inch) all available.

VARIABLE HEAD

Selecting **Variable** in the **Prescribed Head** area sets the polyline to have a linearly variable head along its total length. The desired values are entered in the **Edit Polyline Attributes** box that appears when the user selects the [...] box next to the **Variable** choice. Edit Polyline Attributes window is shown in **Figure 8-4.**

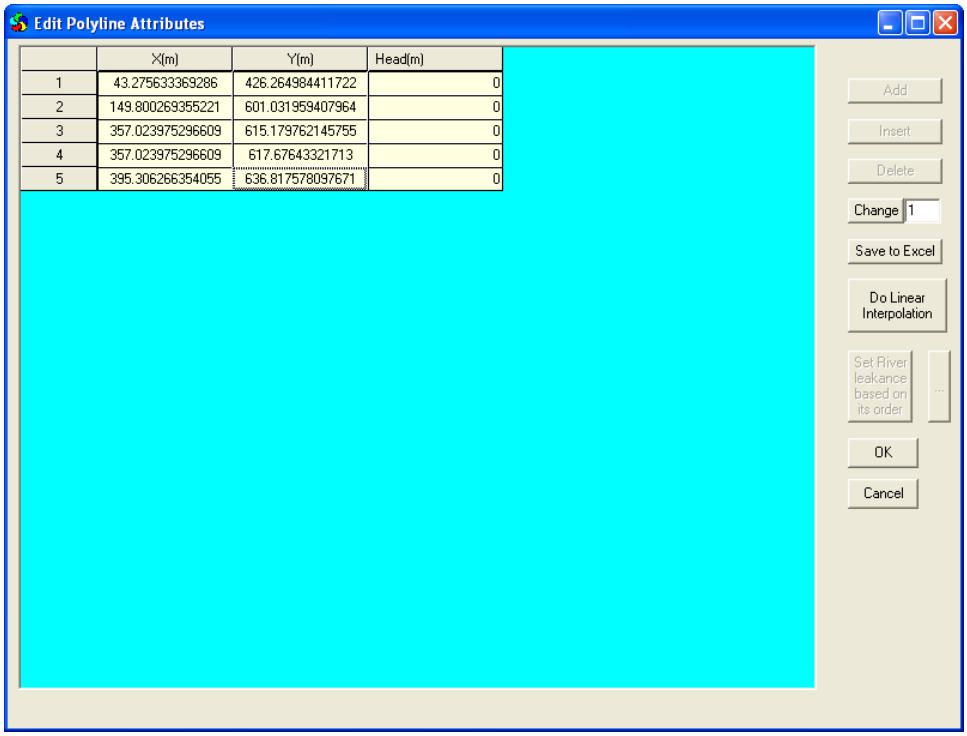


Figure 8-4 Editing Polyline Attributes for constant head

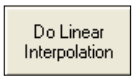
The first and second fields are the X and Y coordinates for the nodes of the polyline. The third field allows the user to specify a head value for each node location, thereby giving the polyline variability across its extent.

> **!** Note that all of the columns have a unit of meter. The user **cannot** change the unit in this table. User should make sure that while making any changes in the values, the values entered in meters.

Any value in the cells of the table shown (X, Y and Head) can be modified by the user by using the [Change] button. To change a value, first type the new value in the field provided next to the [Change] button. Then move then move the cursor and click in the cell where new value will be added (the cell is selected). Move the cursor back and click on the [Change] button. The value in the selected cell is update.

To save data present in the table to Microsoft® Excel®, click [Save to Excel] button. An Excel window opens with all the data already written in the spreadsheet. You may save the spread sheet just as you do in Excel®.

You can do linear interpolation of values across any column in the table. You can also do linear interpolation within a portion of the column. To do linear interpolation, enter the desired values in the first and last desired cells of the column using the [Change] button.

Click to select the first cell, and then holding the shift-key, click in the last cell. This will select all the cells from the first to the last cells. When cells are selected, just click on the in the 

button. You will see the linearly interpolated values from the first to the last cells selected.

HEAD = Y ELEVATION

Selecting **Head = Y Elevation (e.g. water table)** in the **Polyline Type** area sets the head associated with the polyline equal to the y-axis value at each point. This feature is especially useful in building 2D 'profile models'.

R Assigning the water table as river head is useful when modeling cross-sections of unconfined aquifers directly in the Working Area.

8.5.3. Head-Dependent Flux

This field, shown in **Figure 8-5**, allows the user to designate the polyline as either a 1-way or 2-way head-dependent flux, in the same manner that zones are designated in the AE.

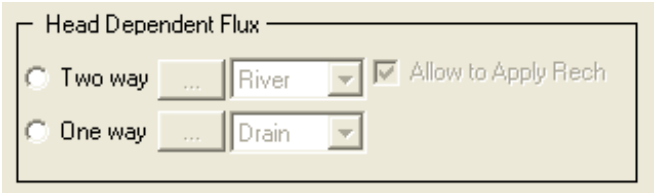


Figure 8-5 Head-Dependent Flux area

The user can specify the type of flux they prefer (river or drain). Using  button opens the window as shown in **Figure 8-6** for editing polyline coordinates and attributes. The user can choose whether or not the recharge flux be added to the 'head dependent flux' cells in the model by checking the box before 'Allow to Apply Rech'.

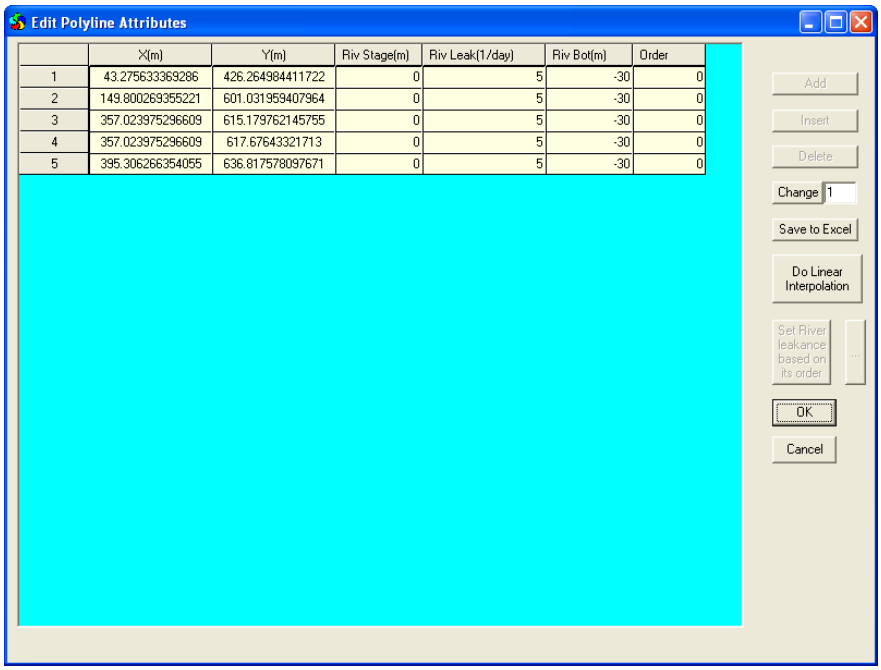


Figure 8-6 Editing Polyline Attributes for head dependent flux

Notice that all the buttons one the right hand side are same as in **Figure 8-4**. They also function the same way as explained for 'Prescribed Head'. However, you can see there are more columns in the table of values. Units for all these values are given in the column headers. The user cannot change the units.

Changing, interpolating, or saving the values to Excel spreadsheets is done the same way as explained above.

8.5.4. Calculate and Display Flux Across the Polyline

This field, shown in **Figure 8-7**, allows the user to estimate seepage and/or solute flux across a polyline. This feature is active only for multiple realizations of Monte Carlo simulations (**Chapter 18**). Total seepage and/or solute flux is estimated at the end of each realization. **Section 18.5.2.3** covers further details of this feature.

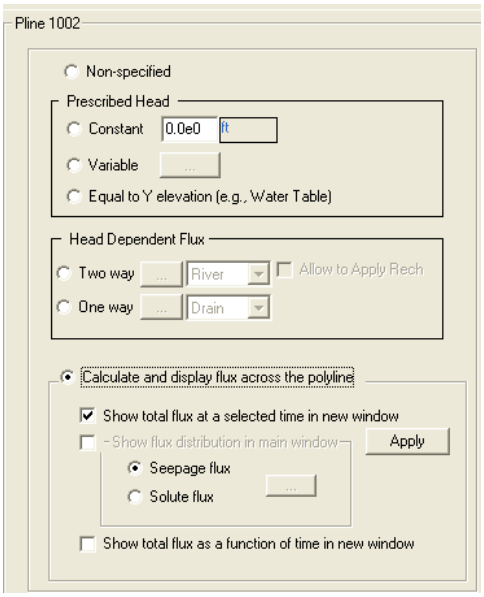


Figure 8-7 Selecting 'Calculate and display flux across the polyline' in the Polyline RHP

> **i** *IGW Version 5.0P* does not show total flux as a function of time for a single realization.

8.5.5. The Display Option Area

In the **Display Option** area, the user can set the width of the polyline in the 'Draw Width' field. This width is based on screen pixels.

The user may also change the color of the line by clicking the **Color** button that subsequently opens the color window.

> **i** The software assigns a polyline to every cell that the polyline is drawn through.

Chapter 9 WELLS

Wells are defined at discrete points in the Working Area, and used to define either source/sink points (injection and pumping wells, respectively), or to monitor heads and concentrations in the aquifer. The following sections describe the implementation and functionality of well features.

9.1. Defining Wells

The first step in defining a well[15] is clicking the 'Add a New Well' button, located at row 2, column 4. This puts the cursor in 'draw mode'. The user may now define a well in the Working Area by simply clicking the mouse at the desired point.

Alternately, the user may type in the coordinates for each well (in the VCI – see **Section 3.13**) instead of clicking the mouse at the each location. This method is not limited by the resolution of the screen/mouse relationship and allows for more precise placement of well features.

i When wells are defined, they become associated with the nearest node. Therefore, when the mouse is clicked to add a well, it may not appear in that exact location – it appears at the associated node. Note that the coordinates for the desired placement of the well are stored internally, so when the resolution of the model changes (or a higher resolution submodel is implemented) so does the location of the well (as it must stay associated with the closest node).

! Multiple wells, although having different real-world coordinates, may be associated with the same node in the model. In this case, the effects of each well are aggregated at the node.

At this point, the cursor is still in 'draw mode' and the user may continue to add wells as desired.

i Along with being a modeling component, a well can also be used as a visualization feature and can be used as a reference point. When a well is defined in the software, it becomes the active feature.

9.2. Selecting Wells

To select a well in the Working Area, first click the **Select a Well and Edit It** button and then click on the desired well. The well becomes red, indicating that is currently selected.

If multiple wells exist at the same location (or are associated with the same node), then this method will only allow the user to select the first well defined. To select the others (or to select the first with an alternate method), access them in the **AE** (**Section 4.1.1**).

9.3. Moving Wells

To move a well in the Working Area, click on **Select a Well and Edit It** button, select the well with the cursor, press and hold the **[CTRL]** key and move the well into the desired position.

[15] Refer to **Section 3.5** of the ***IGW Version 5.0P Tutorials*** document for a step-by-step example of defining a well.

If the model grid is coarse, then it may be difficult to move a well to the desired position. Alternately, the user may change well coordinates using Attributes Explorer/Well Location field.

R When moving an existing well in the Working Area, the software will automatically place the well at the location of the nearest node- not necessarily where the user clicks the button. Therefore, changing the grid size will affect the location of that well as it must stay associated with nodes. It is recommended to make the grid resolution finer in order to move a well closer to its respective real-world coordinates.

9.4. Setting Well Attributes

Well attributes[16] are set in the **AE** (see **Section**).

9.4.1. RHP for Global Attributes of Wells

Clicking on Wells at Group Level (refer to **Figure 4-2**) in **LHP** opens the global **RHP** for a wells' group as shown in **Figure 9-1**. The user can change and/or assign global parameters to the wells' group in this **RHP**.

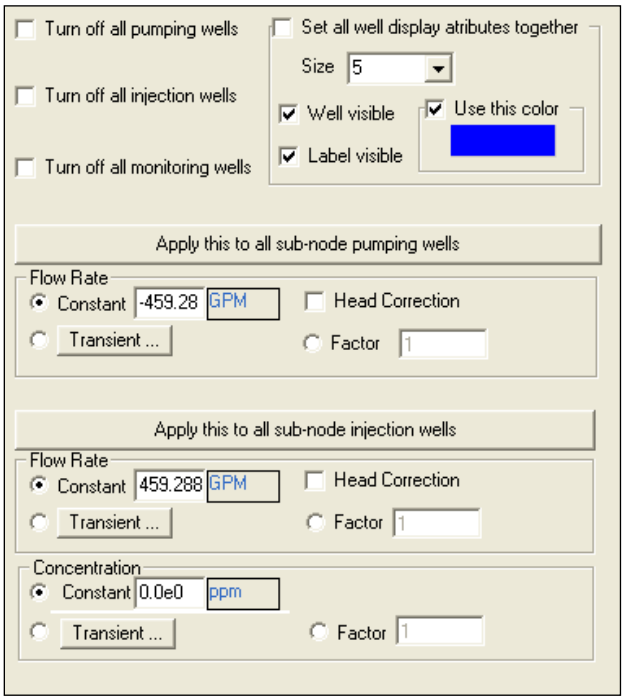


Figure 9-1 Wells Attributes in the AE

TURN OFF WELLS

The top left area of the RHP gives various turn-off options for the wells' group The options include **Turn off all pumping wells**, **Turn off all injection wells**, and **Turn off all monitoring wells.** The user may select as many of these options at once as they desire for modeling.

[16] Refer to **Section 3.9** of the ***IGW Version 5.0P Tutorials*** document for examples of defining well attributes.

DISPLAY OPTIONS

The top left area of the RHP gives various display options for the wells. The options include display size, color, and visibility (for well and/or label) options.

APPLY SETTINGS TO ALL SUB-NODE WELLS

This feature allows the user to specify a given flow rate for wells (in **GPM** by default), a head correction factor, and/or give the wells transient values. It also allows injection wells to have a specific contamination concentration for each location.

Clicking the **Apply this to all sub-node pumping wells** box will implement the stated values and/or conditions to all pumping wells in the group. Likewise, clicking the **Apply this to all sub-node injection wells** box will implement the stated values and/or conditions to all injection wells in the group.

9.4.2. RHP for Attributes of Individual Wells

After accessing the **AE**, the first step is to select the desired well in the **LHP** (see **Section 4.1.1**). Doing this brings up the **RHP** for wells (see **Section 4.1.2**) as shown in **Figure 9-2**.

The well attributes shown in the **AE** are discussed in the following subsections.

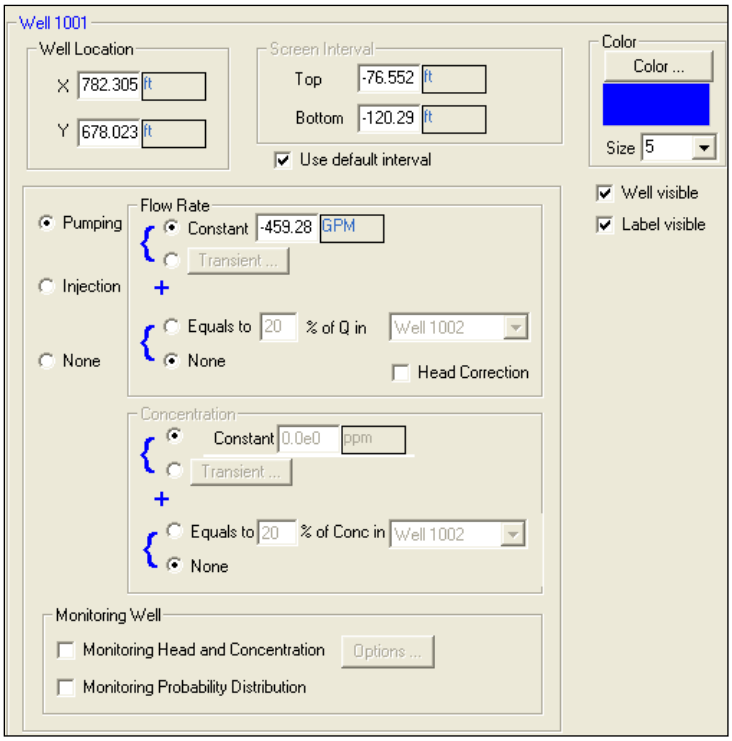


Figure 9-2 RHP for Wells

WELL LOCATION

This field displays the location of the well in terms of X and Y coordinates. Default units are in feet (ft).

SCREEN INTERVAL

The **Top** (upper elevation of the screen interval) and **Bot** (bottom elevation of the screen interval) fields display the depths over which the well is screened. These are set by default (the middle one-third of the aquifer thickness) but can be changed by unchecking the **Use default interval** box

and subsequently entering numbers and selecting the unit of measure (meters [m], by default; centimeters [cm], feet [ft], and inches [inch] available).

R	The screen interval is only considered within a profile model. In all other modeling situations, the well is treated as fully penetrating.

COLOR AREA

The current color of the well is displayed in the **Color Area**. Clicking the **Color** button (or in the sample color patch) opens the color window and allows the user to specify a desired color.

The user may also specify size of the well on the screen (5 is default), and has the option to not display both the well and its label by un-selecting the appropriate boxes.

WELL TYPE AREA

The user can select between four options for the **Well Type.**

1) **Pumping Well**
2) **Injection Well**
3) **None**
4) **Monitoring Well**[17]

The types are discussed in the following subsections.

PUMPING WELL

Selecting 'Pumping Well' (the default) sets the well as one that withdraws water from the aquifer. In the 'Flow Rate' area, shown in **Figure 9-3**, the user may select either 'Constant Q' or 'Transient Q'.

If 'Constant Q' is selected (default), the well pumps constantly at the rate defined in the associated field. The default value is -459.3 (the negative sing (-) indicates water is being withdrawn from the aquifer) and the default unit is gallons per minute [GPM] (available units include: m^3/sec, liter/sec, m^3/day, and million gallons per day [MGD]). The user may change these fields as desired.

If the 'Transient …' button is selected (the deactivated text becomes active), the well is set to pump according to the default transient conditions. The user may edit these transient conditions by clicking the activated 'Transient …' button and then opening the 'Transient Settings' window. This window and its associated parameters are discussed in **Appendix C-I**. (Note that the default parameters for this situation are not necessarily the same as for the situation discussed in the appendix.)

The user also now has the option to specify one well's pumping rate as a percentage of another well's discharge in the group. By default, 'None' is selected for this feature thereby disabling it, however the user may choose to enable this for modeling purposes such as cross-well comparison and sensitivity analysis.

[17] Refer to **Chapter 11** of the ***IGW Version 5.0P Tutorials*** document for an example of defining a monitoring well and viewing the results.

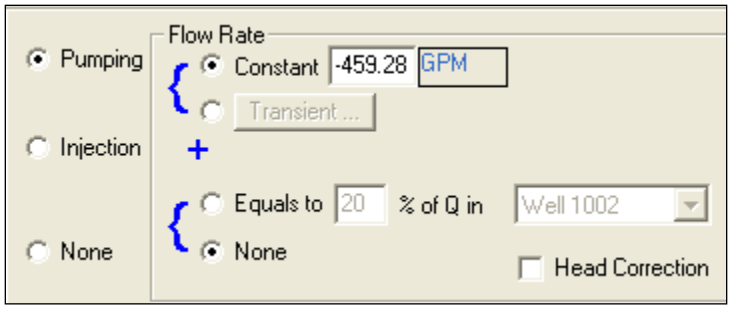


Figure 9-3 Assigning Flow Rate

The 'Head Correction' check box in this field is used to instruct the software to calculate a head value in the well that more accurately reflects the extent of the drawdown (pump-up) caused by the well pumping (injection) -- (it is unchecked by default). When checked, the corrected head value will be displayed in the software but will not be used in any calculations.

> **I** Checking **Head Correction** box for one well activates it for all others in the model although other wells will not indicate that it is active in their respective **Head Correction** check boxes.

INJECTION WELL

Selecting 'Injection Well' sets the well as one that injects water into the aquifer. In the 'Flow Rate' area, the user may select either 'Constant' or 'Transient …'.

The discussion for the 'Flow Rate' area is the same for the injection well except that the lack of a (-) sign indicates that water is being added to the aquifer.

The injection well selection allows the 'Concentration' field to become active. Here, the user can specify a contaminant amount if desired (in ppm by default). Also the user has the ability to allow one well's contaminant level to equal a percentage of another well in the group, in the same manner as the operation found in 'Flow Rate'.

NONE

Selecting **None** sets the well to be displayed in the Working Area but not affect any model calculations or be considered in any way in the model solution.

MONITORING WELL

Monitoring Well area is shown in **Figure 9-4**. Selecting 'Monitoring Well' (the area heading text becomes active) sets the well as one that can monitor heads, concentrations, probabilities, and correlations.

There are two types of monitoring available for a well:

1) Monitoring Head and Concentration, and
2) Monitoring Probability Distribution.

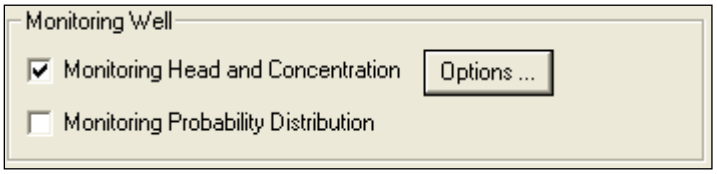


Figure 9-4 Defining a Monitoring Well

Both may be selected simultaneously, but 'Monitoring Probability Distribution' is useful only in Monte Carlo simulations (see **Section 18.5.2.2**).

Clicking the 'Options' button next to 'Monitoring Head and Concentration' opens the 'Input Head / Concentration Data' window. It is discussed in the following subsection.

THE INPUT HEAD / CONCENTRATION DATA WINDOW

The window is shown in **Figure 9-5.** This window allows the user to enter calibration data (head data on the 'Head Data' layer and concentration data on the 'Concentration Data' layer) to construct a graph against which data from the model can be compared.

The user can either manually enter the head/concentration data in the table or import from a file using the 'Import from File' button.

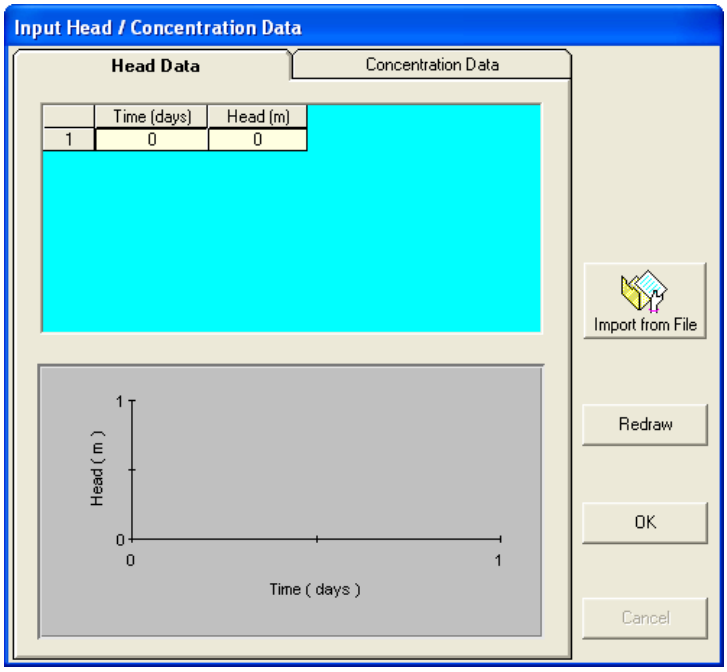


Figure 9-5 Input for Head/Concentration Data

Data is entered manually by clicking the appropriate field and entering the respective number. Additional points can be added by pressing enter on the keyboard (when the cursor is in the data cell). A datum can be deleted by pressing the space bar when the cursor is in one of its data cells (and subsequently confirming the desire to delete it). Clicking the 'Redraw' button updates the plot area to show the data points. Clicking the 'OK' button closes the window.

Clicking the 'Import from File' button first displays a warning "this operation will overwrite the existing head and concentration data in tables. Do you want to continue?" Clicking the 'Yes' button will bring the 'Open' window from where the user can browse to the required file location and upload data into the table.

*IGW Version 5.*0P supports input data files in *.csv format.

Chapter 10 PARTICLES

Various types of particle features can be added to a model to trace flow paths for flow visualization, contaminant migration prediction, etc. The following sections describe the implementation and functionality of particle features.

10.1. Adding Particles

> **i** Unlike other features, adding particles does not require discretizing the model before running it (see **Chapter 12**).

There are four different particle features that the user can implement in the Working Area:

- A single particle
- A zone of particles
- Particles along a polyline, and
- Particles around wells.

These features are discussed in the following subsections.

10.1.1. Single Particle

The first step in defining a solitary particle is clicking the 'Add a Single Particle' button (Button Palette-row 4, column 1). This puts the cursor in 'draw mode'. The user may now place a single particle in the Working Area by simply clicking the mouse at the desired point.

Alternately, the user may type in the coordinates for the particle (in the VCI – see **Section 3.13**) instead of clicking the mouse at each location. This method is not limited by the resolution of the screen/mouse relationship and allows for a more precise placement of single particle features.

When a particle is defined in the software it becomes the active feature. A graphic is displayed, showing particle properties as seen below in **Figure 10-1**.

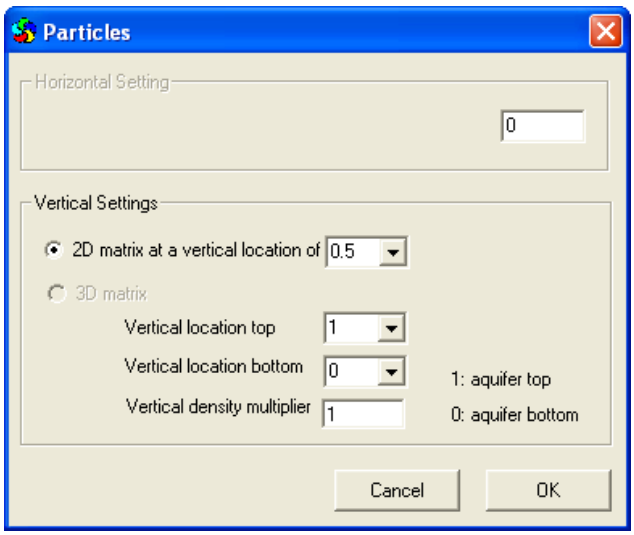


Figure 10-1 Particle Properties

'Horizontal Settings' and '3D matrix' are not applicable to single partilcles.

Vertical Settings area is shown in **Figure 10-2**. '2D matrix at a vertical location of' allows the user to specify the exact release point within the thickness of a given layer by using the pull-down box shown in **Figure 10-2**. By default, the location to release the particles is halfway down the layer (value of 0.5, where 1 is the aquifer top and 0 is the bottom of the aquifer).

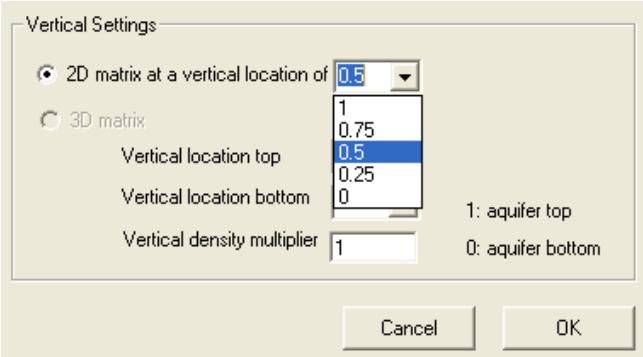


Figure 10-2 Vertical Definition Settings for Particles

At this point, the cursor is still in 'draw mode' and the user may continue to add particles as desired.

10.1.2. Particle Zone

The first step in defining a zone of particles[18] is clicking the '**Add Particles Inside a Polygon**' button (Button Palette-row 4, column 2). This puts the cursor in 'draw mode'. The user may now draw a zone (same methodology as defining a zone – **Section 7.1**) in the Working Area.

As soon as the user finishes drawing the zone, the 'Particles' window appears (see **Figure 10-3**), with the same features as the single particles in **Section 10.1.1**, except that now the 'Horizontal Setting' and '3D matrix' options are active.

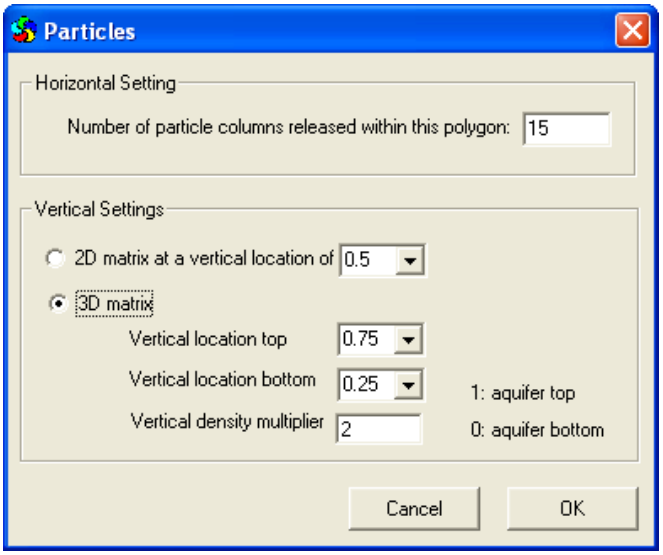


Figure 10-3 Particle Properties

[18] Refer to **Section 8.1** of the ***IGW Version 5.0P Tutorials*** document for a short example of defining a particle zone.

In the Horizontal Setting, the user can specify how many particles per row will be released in the designated region of the model by entering a number in the '**Number of Particles Columns Released within This Polygon**' field. The default value is 15.

When 3D options is selected in Vertical Setting, the user can specify 'Vertical location top' and 'Vertical location bottom' from the drop down options to specify where to release the particles in the three-dimensional realm. 'Vertical density multiplier' allows the user to make changes in particle density in vertical direction. Default multiplier is 1.

Figure 10-4 shows particles added in a polygon in the second layer of a model. A model cross-section across the particles is also shown in the figure illustrating the particles in a 3D realm (more on cross-sections is given in **Chapter 16**).

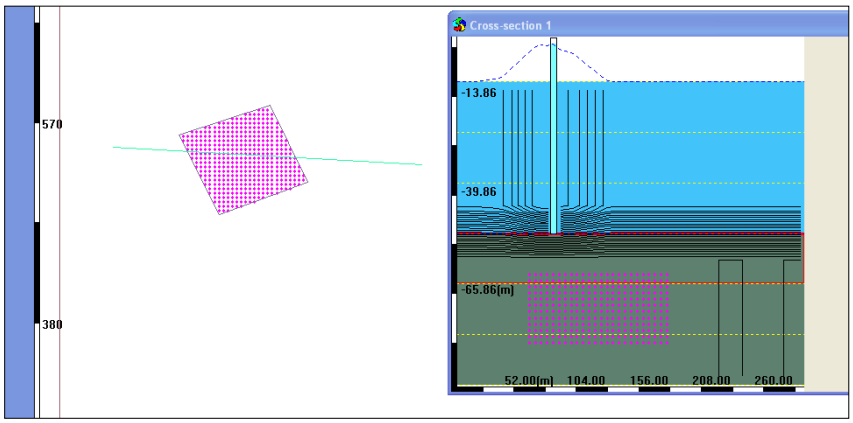


Figure 10-4 Particle added in 3D realm of a model

After a particle zone has been created and particle window closed, the cursor turns back into 'draw mode' and the user may continue to add particle zones as desired.

A particle zone may be redefined in the same fashion as any other zones, using the button at Button Palette-row 11, column 3 (see **Section 7.3**). After a particle zone is redefined be sure to click the 'Initializing Particles' button to completely fill the new zone shape with particles.

A particle zone may be selected in the same fashion as any other zone (see **Section 7.2**).

10.1.3. Particles Along a Polyline

The first step in defining a polyline of particles is clicking the 'Add Particles Along a Polyline' button (Button Palette-row 4, column 3). This puts the cursor in 'draw mode'. The user may now draw a particle polyline (same methodology as defining a polyline – **Section 8.1**) in the Working Area.

After double-clicking (or typing the end command in the VCI) to finish drawing the particle polyline, the 'Particles' window appears (see **Figure 10-5**). This window is identical to the ones discussed in above sections, with one difference: The default number of particles to be released is now 30 instead of 15.

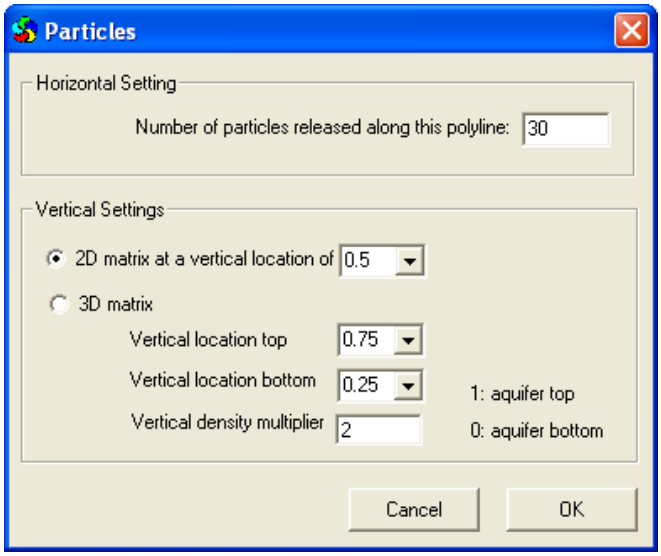


Figure 10-5 Particles Along a Polyline

The user enters a value, or accepts the default value, and clicks 'OK' to create the particle group, or 'Cancel' to abort the procedure. When a polyline of particles is defined in the software it becomes the active feature.

The total number of desired particles is then spaced evenly over the entire polyline length.

When '3D matrix' option is used for polyline, it creates a 'curtain' of particles in 3D realm.

After a particle polyline has been created, the cursor is still in 'draw mode' and the user may continue to add particle polylines as desired.

A particle polyline may be redefined in the same fashion as any other polyline (see **Section 8.3**). After a particle polyline is redefined be sure to click the 'Initializing Particles' button to completely fill the new polyline shape with particles.

A particle polyline may be selected in the same fashion as any other polyline (see **Section 8.2**).

10.1.4. Particles Around a Well

The first step in adding particles around wells is clicking the 'Add Particles Around Wells' button (Button Palette-row 4, column 4). This opens the 'Add Particles To Wells' window (see **Figure 10-6**).

The window shows a list of wells currently in the model (if no wells are present, the user will not be able to add any particles using this interface). The window also shows the wells which already have particles.

The user may select wells individually by clicking in the corresponding box, select them all by clicking the 'All' button or clear all checked boxes by clicking the 'None' button, and the user can also enter the number of particles to add to each well. The default particle setting is 40.

In addition, IGW Version 5.0P allows for vertical settings of these particles (identical to those in previous sections). Using '3D option' will create a 'cylinder' of particles around the well in a 3D realm. The user can also designate the radius from the well that these particles are released (the default is 10, default units in meters (m) with feet (ft), centimeters (cm), inches (in), kilometers (km) and miles (mi) as additional measurement options).

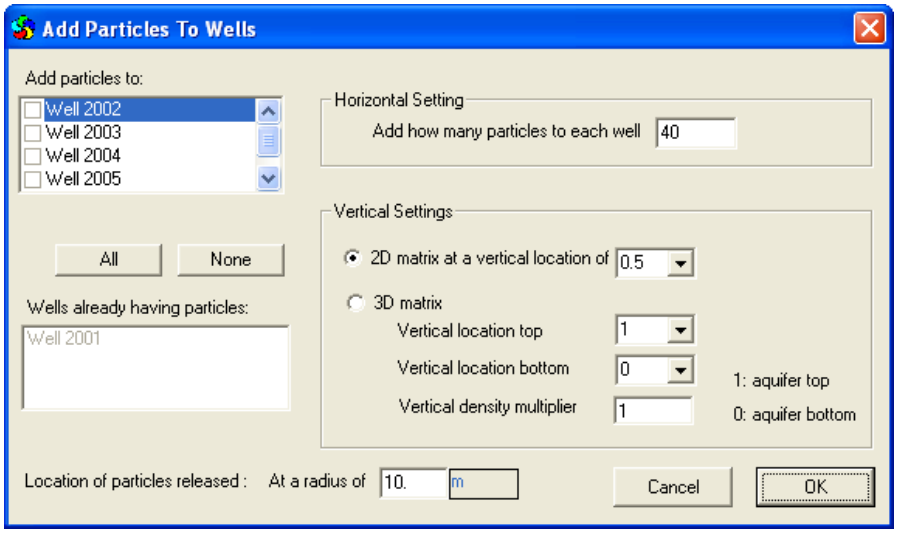


Figure 10-6 Adding Particles to Wells

Once the proper selections have been made, click 'OK' to add the particles or 'Cancel' to abort. When particles are added around wells the particle group associated with the selected well lowest on the list (in the window) becomes the active feature.

R Adding particles around wells and then performing backward particle tracking is a very useful way to delineate well capture zones.

10.2. Setting Particle Attributes

Particle feature attributes are set in the **AE** (see **Section 4.1.2** under the heading **Particle Groups**). After accessing the **AE**, the first step is to expand (if it is not already) the desired particle feature subgroup under the **ParticlesGroup** placeholder in the **LHP** (see **Section 4.1.1**). There are four subgroups, one for each type of particle feature. The **RHP** (see **Section 4.1.2**) is the same for each entry within these subgroups and these are discussed in the following subsections.

R While for advection (linear average groundwater velocity) changing the number of particles does not affect the simulation results, if the user assigns dispersion to the groundwater model, then it is recommended to increase the number of particles in order to capture the variability of groundwater velocity.

It is also recommended to use smaller time steps in order to get more representative results.

! The user can ONLY delete all of the particles after discretizing the model, using **Delete All Particles** button. 

10.2.1. RHP for a Single Particle

A sample of the RHP for a single particle is shown in **Figure 10-7**.

The user has the option of displaying the particle at a single location at each point in time ('Show as Discrete Particles') or displaying the entire pathline the particle has traveled ('Show as

Continuous Pathlines'- the default). The vertical location may also be changed at any time, in the manner of **Section 10.1.1**.

The user can change the number in the 'Size' field to adjust the display size of the particle (default is 12 pixels).

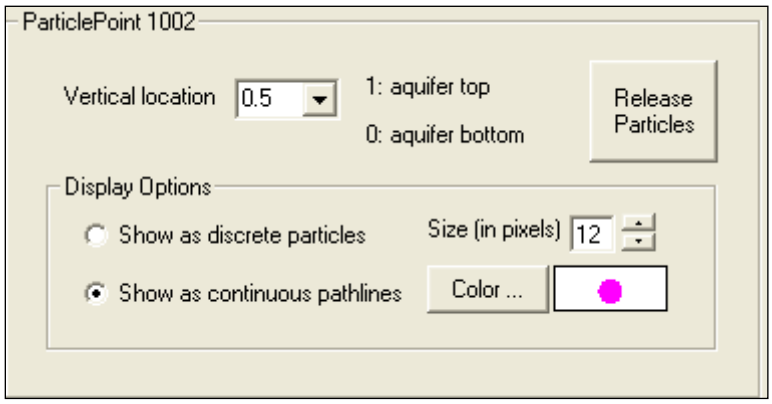


Figure 10-7 RHP for a Single Particle

The user may also click the  button (or the sample particle point box next to it) to open the 'Color' window and subsequently select a new color for the particle. The default color is pink.

To release the particles after reconfiguration, the user can select the 'Release Particles' button.

10.2.2. RHP for a Particle Zone

A sample of the RHP for a particle zone is shown in **Figure 10-8**.

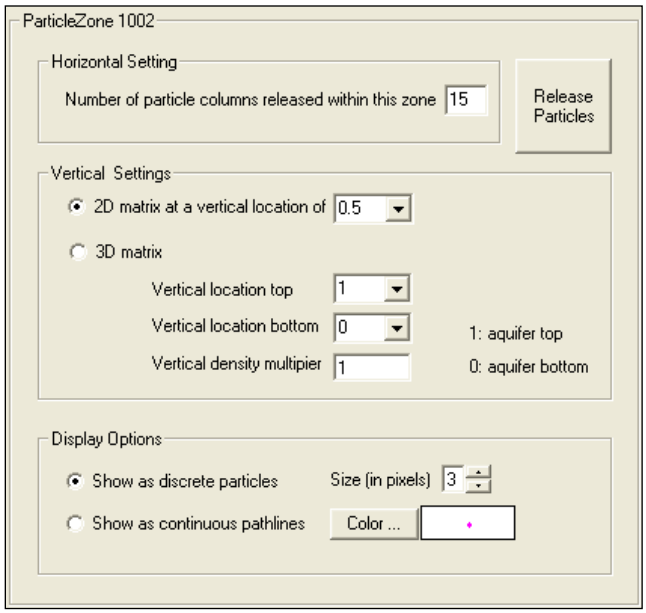


Figure 10-8 RHP for a Particle Zone

The user may adjust the number of particle columns by entering a number in the appropriate field, and also releasing the particles by selecting that button.

The user has the option of displaying each particle at a single location at each point in time ('Discrete Particles' – the default) or displaying the entire pathline each particle has traveled ('Continuous Pathline').

Vertical aspects of the particle location can be defined in the same manner, using the appropriate boxes as described in **Section 10.1.2.**

The user can change the number in the 'Draw Width' field to adjust the display size of the particles in the zone (default is 3 pixels).

The user may also click the 'Color…' button (or the sample particle point box next to it) to open the 'Color' window and subsequently select a new color for the particles. The default color is pink.

10.2.3. RHP for Particles Along a Polyline

A sample **RHP** for the particles along a polyline is shown in **Figure 10-9**.

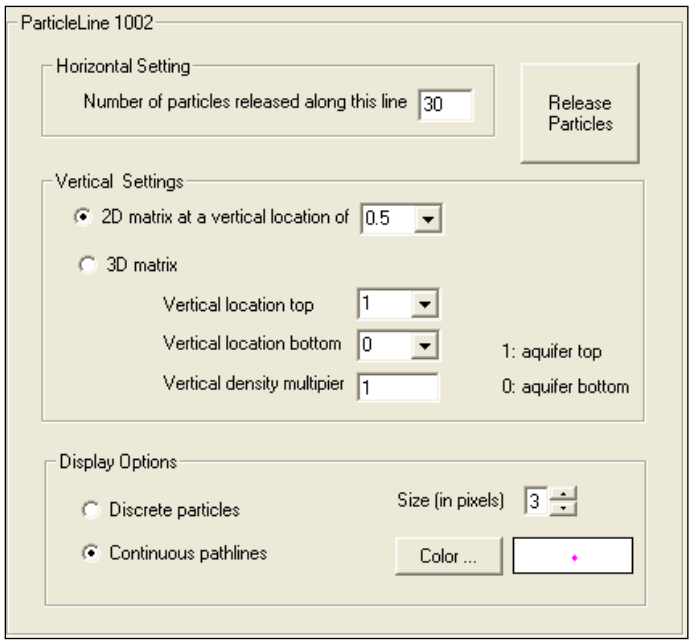


Figure 10-9 RHP for Particles Along a Polyline

The user may adjust the number of particles along the line by entering a number in the appropriate field.

The user has the option of displaying each particle at a single location at each point in time ('Discrete Particles') or displaying the entire pathline each particle has traveled ('Continuous Pathline' – the default).

The user can change the number in the 'Draw Width' field to adjust the display size of the particles along the line (default is 3 pixels).

Vertical aspects of the particle location can be defined in the same manner, using the appropriate boxes as described in **Section 10.1.3.**

The user may also click the 'Color…' button (or the sample particle point box next to it) to open the 'Color' window and subsequently select a new color for the particles. The default color is pink.

R Using particles along a polyline is very useful for delineating a family of streamlines.

10.2.4. RHP for Particles Around a Well

A sample of the RHP for particles around a well is shown in **Figure 10-10**.

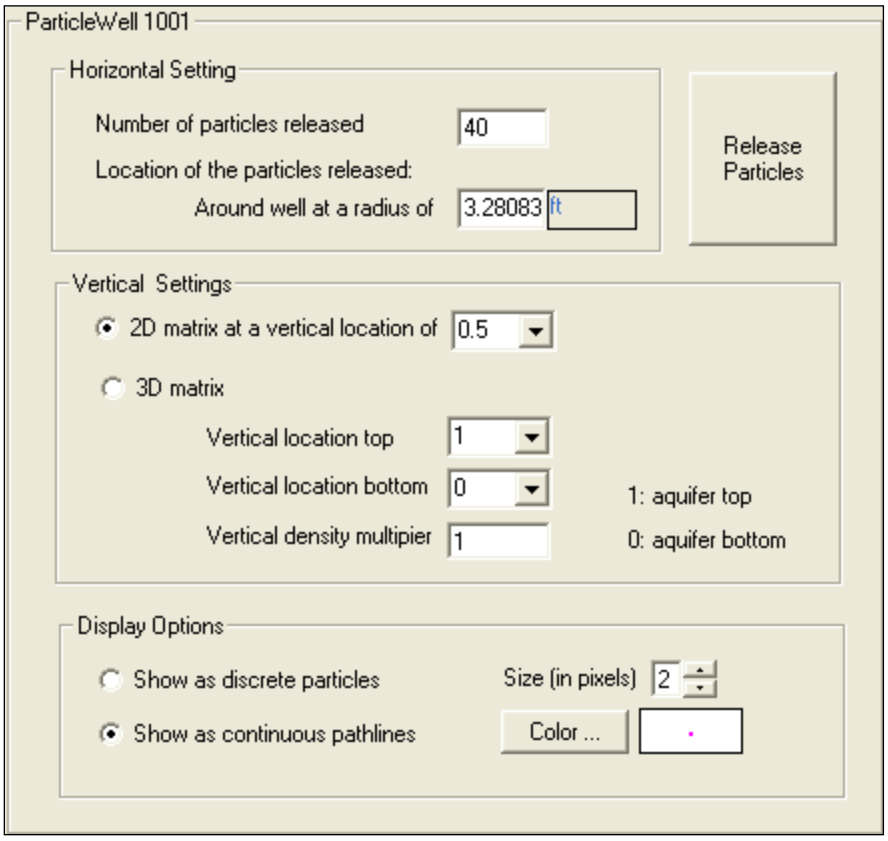


Figure 10-10 RHP for Particle Wells

The user may adjust the number of particles to be released around the well by entering a number in the appropriate field.

i The user may also change the radial distance from the well around which the particles are released, by entering a number in the appropriate field and clicking the 'Release Particles' button. This will update the display in the model for the next iteration.

The user has the option of displaying each particle at a single location at each point in time ('Discrete Particles') or displaying the entire pathline each particle has traveled ('Continuous Pathline' – the default).

Vertical aspects of the particle location can be defined in the same manner, using the appropriate boxes as described in **Section 10.1.4.**

The user can change the number in the 'Draw Width' field to adjust the display size of the particles (default is 2 pixels).

The user may also click the 'Color…' button, or the sample particle point box next to it, to open the 'Color' window and select a new color for the particles. The default color is pink.

10.3. Particle Tracking

IGW Version 5.0P allows the user to perform traditional particle tracking in both forward and backward modes. The implementation of particle tracking is discussed in the following subsections. Refer to the ***IGW Version 5.0P Reference Manual*** for information concerning the mathematics.

R When particles are near wells (for example, when backward particle tracking is implemented after particles have been added around wells) it is best to use small time steps because velocity gradients are high. Sharp gradients and large time steps can yield erroneous particle tracking results. The time step can be adjusted interactively while the model is running by using the SATDI (see **Section 3.5**).

10.3.1. Forward Particle Tracking

Particle tracking[19] is performed whenever the model is run[20] (by clicking the 'Forward' button, located at Button Palette-row 9, column 4 – see **Chapter 14**), and where at least one particle feature is present in the model.

Notice that with particles present, the model solution continuously updates at regular intervals.

The simulation may be stopped at any time by clicking the 'Stop' button.

10.3.2. Backward Particle Tracking

Backward particle tracking[21] is performed whenever the 'Backward Particle Tracking' button is clicked (the button is only active if particles are present in the model).

The model solution continuously updates at regular intervals, with the particles moving in the opposite direction of the velocity vectors.

When backward particle tracking is occurring any plume migration calculations are suspended. Transient flow calculations proceed normally.

Note: The particles may not necessarily return to their original locations when backward particle tracking is performed (this is most pronounced when the simulations have large time steps). This

[19] Refer to **Section 8.1** of the ***IGW Version 5.0P Tutorials*** document for an example and additional information.
[20] **Chapter 14** contains further details about running the model.
[21] Refer to **Section 8.2** of the ***IGW Version 5.0P Tutorials*** document for an example and additional information.

is because for both forward and backward particle tracking, the velocity used for the particle movement is always taken to be the velocity at the current particle location. For example, if a particle moves from location A to B to C to D in 3 consecutive time steps, it will be translated using the velocities at A, B, and C, respectively. But if that same particle is tracked backwards, its first translation will occur using the velocity at its present location, D. If small enough time steps are used, the particle may return to location C. But, it may also be the case that the particle ends up at some different location, C′. The same goes for all subsequent time steps. Therefore, over many time steps, the particle has a good chance of ending up at a different location.

10.4. Initializing Particles

Clicking the 'Initializing Particles' button returns all of the particles in the model to their original locations.

The particle time clock may also be reset to zero by clicking the 'Reset Particle Clock' button to the left of the 'Initializing Particles' button.

10.5. Deleting Particles

All of the particles in the model may be deleted by clicking the 'Delete All Particles' button.

Chapter 11 SIMULATION TIME PARAMETERS

IGW Version 5.0P offers the user a number of options when adjusting time parameters[22] for software simulations. These parameters are discussed in this chapter.

11.1. Simulation Time Parameters Window

The main interface for adjusting the simulation time parameters is the 'Set Simulation Time Parameters' window, pictured in **Figure 11-1**.

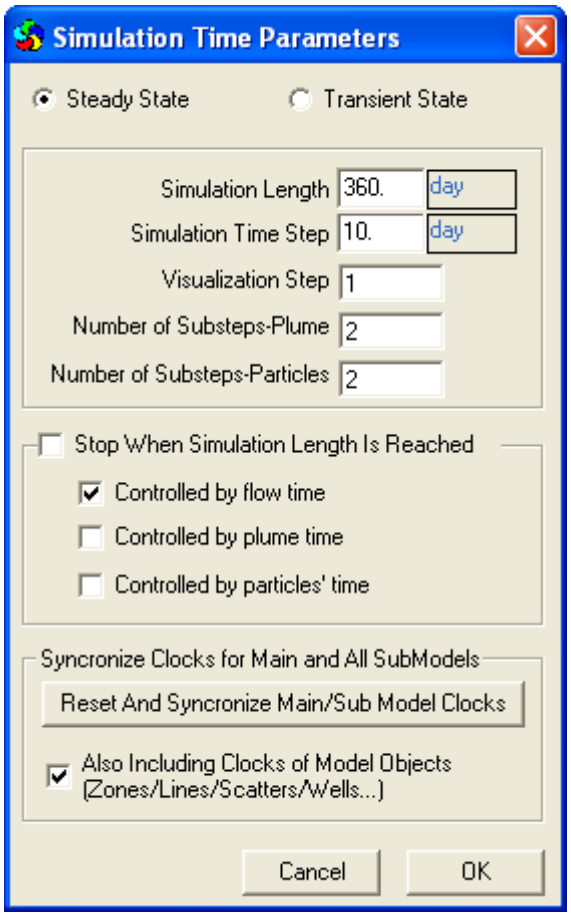


Figure 11-1 Simulation Time Parameters

This window may be accessed through the 'Set Simulation Time Parameters' button on the Button Palette or through the 'Main Model' entry in the AE ('Steady State vs. Transient State…' button). Where indicated (*) these parameters are also adjustable in the SATDI (see **Section 3.5**).

At the top of the window, the user selects either 'Steady State' or 'Transient State', depending upon the model conditions to be simulated.

The other parameters in the window are discussed in the following subsections. Clicking the 'OK' button sets any changes into the software while clicking the 'Cancel' button closes the 'Simulation Time Parameters' window discarding changes (after a verification prompt from the software).

[22] Refer to **Chapter 16** of the ***IGW Version 5.0P Tutorials*** document for additional information about adjusting settings in this window.

SIMULATION LENGTH

Here, the user specifies the desired length of the simulation in the field. The default value/entry is 360 days with default unit as day (hour, second [sec], month, and year also available). This value is only functional when the 'Stop when simulation length is reached' box (near the bottom of the window) is checked (see the **Stop when simulation length is reached** section).

SIMULATION TIME STEP*

This value indicates the current model time step. The default value/entry is 10 with the default unit as day (hour, second [sec], month, and year also available). The user may adjust the time step by entering a desired value in the appropriate field.

VISUALIZATION STEP*

The 'Visualization Step' determines how often the software redraws the Working Area and other windows. A value of 1 (default) indicates that the software will redraw the Working Area after every time step. A value of 2 indicates every other time step, and so on.

NUMBER OF SUBSTEPS-PLUME

This parameter specifies how often the software should perform calculations for concentration plumes. A value of 2 (default) indicates that the software will perform a concentration plume calculation at every other time step. A value of 1 indicates the calculation will be performed every time step, and so on.

R	This setting is not applicable when using the MMOC solver method in IGW Version 5.0P.

NUMBER OF SUBSTEPS-PARTICLES

This parameter specifies how often the software should perform calculations for particle tracking (see **Section 10.3**). A value of 2 (default) indicates that the software will perform a particle tracking calculation at every other time step. A value of 1 indicates the calculation will be performed every time step, and so on.

STOP WHEN SIMULATION LENGTH IS REACHED

Checking this box instructs the software to automatically stop the simulation when the **Simulation Length** value has been reached. In this case, a window will appear indicating that the simulation length has been reached. In the window the user is given the option of continuing the simulation.

If this button is unchecked, the software will run the simulation indefinitely (or until stopped by the user).

This can now also be controlled by selecting if the simulation length is controlled by 'Flow time', 'Plume time', or 'Particles time'.

Further, since *IGW Version 5.0P* has the ability to incorporate sub-models (Error! Reference ource not found.), there is a feature to synchronize all clocks for every model, including the main model and all of its sub-models. This allows the user to run every model simultaneously from the starting time.

11.2. Setting Time-steps for Plume, Particles and Display

SATDI area shown in *IGW Version 5.0P* main window (**Figure 3-1**) provides interface to the user where he/she can adjust the time steps for various parameters. SADTI area is shown in **Figure 11-2**.

The SATDI provides quick access to computational and display adjustments. The time step (DT, 10 by default) can be adjusted using the up/down buttons next to the unit selection field (in days,

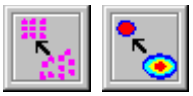
by default). The 'Plume Step', 'Particle Step', and 'Visual Step' (in the bottom area of the SATDI) can be set in a similar fashion, as a ratio of the DT.

Also within the SATDI area three time display fields. These display fields show the time elapsed since the model flow began, since the plume started and since the particles started.

The user can stop the flow simulations at any time by clicking on the '**Stop/Pause Model**' button, reset the time step, and/or particle step, and/or plume step, and/or visualization step.

After making these adjustments, the user can resume simulation by clicking on the '**Run Model Forward**' button

Similarly, after stopping/pausing the simulations, the user can reset the plume(s) and/or particles by clicking on initialize particle(s) and/or initialize plume buttons.



Finally, after stopping/pausing the simulations, the user can reset any or all of the time clocks by clicking on the appropriate buttons for flow clock, concentration clock (plume) and/or particle clock..

The 'Flow Time', 'Plume Time', and 'Particle Time' sections display the computational time for the flow, plume, and particles, respectively. Having three separate clocks allows the user to track these model components separately and becomes very useful when model features are added during a simulation or when the clocks associated with certain features are reset to zero (the other clocks remain unaffected).

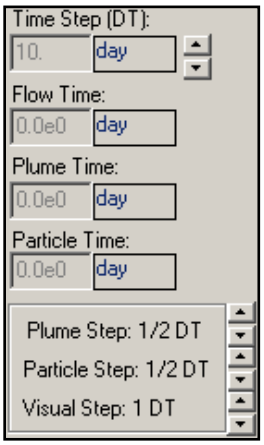


Figure 11-2 The SATDI

The ability to change the time step during the model simulations and adjusting the frequency of the plume and particle calculations is very useful, as the characteristic time scales between these transport processes and flow may often be different.

Typically, performing multiple transport calculations per time step yields a more accurate solution although it will increase computational time. However, since *IGW Version 5.0P* provides the user with real-time dynamic visualization of model simulations, the user can decrease the time step when the plume/particles are in a critical location, usually with very high head gradients (e.g., very close to a pumping well) and keep the time step larger when the gradients are flatter.

i It is recommended to use smaller time steps for transport modeling, since concentration in groundwater usually changes at a faster rate than hydraulic head changes.

IGW Version 5.0P also provides the user a choice for visualization time step. Since much of the software computational time may be spent refreshing the Working Area (see **Section 3.13**) display, adjusting the 'Visual Step' to a higher number (instructing the software to update the display less often) may speed up the solution process.

Chapter 12 GRID OPTIONS AND DISCRETIZING

The following sections describe the IGW Version 5.0P functionality related to adjusting the nodal grid, and discretizing[23] the model features onto it.

12.1. Defining Model Grid and Discretization

The 'Define Model Grid' (deep discretization) window (see **Figure 12-1**) is the main interface for defining model grids within the model. This window may be accessed by clicking the 'Deep Discretization' button on the Button Palette or by clicking the 'Define Model Grid…' button on the 'Main Model' entry in the AE.

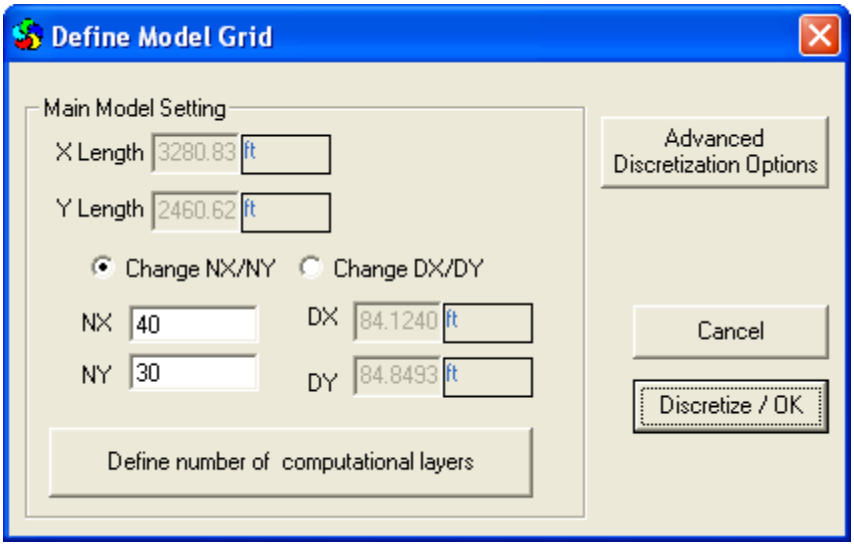


Figure 12-1 Defining Model Grid

The parameters displayed in the window are discussed in the following subsections. Clicking the 'Discretize/OK' button closes the window and discretizes the model (see **Section 12.2**) with the new grid settings. Clicking the 'Cancel' button discards any changes and no discretization is performed.

To define the grid, user can choose between 'Change NX/NY' or 'Change DX/DY' by checking the desired radio button.

X LENGTH
This field displays the conceptual (real-world) x-direction length of the Working Area (see **Section 3.3.5** for available units).

Y LENGTH
This field displays the conceptual (real-world) y-direction length of the Working Area (see **Section 3.3.5** for available units).

Change NX/NY
When 'Change NX/NY' is selected, user can directly define the number of grid cells in the X and Y directions of the model.

[23] Refer to **Section 4.1** of the ***IGW Version 5.0P Tutorials*** document for an interactive exploration of these concepts.

NX field displays the number of cell-center nodes in the x-direction. The first column of nodes are placed on the left most edge of the Working Area and are evenly spaced across the Working Area in the x-direction. The user may enter the desired number in this field. The default value/entry is 40. Entering a value for NX automatically causes the **NY** value to update (the NY value will update in proportion to the dimensions of the Working Area by making an approximately square grid cell).

NY field displays the number of cell-center nodes in the y-direction. The first row of nodes are placed on the bottom most edge of the Working Area and are evenly spaced across the Working Area in the y-direction. Although **NY** field is updated with **NX** field, but user may change it independent of the **NX** value. A number largely different from the automatically adjusted one will make rectangular grid cells.

When Change NX/NY is selected, the values in fields **DX** and **DY** are automatically updated based on the following formulae:

DX field displays the x-direction extent of each node-centered cell in the Working Area. This number is calculated as:

$$DX = \frac{XLENGTH}{NX - 1}$$

Where XLENGTH=total length of X axis of the modeling domain.

DY field displays the y-direction extent of each node-centered cell in the Working Area. This number is calculated as:

$$DY = \frac{YLENGTH}{NY - 1}$$

Where YLENGTH=total length of Y axis of the modeling domain.

Change DX/DY

When 'Change DX/DY' option is selected, the user can directly input the desired length of grid cells **DX** and **DY** in X and Y directions respectively. The software automatically calculates the **NX** and **NY** using the above mentioned formulae.

Entering a value for **DX** automatically causes the **DY** value to update (the **DY** value will update in proportion to the dimensions of the Working Area by making an approximately square grid cell). Although **DY** field is updated with **DX** field, but user may change it independent of the **DX** value. A number largely different from the automatically adjusted one will make rectangular grid cells.

> **i** Discretizing the model turns the conceptual model into a numerical model that the computer can solve.

> **i** A higher grid resolution yields a more accurate solution but increases computational time. It is generally good practice to perform sensitivity analyses with respect to grid resolution and time step size.

12.2. Defining Geological and Computational Layers

The user can add multiple geological (or conceptual) layers in the model. IGW Version 5.0P also allows the user for vertical discretization of geological layers into desired number of computational layers. The following subsections explain how geological and computational layers can be added to in a model.

12.2.1. Adding Geological Layers in the Model

The user can create multiple geological layers in the model by selecting **File** menu in the **Menu Bar** and then selecting **Create New Model Layer**. User can add one layer at a time using this option.

When a model domain is defined by a zone in a layer, new layers can be created using the same domain. User can select the zone defining the model domain and then press "Alt+L" on the keyboard. 'Add new layer(s)' window appears (**Figure 12-2**). The user can decide on the number of layers to be added. This option allows the user to add multiple layers at once in the model.

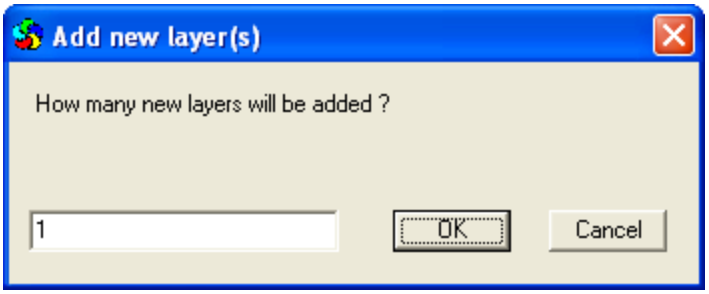


Figure 12-2 Add new layer(s) window

The desired number of new layers in the model are added below the existing layers by default. However, before adding the layers in the model, 'Set Layers Position' window pops up as shown in **Figure 12-3**. Using the up and down arrows, user can adjust the location of the new layer(s) between the existing layers.

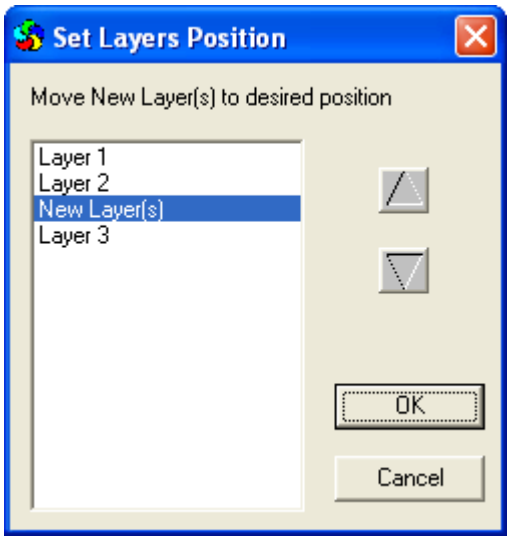


Figure 12-3 Set Layers Position window

By clicking OK in 'Set Layers Position' window, a message window pops up (**Figure 12-4**).

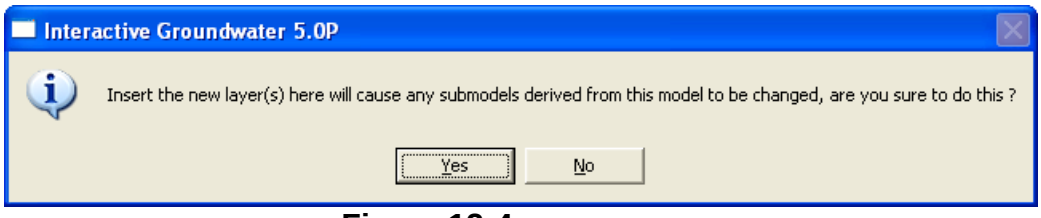


Figure 12-4 Message window

By clicking 'Yes' in the message window, the software adds the decided number of layers in the model at the selected positions.

12.2.2. Adding Computational Layers in Geological Layers

The user can also create multiple computational layers by clicking on **Define Number of Computational Layers** button in 'Define Model Grid' window (**Figure 12-1**).

Clicking the **Define number of computational layers** button brings up the **Vertical Discretization** window. Here, the user can click on the desired geological layer and create multiple vertical computational layers in it. Also, same number of computational layers can be created in all of the geological layers by simply checking the **Apply to All Layers** box (**Figure 12-5**).

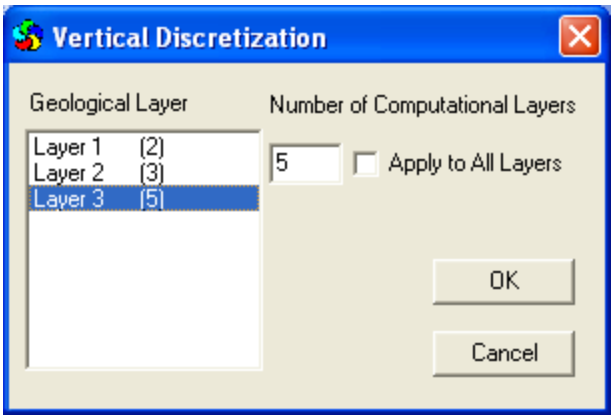


Figure 12-5 Vertical Discretization Window

The user can create multiple modeling zones of different sizes within different geological layers. **Figure 12-6** demonstrates a cross-section (see **Chapter 16** for detailed information on cross-sections) along the **Working Area**, after discretizing and running a simple model (see **Chapter 12** and **Chapter 14**) with three geological layers and different number of computational layers. **GL #** and **CL #** indicates the number of geological layers and computational layers, respectively. Computational layers are separated by a dashed line within a geological layer. As one can see, dimensions of geological layers are different from each other. Contours demonstrate the groundwater head.

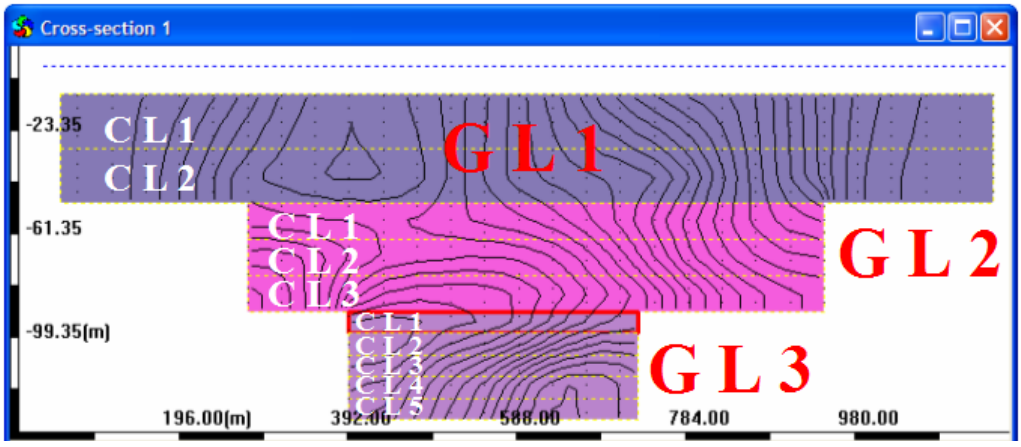


Figure 12-6 Geological layers with different dimensions and different number of computational layers

12.2.3. Navigating between Model Layers

The user can switch between different geological layers using the **Layer Selector** (shown in **Figure 12-7**; also refer to **Section 3.10**) area in IGW Version 5.0P main window. The user can move the sliding button to select the desired layer. The tick marks along the sliding bar correspond to the computational layers. The user can also directly type in the number corresponding to the desired geological and/or computation layer in the fields provided at the top. The selected layer will show up in the working area.

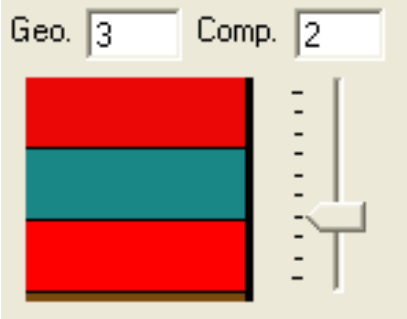


Figure 12-7 Layer Navigation Window

> **i** Another way to switch between different layers is using the layer selection tool at the bottom left of the **Working Area**. 
>

12.3. Advanced Discretization Options

Clicking the 'Advanced Discretization Options' button on 'Define Model Grid' window opens the 'Advanced Options' window as shown in **Figure 12-8**.

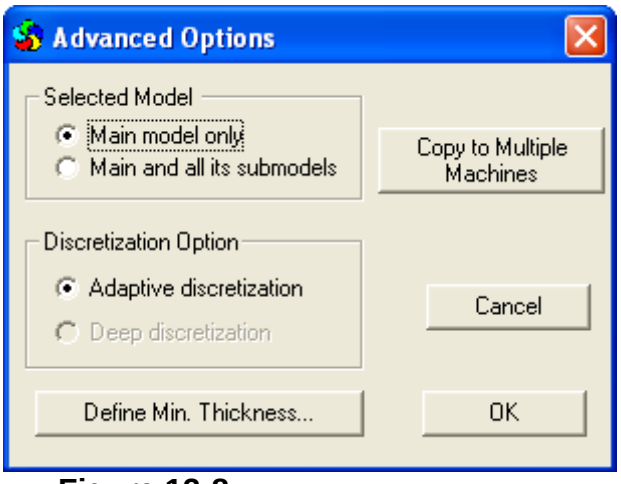


Figure 12-8 Advanced Discretization Options

This window affords the user several options, including the choice to discretize the main model only, or instead discretize the main model and all of its sub-models (see **Chapter 15** for more on submodels).

The option of **Adaptive Discretization** or **Deep Discretization** is shown in **Figure 12-8**, though at this time the deep discretization feature is not offered.

The minimum thickness of a geological layer can also be defined by clicking on the 'Define Min. Thickness…' button in the bottom of the window. This opens a separate screen (**Figure 12-9**) that gives the user options to change these values, based on percentage values of either the conceptual or computational model being used.

> **!** Some of the active model cells may become dry, if groundwater head goes below the bottom elevation of the aquifer. This is typical especially with unconfined aquifers, since the groundwater flow equation is nonlinear, and so convergence problems may occur. **IGW** lets the user to define the minimum thickness of the aquifer in 'Define Min. Thickness' window (see **Figure 12-9**), so they can prevent those particular cells from going dry.

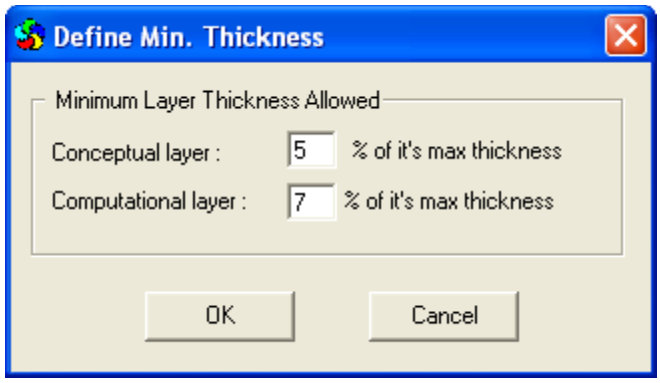


Figure 12-9 Assigning Minimum Aquifer Thickness

Clicking the '**Copy to Multiple Machines**' button opens the 'Parallel Hosts and Tasks' window shown in **Figure 12-10**. Detail functionality of this window is explained in parallel computing features of *IGW Version 5.0P*. Please refer to **Section 18.7** for more details on parallel computing.

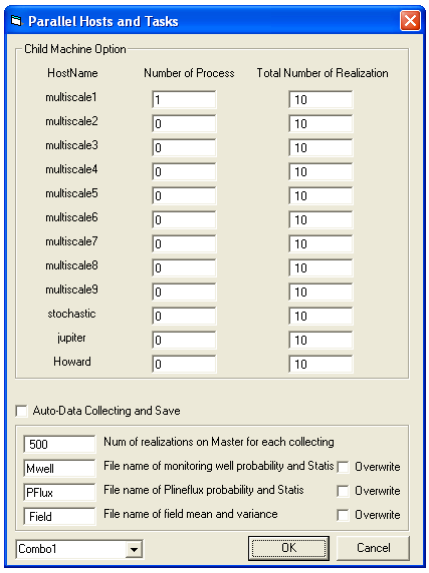


Figure 12-10 Parallel Hosts and Tasks window

12.4. Shallow Discretization of Model

Once a model has been discretized, IGW allows the user to add or delete features/parameters in the model or make changes in any of those. Also, when models are built incrementally, all model features and complexity is not introduced in the model right from start. Every time a new model feature is added, an existing feature deleted or any change made in an existing feature, the model needs to be re-discretized to map the latest change(s) in the model to the numerical grid.

> **!** The model should be discretized every time an addition, deletion, or change is made in any features or settings of the model (however, some features, such as particles, can be executed in the model without discretization).

Re-discretizing the whole model every time a change is made may not be very efficient in terms of computing time, especially in case of more complex models.

IGW Version 5.0P allows the user to implement any additions/changes in conceptual model features to the numerical grid by shallow discretization. Clicking the '**Shallow Discretize**' button discretizes the model incrementally, in order to introduce latest change(s) in the model parameters – **Section 12.1**.

By right clicking any where in the Working Area, the Right-Click Menu pops up (**Section 3.17**). Selecting 'Discretization Table' from this menu opens the **Discretization Table** window as shown in **Figure 12-11**. The window shows, separately for every layer in the model, any attribute which has been modified since the model was last discretized. The checked boxes before the modified parameters are already selected when the window is opened. The user can deselect any of the modified parameters in this window. Shallow discretization will apply only to the parameters selected in this window.

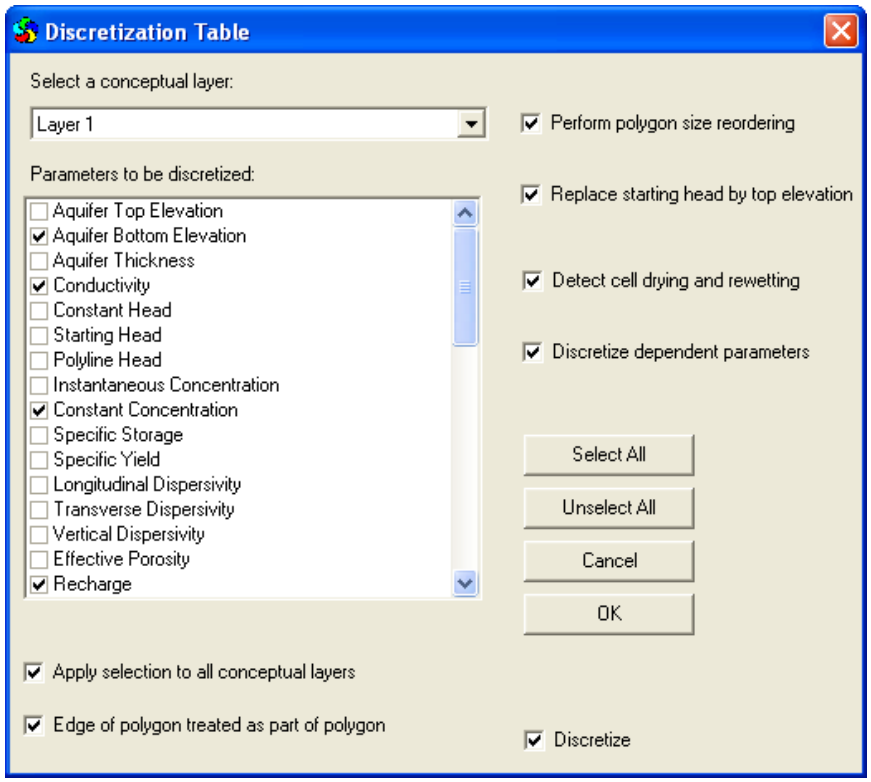


Figure 12-11 Discretization Table window

R Having a look at the Discretization Table before shallow discretization is a good idea. The user can see before hand what all parameters have been modified since last discretization and make up for any omissions at this stage.

! If any changes are made in the grid size or shape and/or any change is made in the number of conceptual or computational layers, shallow discretization will not implement those changes. Any change in model grid and/or layers will require deep discretization.

R For the first discretization of a model, functions of both deep and shallow discretization buttons are the same. After some changes are made in the model using Working Area or Attributes Explorer (e.g. modifying locations and values of modeling features), it is enough to use shallow discretization in order to introduce those changes. This will ONLY implement the incremental changes to the model instead of discretizing the whole model, reducing the computational time.

R If user feels that some changes do not seem to take effect in the model he/she may go for deep discretization to fully implement all changes made.

12.5. Displaying the Grid

By default, the grid is not displayed in the Working Area. The grid line is activated in the 'Main Model - Draw Option' window (see **Section 19.1**) by checking the box next to 'Grid Line' (in the 'Reference Coordinate Visualization' area) and then clicking the 'OK' button.

The grid line is node-centered. This means that a node exists at the intersection of the horizontal and vertical lines. Therefore the corners of each node-centered cell exist at the centers of the grid squares.

Chapter 13 SOLVER ENGINE SETTINGS

The **Solver Engine** (**Figure 13-1**) is the main interface for defining solver options for the software. This window may be accessed by clicking the **Solver Engine** button on the **Button Palette** or by clicking the **Model Solver** button on the **Model** entry in the **AE**.

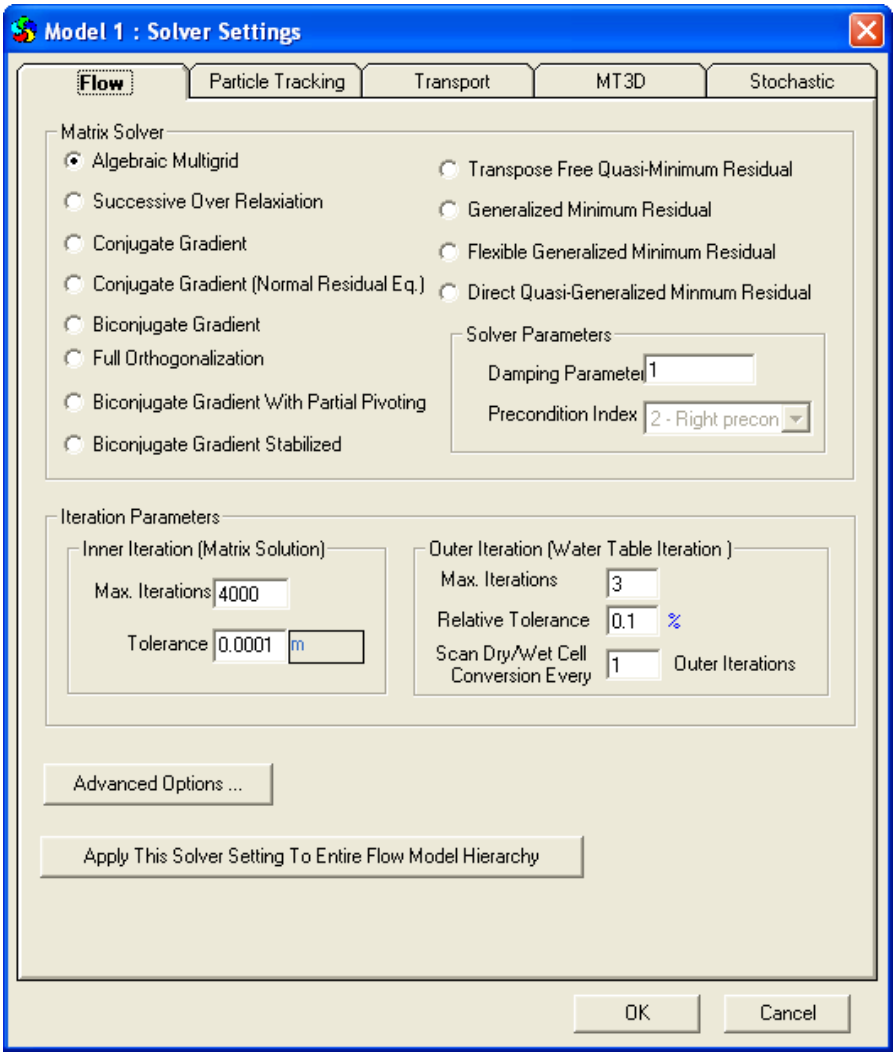


Figure 13-1 The Solver Engine with Flow layer visible

The solver window has five main layers: **Flow**, **Particle Tracking**, **Transport**, **MT3D**, and **Stochastic Model**. The first three settings form the general mathematical solution, **MT3D** allows a transfer into an entirely different groundwater model, and **Stochastic Model** deals with setting the software to perform stochastic modeling. These layers and associated parameters are discussed in the following sections.

13.1. Flow Layer

The **Flow** layer is visible in **Figure 13-1**. It has three main areas that are discussed in the following subsections. The solvers are all iterative in nature: as IGW uses the finite difference solution scheme, these methods compute the head for a cell based upon the heads of the surrounding cells. See the ***IGW Version 4.7 Reference Manual*** for more information concerning the general finite difference solution schemes.

13.1.1. Numerical Solving Methods

MATRIX SOLVER

This region has 12 possible solving methods for the model:

- **Algebraic Multigrid**
- **Transpose Free Quasi-Minimum Residual**
- **Successive Over Relaxation (SOR)**
- **Generalized Minimum Residual**
- **Conjugate Gradient**
- **Flexible Generalized Minimum Residual**
- **Conjugate Gradient (Normal eq.)**
- **Direct Quasi-Generalized Min. Residual**
- **Biconjugate Gradient**
- **Full Orthogonalization**
- **Biconjugate Gradient with Partial Pivoting**
- **Biconjugate Gradient Stabilized**

SOLVER PARAMETERS

The user has the option to adjust the **Damping Parameter** (default set to 1), or the **Precondition Index**, which currently is not being utilized.

13.1.2. Iteration Parameters

INNER ITERATION (MATRIX SOLUTION) AREA

In the **Inner Iteration (Matrix Solution)** area, the user specifies the maximum number of iterations to be performed (in the **Max. Iterations** field – 4000 by default) and the minimum absolute error allowed between iterations (in the **Tolerance** field - 0.0001 meters by default) before the solution is considered 'converged' (see **Section 13.3**).

OUTER ITERATION (WATER TABLE ITERATION) AREA

In situations where there is a variable water table the software must also perform a water table (outer) iteration for every inner matrix solution. This is due to the fact that flow is dependent on the transmissivity, which in turn is dependent on the unknown head values being calculated in the inner iteration.

The settings for this scenario are in the **Outer Iteration (Water Table Iteration)** area. The user specifies the maximum number of iterations (in the **Max. Iterations** field – 3 by default) and the relative tolerance allowed between iterations (in the **Relative Tolerance** field – 0.1 % by default) before the solution is considered 'converged' (see **Section 13.3**).

The user also specifies how often the software checks to determine the dryness / wetness criteria of each cell. The default value/entry, 1, indicates that the software will perform the check during every outer iteration. Higher numbers instruct the software to perform the check less often.

! Iterative solver techniques require a threshold 'error' value to determine when the solution has converged. When the difference between an iteration and the next one is smaller than this number, the solution is considered 'converged'. Although the software provides a default value for these errors, these may not be appropriate for a given situation. For example, when modeling a situation that involves very flat gradients the error threshold number may need to be decreased to prevent the software from prematurely terminating the solver.

13.1.3. Advanced Options

Clicking the **Advanced Options** button opens the **Advanced Options for Flow Solver** window (**Figure 13-2**). The user chooses between using the **Traditional Finite Difference** scheme, or the **Improved Finite Difference** scheme when anisotropy is not aligned with grid orientation. The software uses the improved scheme by default. There are also four options for running and checking the model. Refer to the ***IGW Version 4.7 Reference Manual*** for more information.

Finally, the user has the option to **Apply This Setting to Entire Flow Model Hierarchy**, giving the model and all of its sub-models the properties defined above.

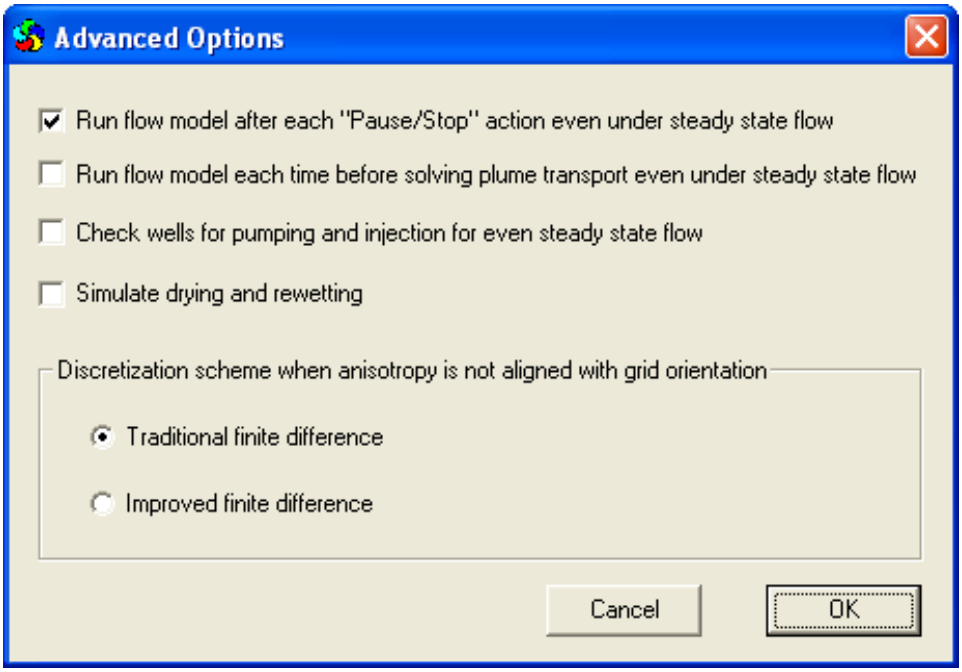


Figure 13-2 Advanced Solver Options

13.2. Particle Tracking Layer

This layer allows the user to manipulate the tracking and grid schemes of the particle display, how to present the particles, and the type of velocity interpolation scheme to use. The layer is shown in **Figure 13-3.**

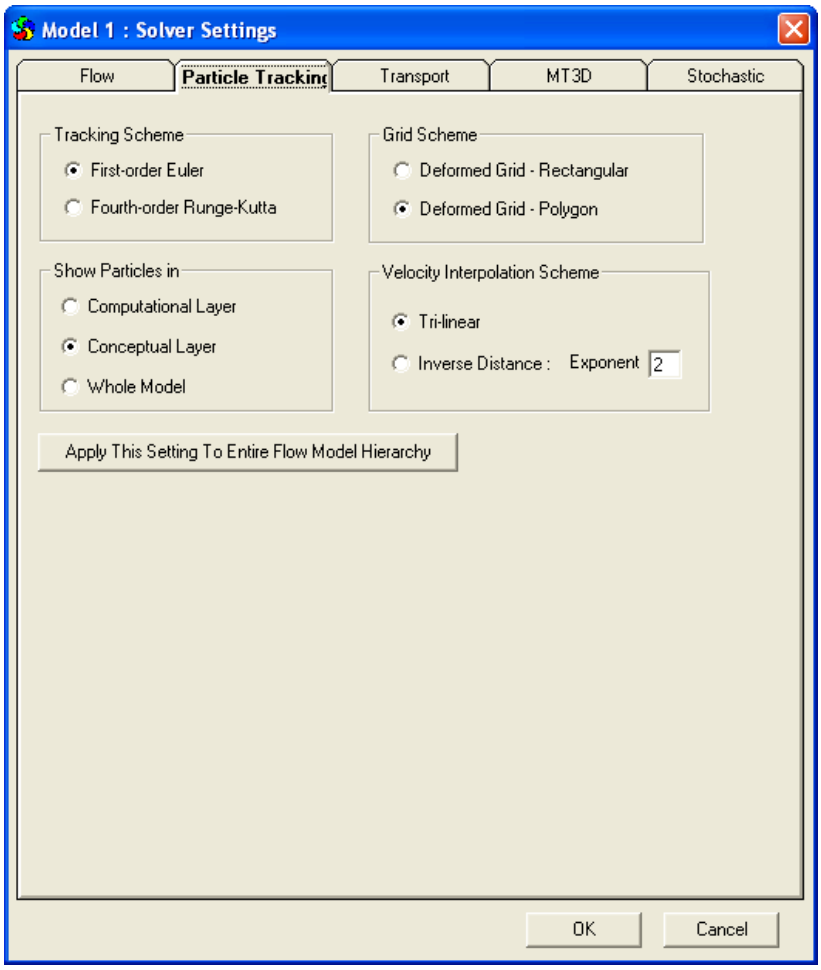


Figure 13-3 Solver Settings for Particle Tracking

13.2.1. Tracking Scheme

The user has two options for the tracking scheme:

1) **First-order Euler (the default)**, and
2) **Fourth-Order Runge-Kutta.**

Explanations for these methods can be found in the ***IGW 4.7 Reference Manual***.

13.2.2. Grid Scheme

Two options are available to change the grid scheme:

1) **Deformed Grid – Rectangular**, and
2) **Deformed Grid – Polygon** (default).

13.2.3. Particle Display Options

Three options are available to the user in terms of particle presentation: Displaying these in the **conceptual model** only, in the **computational model** only, or in the **whole model** (both conceptual and computational). The **conceptual model** is the default choice.

One benefit of allowing the particles to be seen in the conceptual model and not the computational model is the user can iteratively manipulate the particles until they have the desired configuration, and then incorporate it into the numerical model. This would be beneficial to those seeking exploratory analysis of uncertain plumes, and also for sensitivity analysis.

13.2.4. Velocity Interpolation Scheme

There are two ways to manipulate the velocity interpolation scheme:

1) **Tri-Linear** (default), and
2) **Inverse Distance**, in which the user can also change the exponent value.

Explanations for these methods can be found in the ***IGW Version 4.7 Reference Manual***.

13.3. Transport Layer

This layer allows the user to change features relating to the particle transport attributes (**Figure 13-4**).

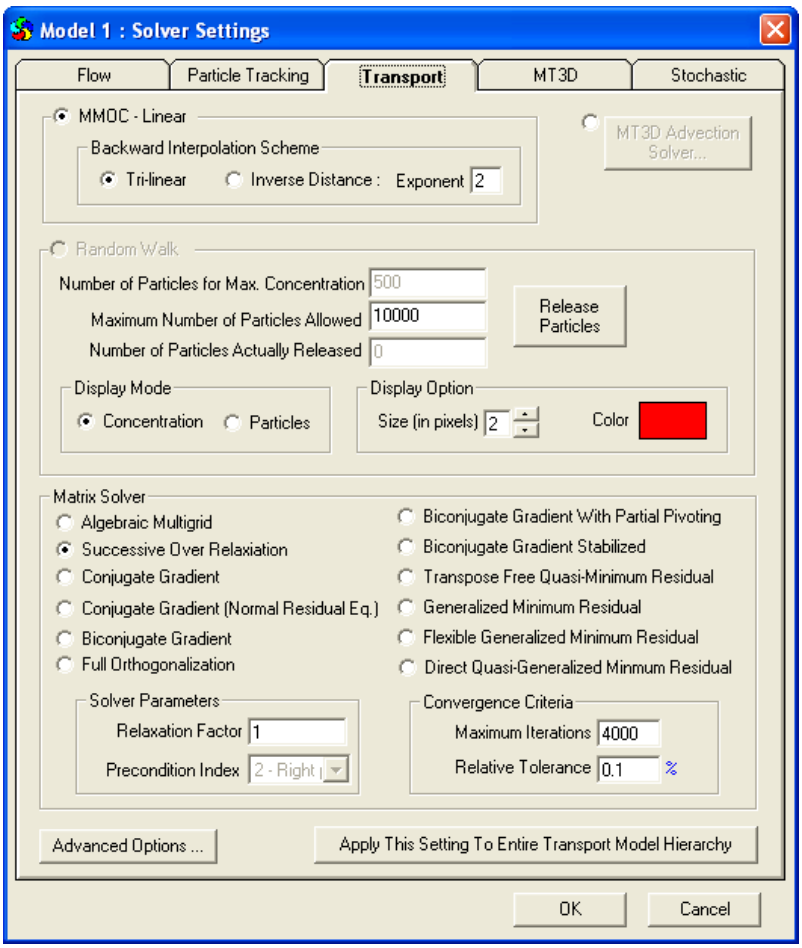


Figure 13-4 Solver Settings for Transport

13.3.1. Modified Method of Characteristics

The Modified Method of Characteristics (MMOC) calculates the concentration at a given node, m (for a given time step, n+1), by employing the following algorithm:

1) A particle at m is traced backwards to time step n, to determine its previous location (p).
2) The concentration at p is determined by interpolation from the surrounding nodes (based on time step n data).
3) This concentration is adjusted for any dispersion and reaction that would occur over the time step, and the resulting value is the concentration at m.

Refer to the ***IGW Version 4.7 Reference Manual*** for further information and mathematical details.

The **MMOC** is beneficial because it requires only one particle for each cell and resets these particles for each time step (so there is no need to store particle location data). These result in decreased computer memory usage.

However, the **MMOC** has an inherent problem known as 'numerical dispersion'. This is due the interpolation portion of the scheme (2^{nd} step in the algorithm, above), and this tends to cause the modeled plume to disperse faster than it would in the real world.

13.3.2. MT3D Advection Solver

The **MT3D Advection Solver** (**Figure 13-5**) gives the user several options for particle tracking, based on the groundwater modeling program **MT3D**, a three-dimensional mass transport simulation interface. For further information on this program and features associated with this layer, please refer to the ***IGW Version 4.7 Reference Manual***.

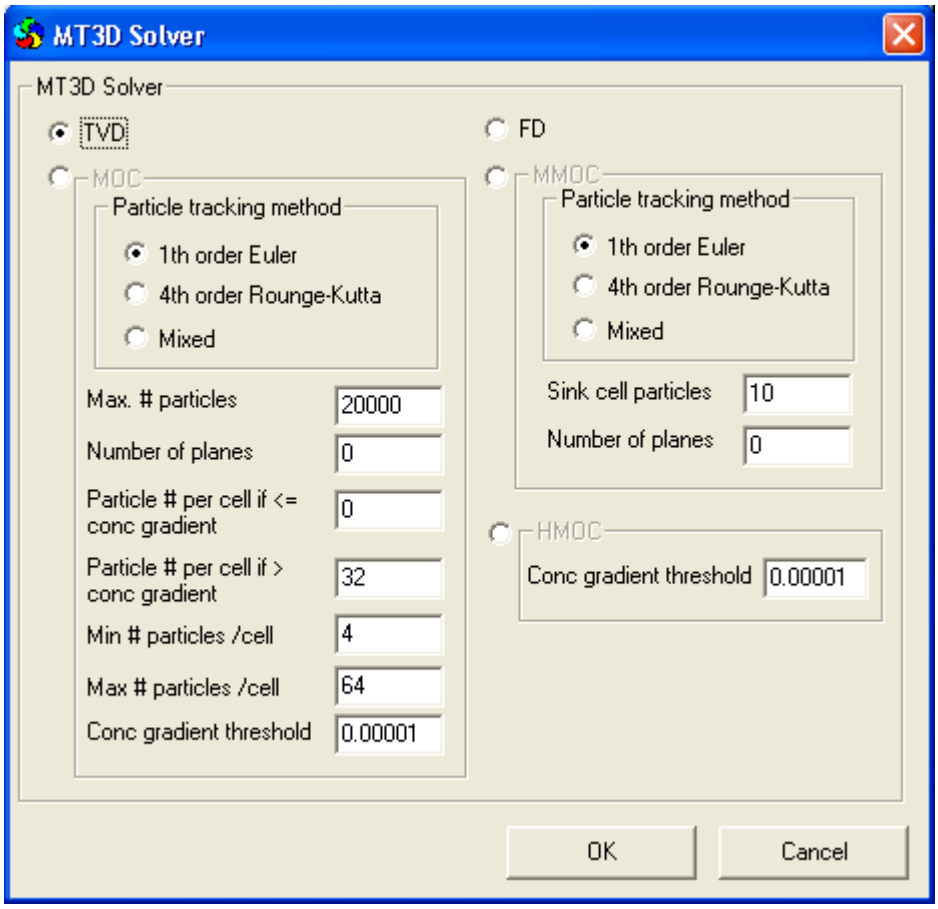


Figure 13-5 **MT3D Solver**

13.3.3. Random Walk

The **Random Walk** transport solver method calculates the concentration for a given cell as a proportion to the number of particles present within it. The following algorithm approximates the entire process:

1) A number of particles are placed in the model and each is assigned the same mass (this is done implicitly by the software based on the number of particles released and the user specified concentration).
2) Each particle is tracked using conventional techniques. Any dispersivity, retardation, and reaction effects are incorporated.
3) The concentration in each cell is determined based on the number particles present within it.

The **Random Walk** method is beneficial because it eliminates all numerical dispersion, conserves mass and is computationally efficient. However, it is problematic in that:

1) A continuous concentration plume is represented by a finite number of discrete particles (therefore the number of particles used influences the results),
2) It becomes less effective as dispersion effects increase, and
3) Numerical difficulties arise in the presence of irregularly-discretized space, or stagnation zones created by sources / sinks.

Refer to the ***IGW Version 4.7 Reference Manual*** for further information and mathematical detail.

RANDOM WALK AREA

In this area, the user may view and specify a number of parameters associated with the random walk solver method. These are listed below:

- **Number of Particles for Max. Concentration**: This field indicates the number of particles that correspond to the maximum concentration in the model. 500 is the default value.
- **Maximum Number of Particles Allowed**: In this field, the user defines the maximum number of particles allowed in the model. The default value is 10,000.
- **Number of Particles Actually Released**: This field displays the number of particles that currently exist in the model (0 by default).
- **Release Particles**: This button allows the user to release the particles into the model at their discretion.

13.3.4. Display Mode

The display mode simply allows the user to determine if they want the contaminant to appear as either particles (i.e. a discreet simulation) or as a concentration plume within the model (i.e. a continuous simulation).

Concentration plumes will show more of history of overall movement of contaminants over time, whereas particles will show an active location for the head of the contamination plume, not recording where the plume has been in previous time steps.

13.3.5. Display Options

This field allows the user to determine the size of their particles on the screen and the color they would like these to appear, using the color button and associated palette.

13.3.6. Matrix Solver

This region is identical to the section discussed in **Section 13.1.1**.

MATRIX SOLUTION CONVERGENCE CRITERIA

In this area, the user may specify the maximum number of iterations to be used in the **MMOC Solver** (in the **Max. Iterations** field). The default value is 4000. The user may also specify the convergence error value (as the change in concentration divided by the original concentration) in the **Relative Tolerance** $\left(\frac{dC}{C}\right)$ field. The default value is 0.1 percent.

13.3.7. Advanced Options

Clicking the **Advanced Options** button opens the **Advanced Options for Transport Solver** window (**Figure 13-6**). In this window, the user may choose to employ the **improved finite difference scheme** (default) or the **Traditional finite difference scheme** for cases when the anisotropy (flow direction) is not aligned with the grid orientation. The user can also choose to **Skip Transport Model (run flow model only)** and thereby only execute the flow model. These schemes are discussed further in the ***IGW Version 4.7 Reference Manual***.

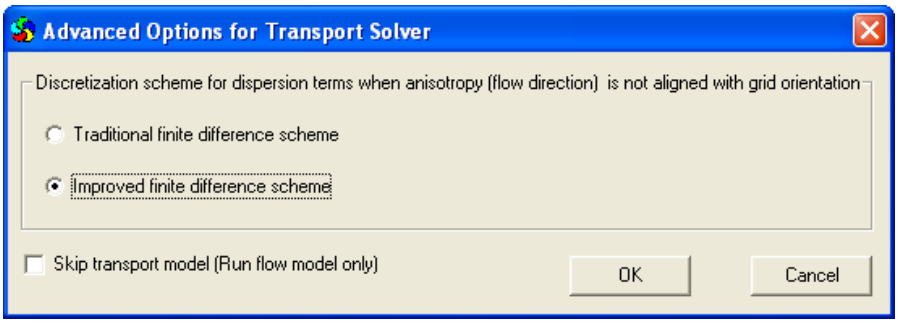


Figure 13-6 Advanced Solver Options for Transport

13.4. MT3D Layer

This layer is currently not active in **IGW** Version 5.0P. For details about the **MT3D** modeling program and its applications in **IGW**, please see the ***IGW Version 4.7 Reference Manual***.

13.5. Stochastic Model Layer

The **Stochastic Model** layer in the Solver Settings can be seen in **Figure 13-7.**

The **Select Simulation Methods** area is used to set the stochastic behavior of the model. By default the model is set to **Single Realization**, meaning that the model will operate in:

1) Deterministic (mean) mode, if no data point simulations are specified (through scatter points or random settings, see **Section 7.7**), or
2) Single realization mode, if at least one data point simulation is specified (in this case only one of all possible statistically equivalent models is analyzed). Selecting **Monte Carlo Simulations** sets the software to solve multiple statistically-equivalent realizations of model parameters. If no data simulations are specified, then the mean model will be solved repeatedly.

Please refer to **Chapter 18** for more information concerning stochastic modeling. **Monte Carlo Simulation** options in this tab are discussed in **Section 18.3**.

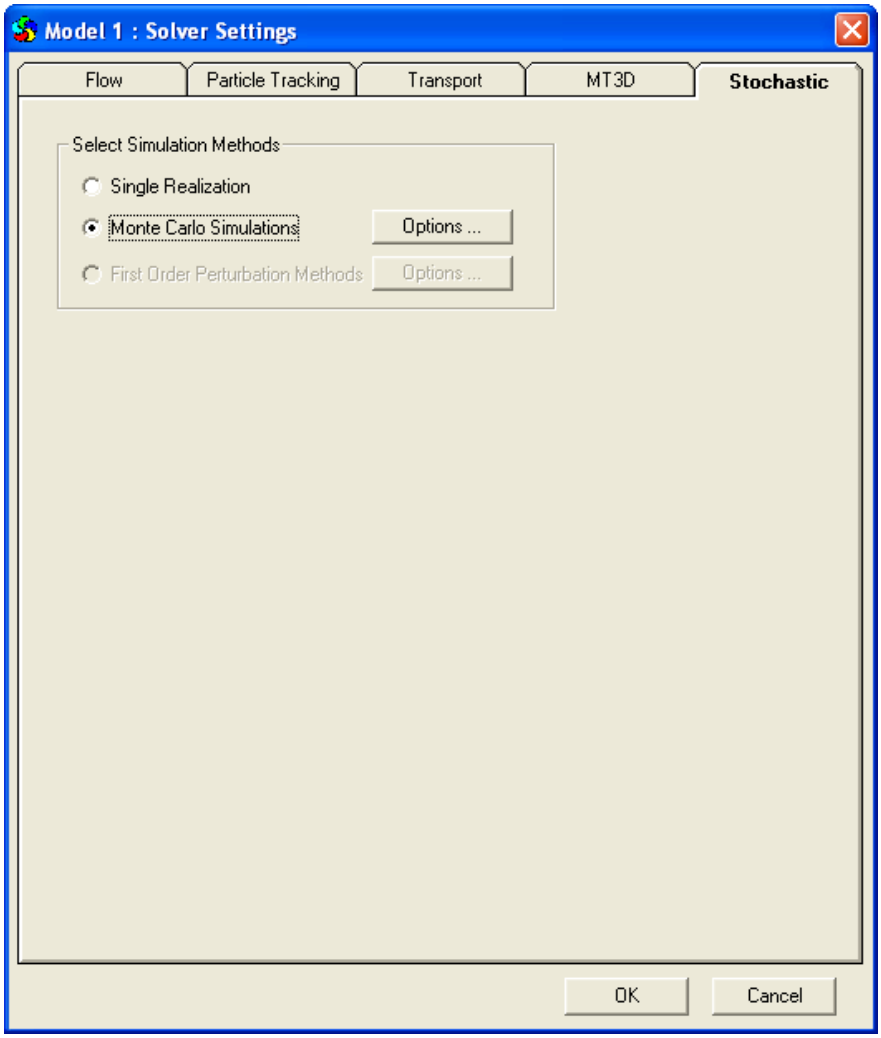


Figure 13-7 Stochastic Model Settings

Chapter 14 RUNNING THE MODEL

This brief chapter describes the basic procedures for running the model and initializing features of the model.

14.1. Running the Model

Running the model requires simply clicking the 'Forward' button.

The model should be discretized before attempting to run it. If not, the software will remind the user to do so with an error window with the message 'You should create/discretize the model first!'.

While running, the software will indicate its current progress/function in the LMA (see **Section 3.13**). The RMA (see **Section 3.13**) will display the number of iterations needed for the solution.

By default, the software only displays the final flow solution. This can be changed by right-clicking on the either the 'Solver' button, the 'Backward Particle Tracking' button, the 'Stop' button, or the 'Forward' button, and selecting 'Step by Step' from the menu that appears. This sets the software to display the solution after every outer iteration (see **Section 13.1.1**). This setting lasts only once and therefore must be repeated every time this display option is desired.

After the flow solution is finished the software will continue simulation of any particles or plumes without recalculating the flow (unless the model is in transient state).

If the model is in transient state, the flow will be recalculated and displayed with every time step. Any particle or plume calculations will be performed with respect to the transient flow field.

A window will appear when the simulation length has been reached, if the **Stop when simulation length is reached** option was checked in the 'Simulation Time Parameters' window (see **Chapter 11**).

> **i** While a simulation is in progress, the 'Stop' button is the only one that is available. The software will finish its current time step calculations before stopping the simulation.

The simulation time is tracked by the clock in the WAAD (see **Section 3.13**) and the 'Flow Time:', 'Plume Time:', and 'Particle Time:' clocks in the SATDI (see **Section 3.5**). The WAAD clock displays the greatest time from the three SATDI clocks.

> **i** Every time the model is run, the solution is based on the current settings. In other words, the present state of the model (velocity vectors, heads, etc) is the initial condition for the next solution.

14.2. Zone Budget

In order to activate the **Zone Budget** option for a zone (defined by a polygon) in the model, the user should select that zone in AE and check the 'Zone Budget' box located in the bottom of RHP (**Figure 14-1**). Any existing polygon can be used for zone budget or user can create polygons in models exclusively for zone budget. Any number of zones can be selected for zone budget. The software would separately show zone budget for each of these.

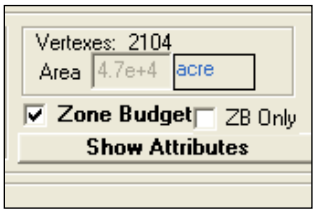


Figure 14-1 Zone Budget Activation

IGW Version 5.0P calculates zone budget for the entire Geological Layer in which the polygon for zone budget is selected.

As soon as the zone budget box is checked for a zone in the model, a hierarchical structure of model zones appears in TPS (refer to **Section 4.1.3** for TPS) for all zones selected for zone budget. Under every zone there are two items, viz., 'Water Balance' and 'Plume Mass Balance'. In order to display the 'Water Balance' or 'Plume Mass Balance' for any selected zone, the user has to check the box(es) before the desired item(s) in the TPS as shown in **Figure 14-2**. The software will display each selected item in a separate window.

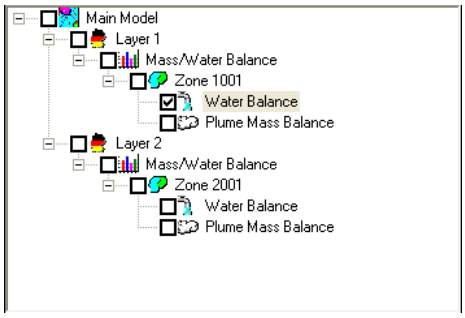


Figure 14-2 Item Selection for Demonstrating Water/Plume Mass Balance in TPS

The zone budget gives the water balance of a selected model zone in terms of inflow and outflow RELATIVE to groundwater. It is demonstrated with a graph of where the horizontal axis represents the features contributing to the model, and the vertical axis represents the water flux with a unit of m^3/day. The zone budget is layer based. The last component of the zone budget (in a multi-layered model) indicates how much flux is being exchanged from the selected zone within a layer to the other model layers which are immediately on top and bottom of this layer.

(+) values on the flux axis define the inflow from existing modeling features to groundwater, and (-) values on the flux axis define the outflow from groundwater to existing modeling features. *IGW Version 5.0P* can generate graphs for cumulative (end of simulation) (**Figure 14-3**) as well as transient water balance for different components in the model (**Figure 14-4**).

To see the exact values of water balance components in **Figure 14-3**, the user can right click anywhere inside the chart and a '2D Chart Control Properties' window will up. In this window, clicking on the 'ChartGroups' tab, then selecting 'Data' tab and clicking on the 'Edit' button will display the exact values for water balance components. This is explained in more detail in **Section 14.3**.

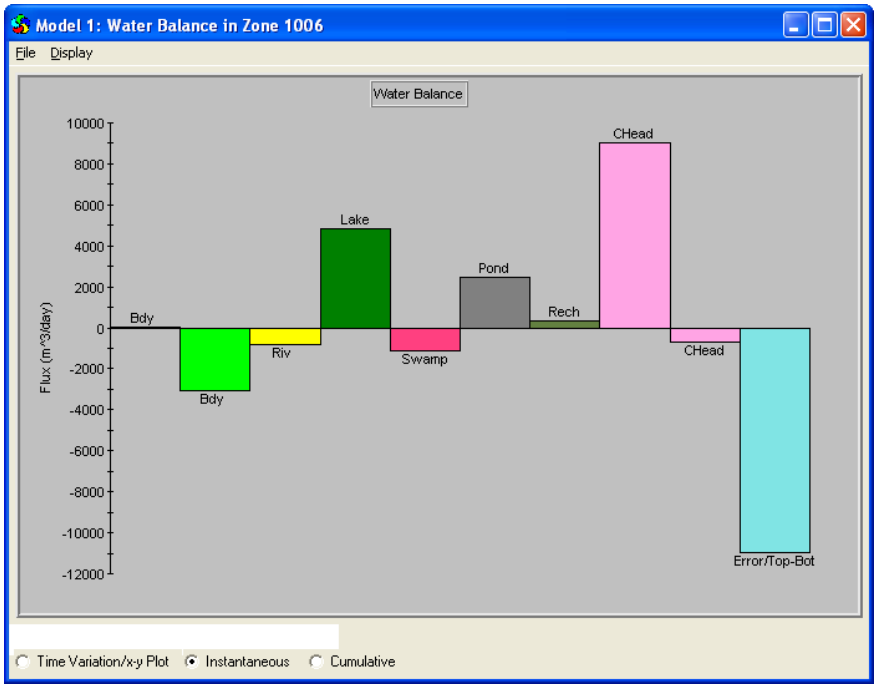


Figure 14-3 Water Balance

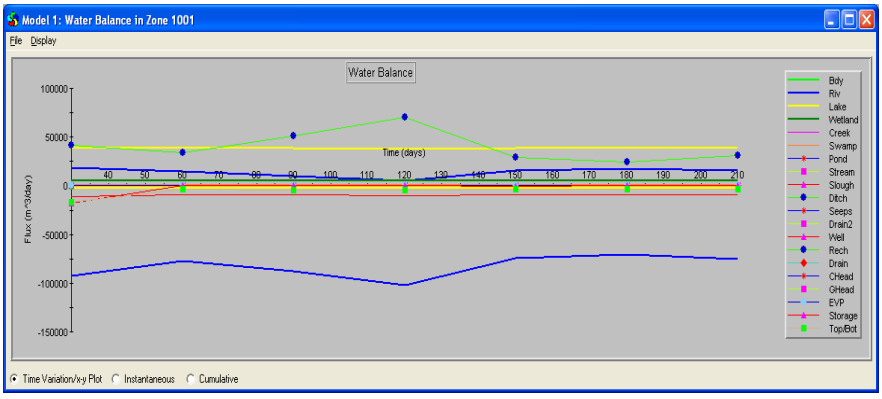


Figure 14-4 Time variant water balance

> **i** Here, only instantaneous display is shown. For other display options, please refer to **Appendix D-I-I**.

> **i** While some of the water balance components can be conceptually and mathematically evaluated in the same way (e.g. river and stream), IGW can handle them separately for zone budget, depending on the '*name'* assigned to the feature.

Water balance components included in *IGW Version 5.0P* are summarized in **Table 14-1**.

Table 14-1 Water Balance Components

WATER BALANCE COMPONENTS		
Category	**Sub Category**	**Boundary Condition**
Boundary	Boundary	Inflow/outflow through the boundary of a neighbor zone
	Top/Bottom	Vertical inflow/outflow between two connected layers
	Chead	Inflow/outflow through a prescribed head boundary
	GHead	Inflow/outflow through head dependent flux boundary
Point	Well	Inflow/outflow through head dependent flux boundary
Polyline	River	Inflow/outflow through head dependent flux boundary
	Creek	Inflow/outflow through head dependent flux boundary
	Stream	Inflow/outflow through head dependent flux boundary
	Drain	Inflow/outflow through head dependent flux boundary
Polygon	Wetland	Inflow/outflow through head dependent flux boundary
	Swamp	Inflow/outflow through head dependent flux boundary
	Pond	Inflow/outflow through head dependent flux boundary
	Slough	Inflow/outflow through head dependent flux boundary
	Ditch	Inflow/outflow through head dependent flux boundary
	Drain	Inflow/outflow through head dependent flux boundary
	Recharge	Inflow/outflow through head dependent flux boundary
	Evapotranspiration	Inflow/outflow through head dependent flux boundary
Storage		Storage at the corresponding zone

14.3. 2D Chart Control Options

Right-clicking on the water balance window brings up the **2D Chart Control Properties** window. Zone budget 2D Chart Control Options (**Figure 14-5**) are briefly given in **Table 14-2**.

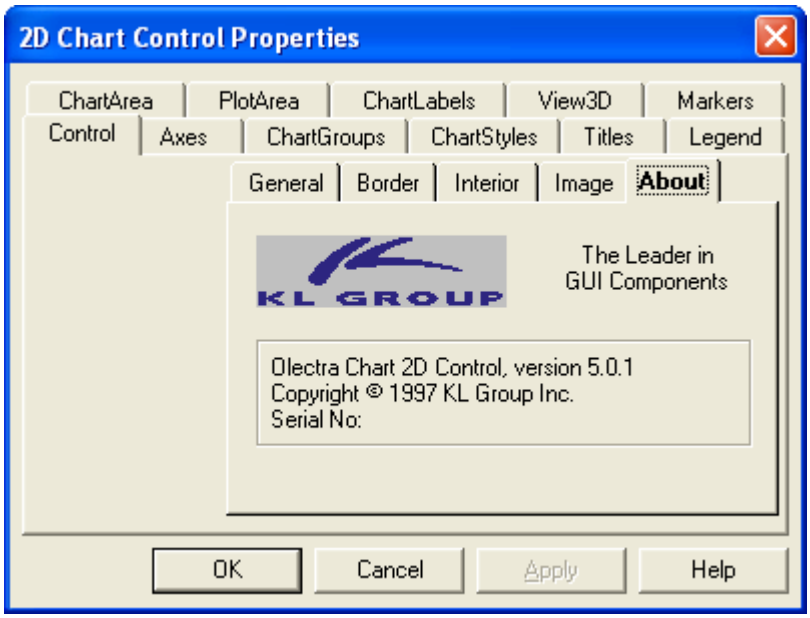


Figure 14-5 2D Chart Control Properties window

Table 14-2 2D Chart Control Properties

2D CHART CONTROL PROPERTIES

TAB	SUB BUTTONS	FUNCTION
Control	General	Loading/saving 2D chart files in *.oc2 format
	Border	Changing the view of the chart border
	Interior	Changing the color of the chart background and axis components
	Image	Importing image to the background of the chart
	About	Credits for the manufacturer of the chart interface
Axes	General	Changing the axis demonstration
	Annotation	Changing the view of the axes components
	Scale	Changing the scale of the axes
	Title	Entering title for the chart
	Grid	Drawing grid along axes
	Grid Style	Changing the grid style and color
	Font	Changing the font style and size for axes
	Polar/Radar	Changing the annotation angle of axes
Chart Groups	General	Changing the chart style
	Data	Editing data from the water balance chart **(Figure 14.2.2)**
	Bar	Stretching and changing length of the x axis
	Labels	Adding/removing/modifying labels
	Internet	Giving data path for data from internet
	Binding	Adding/removing data fields
Chart Styles	Fill Style	Changing the color of the chart components (by default, bars)
	Line Style	Changing the style and color of the lines
	Symbol Style	Changing the style and color of budget columns
Titles	General	Changing the location and visibility of the chart title using defined options
	Label	Changing the name of the chart title
	Location	Changing the location of the chart title by entering coordinates
	Border	Changing the view of the chart title box
	Interior	Changing the background and foreground color of the title box
	Font	Changing the font style and size of the chart title
	Image	Importing image to the background of the chart title box
Legend	General	Changing the location and visibility of the chart legend
	Location	Changing the location of the chart legend by entering coordinates
	Border	Changing the view of the chart legend border
	Interior	Changing the background and foreground color of the chart legend
	Font	Changing the font style and size of the chart legend
	Image	Importing image to the background of the chart legend
Chart Area	General	Changing the view style of the chart components
	Location	Changing the dimensions of the chart area
	Border	Changing the view of the border of the inner chart area
	Interior	Changing the background and foreground color of the inner chart area
	Image	Importing image to the background of the inner chart area
Plot Area	General	Delineating the plot area of the chart
	Interior	Changing the background and foreground color of the plot area
	Image	Importing image to the background of the plot area
Chart Labels	General	Changing the name and location of the chart labels
	Label	Changing the name of the chart labels
	Border	Delineating chart labels
	Interior	Changing the background and foreground color of chart label boxes
	Font	Changing the font style and size of the chart label boxes
	Location	Adding/removing labels
	Image	Importing image to the background of the chart label boxes
View 3D	General	Adding 3D effect to the plot area
Markers	General	Adding markers to the axes
	Attach	Changing the location of markers by data or value
	Line Style	Changing the line style of the axes

The user can select **Chart Groups** tab, and then **Data** sub tab. Clicking on **Edit** button in **Data** sub tab (**Figure 14-6**) opens flux (inflow/outflow) data of in a spreadsheet. On this spreadsheet,

water budget is given in m^3/day through multiple rows for the flow features that are represented in the model.

An example of this spreadsheet is shown in **Figure 14-7**. The values in this figure correspond to the water balace graph shown in **Figure 14-3**. Rows follow the same sequential order as the modeling features given by the x-axis of the water balance graph. A (-) value indicates groundwater outflow, while a (+) value indicates inflow to groundwater. However, this spread sheet does not show the corresponding names of the components as they appear on the graph. The user has to follow to sequence.

The user may copy/paste the data onto a spreadsheet such as Microsoft Excel for further evaluation.

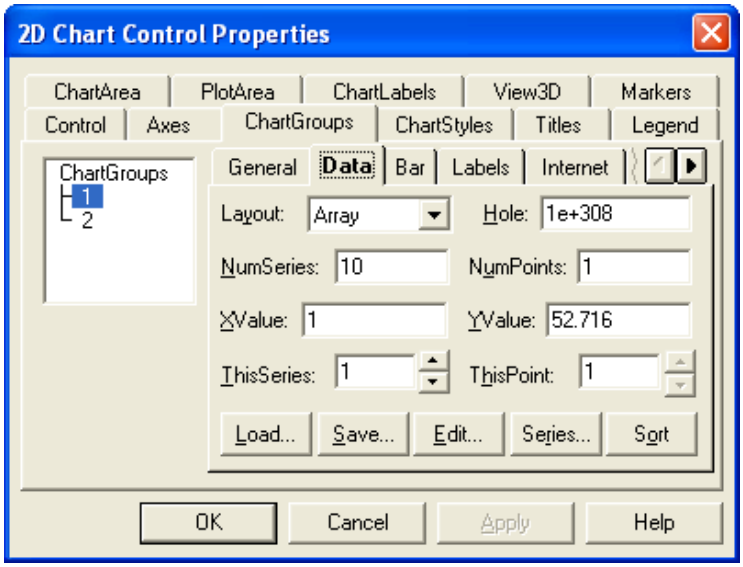
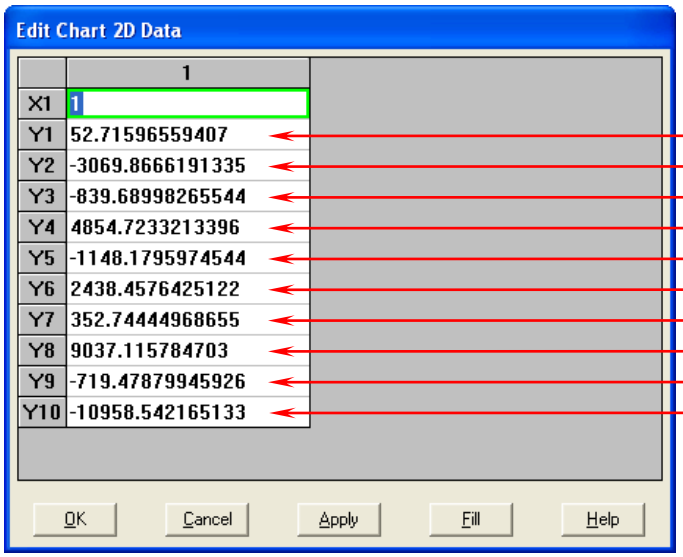


Figure 14-6 2D Chart Data Control Properties – Chart Groups tab with Data sub-tab



The following water budget components and their values correspond to the graph in **Figure 14-3:**

- Boundary (in)
- Boundary (out)
- River
- Lake
- Swamp
- Pond
- Recharge
- Constant Head (in)
- Constant Head (out)
- To Bottom Layer

Figure 14-7 2D Chart Data Spreadsheet

14.4. Initializing Model Features/Clocks

Various aspects of the model can be initialized independently of each other.

Concentration plumes are initialized by clicking the 'Initializing Plume' button. The plumes return to their starting locations and concentrations.

Particle features are initialized by clicking the 'Initializing Particles' button. The particles return to their initial locations.

The clocks associated with these features are reset by clicking the 'Reset Flow Clock', 'Reset Concentration Clock', and 'Reset Particle Clock' buttons located on the Button Palette at row 10, columns 1, 2, and 4.

Please also refer to **Section 11.2**.

Chapter 15 HIERARCHICAL MODELING

In *IGW Version 5.0P* the user can built submodels (or **child models)** with a finer grid within a main model (or **parent model)** with coarser grid. A submodel is a localized model that yields greater detail over a specific region of the Working Area. Submodels use the parent model solution as starting and boundary conditions.

The term '*hierarchical modeling*' implies that *IGW Version 5.0P* allows the user to keep refining the grid across *multiple scales* by building child models within the parent models in a hierarchical manner.

Another feature of hierarchical modeling in *IGW Version 5.0P* is the option of '*up scaling'* and '*downscaling'*. When a child model derives its boundary conditions and starting conditions from the parent model (at the common nodes), the process is termed as *downscaling*. The child model, after solving its domain passes back the refined solution to the parent model at the common nodes. The main model updates itself based on the feedback from the child model. This process is called *up scaling*. After a number of iterations between downscaling and up scaling, the solutions at the boundaries between the parent and child models should converge. The user can set the convergence criteria between up scaling and downscaling.

The following sections describe the implementation and functionality of multi-scale hierarchical modeling in *IGW Version 5.0P*.

> **i** Submodel solutions, because of their finer grid settings (assuming the grid density is not significantly reduced by the user), are extremely useful for modeling around areas of sharp head gradient.

15.1. Defining Submodels

The first step in defining a submodel is clicking the 'Create Submodel' button. This puts the cursor in 'draw mode'. The user may now define a submodel in the Working Area by defining its shape (same as defining a zone – **Section 7.1**).

Alternately, the user may type in the coordinates for each vertex of the submodel area (in the VCI – see **Section 3.13**) instead of clicking the mouse at the desired location. This method is not limited by the resolution of the screen/mouse relationship, and allows for more precise development of submodel features.

As soon as a submodel polygon is created in the Working Area, it becomes the active feature. If the user opens the AE, it will already have the submodel selected in LHP. The corresponding RHP for submodel will also appear in the AE.

> **i** The user can define the submodel area as a polygon, however *IGW Version 5.*0P will always create a rectangular zone for a submodel. The software automatically snaps the polygon's vertices to the nearest nodes in the parent model to form the rectangular region.

> **i** The (finer) grid size in the submodel is defined by specified fractions of the grid size of the main model. Therefore, if the grid size in main model is modified, the submodel grid is also modified.

! The original polygon drawn by the user for the submodel is not retained by the software. If the grid size is changed in the main model at a later stage, the vertices of the rectangular submodel will be snapped to the new grid in the main model, rather than those of the original polygon.

The following sections explain the attributes of a submodel.

15.2. RHP for Submodels in Attributes Explorer

The submodel polygon drawn by the user becomes accessible in the LHP of AE. When sub-model is selected in the LHP, the sub-model RHP appears as shown in **Figure 15-1**. The submodel RHP has three tabs, plus a general area in the bottom of the RHP.

The general area in this window provides display options (explained in next section) and solver options for the submodel.

Clicking the Submodel Solvers ... opens the '**Model x-x : Solver Settings**' window. This window is exactly the same as shown in **Figure 13-1**. The functionality of this widow is explained in **Chapter 13**. Submodel Solvers become available only after the submodel has been discretized.

The three tabs in the RHP, viz., **Model Grid**, **Down/Upscaling** and **Scatter Point Control** are used to assign attributes to the submodels. These are discussed in detail in Section

R The performance of different solvers can be tested by defining multiple submodels over the same area, selecting different solvers for them, and comparing the results.

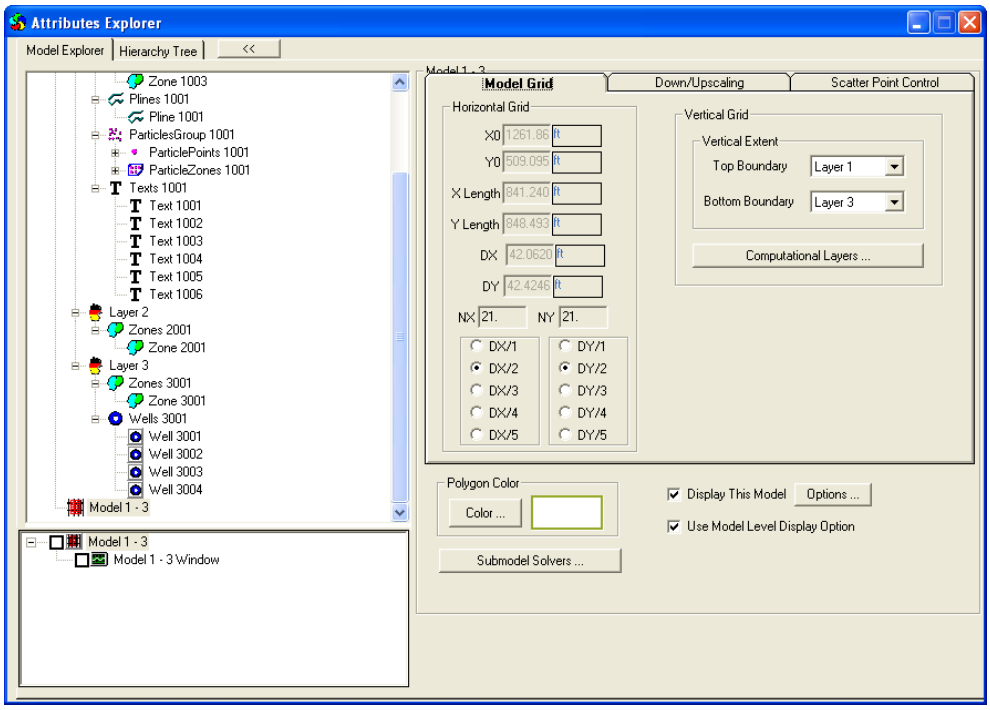


Figure 15-1 Submodel Parameters

15.3. Submodels Display Options

In the general area of submodel RHP (under the tabs), the user can choose the **Color** of submodel polygon. The user can also choose whether or not to display the submodel in the main model by checking the '**Display This Model**' box. By default this box is checked. By clicking on '**Options …**' button, '**Display Options for Model 1- x**' window appears. This widow is exactly similar in appearance and function as for main model display options discussed in **Chapter 19** (please also refer to **Figure 19-1**).

The '**Show Tree Map**' option, when checked will display the a hierarchical map (please refer to **Figure 15-2**) of all the submodels. When the model is running, this map also dynamically shows the direction of flow of information between the parent models and child models

When '**Use Model Level Display Options**' box is checked, the contours and/or color gradients within the submodel display will be independent of the main model. This box is checked by default. However, in order to match the color code and contour scale of the main model, the user have to uncheck this box and supply the 'Given Values' for maximum and minimum range of the attribute being represented by color gradient and/or contour lines (please also refer to **Section 19.4.1**).

15.4. Parent-Model Child-Model Interface

The following subsections briefly describe some details about IGW submodels that may be of interest to the user.

15.4.1. Boundary Conditions

In *IGW Version 5.0P*, submodels use constant head or constant flux values as the boundary conditions. These values are taken from the window in which the particular submodel was drawn. For instance, a submodel drawn in the Working Area will use the main model for its boundary conditions while a submodel drawn in a submodel window will use that submodel for its boundary conditions. Please refer to **Section 15.5.2** for how these conditions are defined for the submodels.

15.4.2. Starting Head

The starting heads associated with the submodel solver iterations are taken from the parent model. Each node (excluding the constant head boundary nodes) is given a starting head value that is determined by linear interpolation between each node in the parent model.

15.4.3. Parameter Interpolation

The parameters associated with each node in a particular submodel are taken from the parent model. The values are assigned based on linear interpolation between each node in the parent model.

15.5. Assigning Attributes to Submodels

User can define/assign various attributes to the submodels using the three tabs in the AE as shown in **Figure 15-1**. These are explained in the following sub-sections.

15.5.1. Model Grid

Horizontal and vertical grid options for the submodel can be selected from the **Model Grid** tab, shown in **Figure 15-1**.

15.5.1.1. Horizontal Grid

Vertical grid settings are shown in the **Horizontal Grid** area under the **Model Grid** tab. Various fields in this area are explained below. All these fields are calculated by the software. The user only decides to choose specified fractions of parent model grid size in X and Y directions that will become the child model grid size in X and Y directions respectively.

X0
This field displays x-coordinate location (in the Working Area) of the lower left corner of the submodel. This number corresponds to the software-interpreted shape of the submodel, versus the user-drawn shape displayed in the Working Area.

Y0
This field displays y-coordinate location (in the Working Area) of the lower left corner of the submodel. This number corresponds to the software-interpreted shape of the submodel, versus the user-drawn shape displayed in the Working Area.

X LENGTH
This field displays the x-direction extent of the submodel area (relative to the submodel origin – X0).

Y LENGTH
This field displays the y-direction extent of the submodel area (relative to the submodel origin – Y0).

NX
This field displays the number of cells in the child model in the X-direction.

NY
This field displays the number of cells in the child model in the Y-direction.

DX
This field displays the grid cell size of child model in X-direction.

DY
This field displays the grid cell size of child model in Y-direction.

15.5.1.2. Vertical Grid

Using the vertical grid options, the user can choose the vertical extent of the submodel within the conceptual layers of the main model.

'Vertical Grid' area in the submodel RHP (**Figure 15-1**) allows the user to specify the layer of the model in which to place the boundaries of the submodel. This save computational time/effort if refined solution is not desired for all geological layers. The user can select which layers will be the top and bottom boundaries of the model by using the pull-down menus next to the respective boundary name.

Clicking on the 

button opens the vertical discretization window for submodel as shown in **Figure 15-2**. Here the user can choose a *fraction* of parent models' computational-layer thickness (within a geological layer) which would be applied to each layer in the child model. The user can choose different *fractions* for different geological layers as seen in the figure below. For example, Geological Layer field in the figure shows that geological layers 1 and 2 are selected as the vertical extents

of the submodel. The line "Layer 2: Parent(3) Child(9)" shows that the parent model has 3 computational layers in Layer 2, while the child model has 9 computational layers in Layer 2.

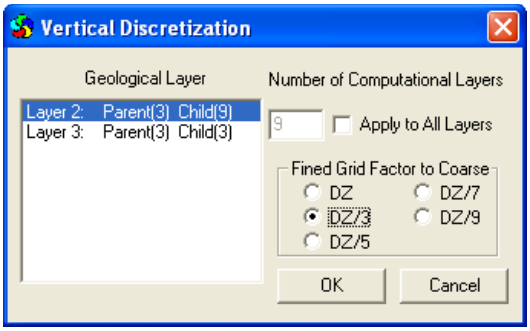


Figure 15-2 Vertical Discretization window for submodel

> **i** The vertical extent of submodels can only be chosen between the geological layers of the main model. Submodels or sub-submodels at any hierarchical level cannot be assigned a vertical extent covering only part of any geological layer.

15.5.2. Up scaling/Downscaling

Up scaling/Downscaling tab in the RHP, shown in **Figure 15-3** provides the options to the user for boundary conditions, interpolation schemes, and iteration criteria.

When '**Downscaling from parent**' option is selected, the user can choose 'Boundary Conditions' between 'Prescribed head' and 'Prescribed flux'. For detailed information on the usage of each, please refer to the ***IGW Version 4.7 Reference Manual***. Also in this area user can choose between interpolation schemes. 'Linear interpolation with conceptual layer' and 'Linear interpolation' are currently available in *IGW Version 5.0P*.

> **!** The user needs to select submodel boundaries so that the results are meaningful. This means the submodel boundaries must be defined far enough away from sharp gradients so that the boundary conditions have not been compromised by the limitations of the coarse grid (parent model) solution.

When '**Up scaling to parent**' option is selected, the user can select boundary conditions for up scaling operation. The available choices are 'Prescribed head' and 'Prescribed flux'. Currently only one interpolation scheme for up scaling operation is available in *IGW Version 5.0P*. When **Up scaling to parent** option is selected, the software performs a number of iterations to between Up scaling and Downscaling operations to converge the modeling results at the interface of parent and child models. **Upscaling/downscaling Iteration** area allows the user to select Max. Iterations (default = 20), Relaxation factor (default = 1) and Head tolerance (default = 0.01m).

> **i** The exact values assigned to the boundary nodes are determined by linear interpolation between each node in the parent model.

> **R** Relaxation factor is only required when 'Prescribed flux' boundary condition is chosen. Recommended value is 0.4.

There are three more options at the bottom left of **Down/Upscaling** tab. All of these options are checked by default.

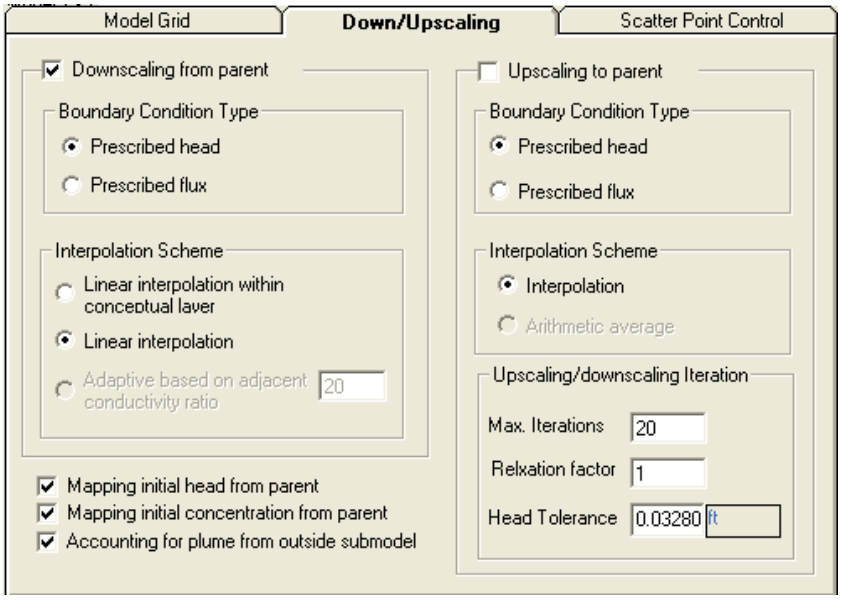


Figure 15-3 Up scaling/downscaling options for submodels

15.5.3. Scatter Point Control

Scatter points are often used in models to interpolate spatial attributes in a model (please refer to **Chapter 20**). Since *IGW Version 5.0P* allows multi-scale hierarchical modeling to very fine scales, the number of scatter points available in (smaller) domains of submodels may become too small to have reliable interpolation of a certain attribute.

Scatter Point Control tab is shown in **Figure 15-4**. Using options given in the Scatter Point Control tab, the user can assign scatter points outside of the submodel domain to be included in the interpolation of attributes.

Options include making scatter points available in an area equal *n* times greater than the submodel size (default value of $n = 2$), or making the sub-area a circular radius, with size based on either a set distance (default = 200m), or a multiple of the submodel's X-length value (default = 100).

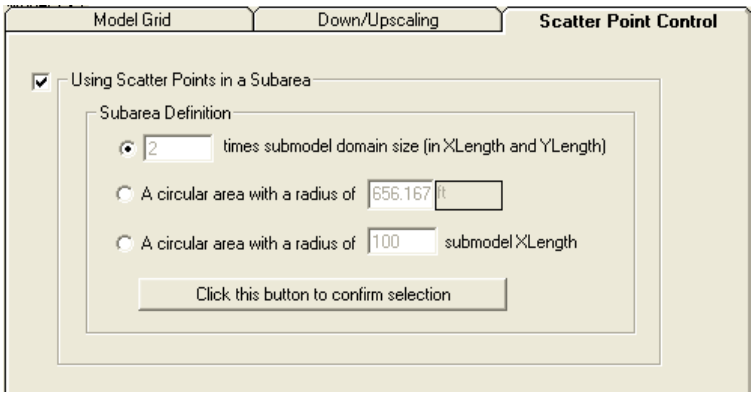


Figure 15-4 Scatter Point Control for the Submodel

In order to include scatter points in submodel from a bigger area, the user may check the box before '**Using Scatter Points in a Subarea**' on the Scatter Point Control tab, then select one of

the options from **Subarea Definition** and finally click on the 

button to confirm selection.

15.6. Discretizing Submodels

After defining a submodel and assigning its attributes, the user needs to discretize the model by clicking on the '**Deep Discretization**' button on the Button Palette. 

When '**Define Model Grid**' window appears (**Figure 15-5**), the user must click on the Advanced Discretization Options button.

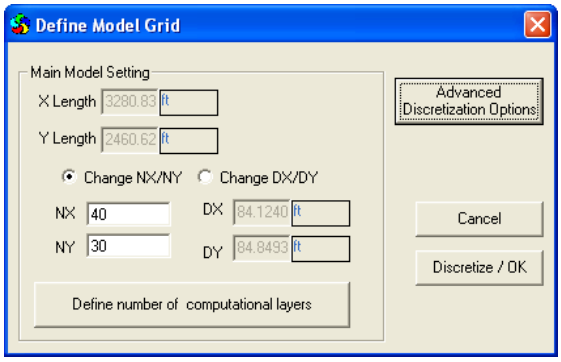


Figure 15-5 Define Model Grid window

This will open Advanced Options window (**Figure 15-6**). In the Selected Model area, the user should choose 'Main and all its submodels' options and click OK. And then click Discretize / OK in the Define Model Grid window.

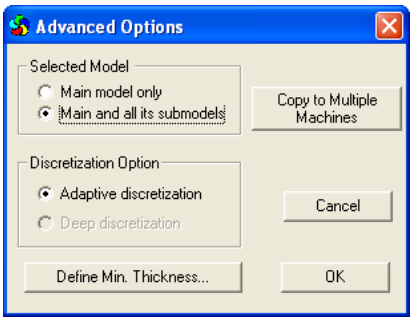


Figure 15-6 Advanced Options window

15.7. Selecting a Submodel

To select a submodel in the Working Area first click the 'Select Submodel Area' button and then click the cursor inside the desired submodel. The submodel becomes outlined in red, indicating that is currently selected. 

Alternately, the desired submodel may be selected in the AE (see **Section 4.1.1**).

15.8. Redefining the Submodel Area

A submodel that has been defined in the software can be redefined by placing the cursor in 'Node Edit' mode (see **Section 3.16**). The user may change the shape of the submodel in the Working Area by either moving the existing vertices, and/or creating new vertices.

To move an existing vertex, click and hold the mouse above the black square that corresponds to the desired vertex, drag the cursor to the desired vertex location, and release the mouse button.

To create a new vertex, click and hold the mouse above the blue crosshair symbol (one exists between each vertex) nearest to the desired location of the new vertex. Drag the cursor to the desired vertex location and release the mouse button.

These steps may be repeated as many times as necessary, until the desired submodel shape is achieved.

> **!** The original polygon drawn by the user for the submodel is not retained by the software. After discretization, the submodel area is converted into a rectangle. The nodes of the new rectangular area can again be edited by the user in the similar manner as described above.

15.9. Viewing Submodels in Separate Windows

After defining a submodel in the working area the user can assess AE and see that all submodel can be seen in TPS (please refer to **Section 4.1.3** for TPS) window as well. If the box next to a submodel is checked, that model will appear as a separate window on the screen. By default, these boxes are already checked. Every submodel created by the user appears in a separate window as soon as the model is discretized. TPS with submodels are shown in **Figure 15-7**.

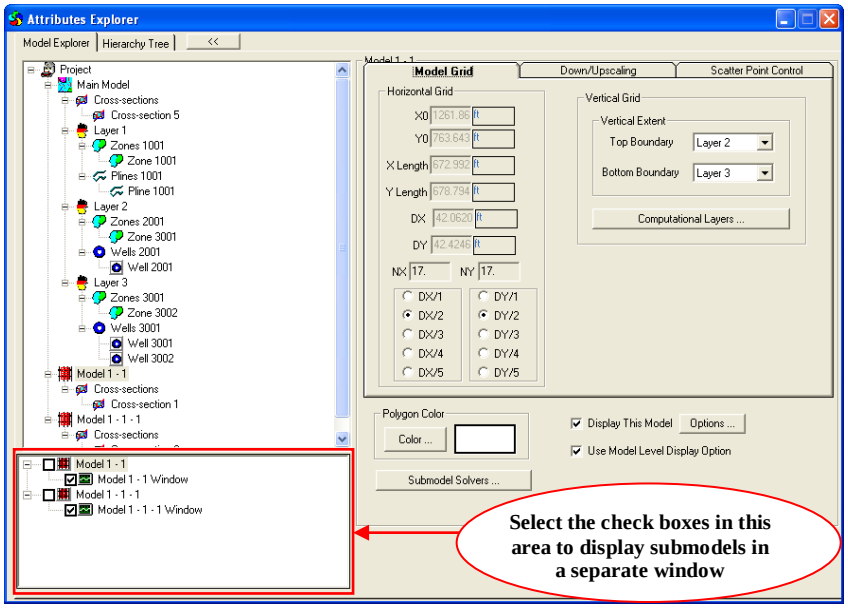


Figure 15-7 Submodels seen in TPS

15.10. Submodel Window

For each submodel defined, a separate window titled 'Model X – X' (where X – X is a software-assigned number; please refer to **Section 15.11.2** for submodel naming convention) will appear when the model is discretized (**Section 15.6**). The windows will cascade from the upper-left-hand corner of the window corresponding to the submodel that appears above it in the LHP (**Section 4.1.1**) of the AE. The location of the initial window will be in the upper left-hand corner of the monitor. The spacing of existing open windows and the required sizes of the newly opened windows may affect the initial display location of the newly opened windows.

Model features (zones, polylines, wells, etc) can be drawn in a submodel window. The increased detail of the submodel window allows for more accurate rendering and placement of smaller features. The effects on the main model will be the same as if the feature was drawn in the Working Area.

Right-clicking the mouse in a submodel window opens a menu from which the user may select any of the following options. (Please also see 'The Right Click-Menu' in **Section 3.17**):

1.) 'Copy'-copies the submodel. The user can paste the copied submodel in the AE by using **[CTRL]+V**.

2.) 'Node Edit' - (the specific submodel must first be active) sets the cursor 'node edit' mode (**Section 3.16**) and subsequently allows the user to redefine the shape of the submodel (same as redefining a zone – **Section 7.3**).

3.) 'Refresh' – Selecting this is the same as clicking the 'Refresh' button on the Button Palette (**Section 3.4**).

4.) 'Display Options' – Opens the 'Submodel Draw Option' window discussed in **Section 15.3**.

5.) 'Show 3D Surface' – Discussed in **Section 21.1**.

6.) 'Show 3D Volume' – Discussed in **Section 21.2**.

7.) 'Export Data' – Begins the process exporting data contained in the submodel

8.) 'Discretizing Table' – Discussed in **Section 12.4.**

9.) 'Grid Based Operation' – Discussed in **Chapter 17.**

10.) 'Unlock Cursor / Discretization Flags' – Discussed in **Section 3.3.5.**

i Functions of which details given above can also be accessed by right-clicking on the submodel polygon in the Working Area. Additionally, the user can 'Copy' option to copy and paste the submodel in the AE by using **[CTRL] + V**.

Closing a submodel window simply stops its visualization and computation. This is sometimes desirable, as it increases Working Area visibility and saves computational power without deleting the submodel. It can be restored at a later time (**Section 15.9**).

i When a submodel is deleted, all of its associated subordinate submodels and associated features will also be deleted.

! Submodels must be deleted prior to Monte Carlo simulation, and/or when modeling multiple scenarios. Otherwise, the program will crash.

R When redefining the shape of a submodel, it is more convenient to do so in the Working Area, as opposed to a submodel window that is limited in extent and will not allow vertices to be moved outside of its borders. Therefore, it cannot be made larger by redefining it exclusively in a submodel window.

15.11. Hierarchical Models

Hierarchical modeling is a very powerful feature of *IGW Version 5.0P*. The following sections describe its conceptual aspects and implementation.

15.11.1. Conceptual

Under the hierarchical modeling system, we model the system incrementally, visualize the results on-the-fly, and ‘‘zoom’’ into sub-areas when and where we feel there is a need to. We begin with modeling the entire area using a coarse-grid and then make localized-corrections by adding submodels or submodels-in-a-submodel where the solution is judged to be inaccurate. The submodel boundaries can be interactively located where the parent dynamics are deemed to be adequately resolved. This process is often iterative and one must use judgment to determine the needed submodel extent/resolution based on the parent-solution behavior and local system characteristics. Since the entire model-hierarchy is dynamically-coupled, one can readily evaluate how different patch refinement schemes affect the ultimate solutions. (Li et. al., 2006) .

The concept of hierarchical modeling is illustrated in

Figure 15-8 (from Li et. al., 2006). The term ‘patch’ used in the figure refers to a submodel.

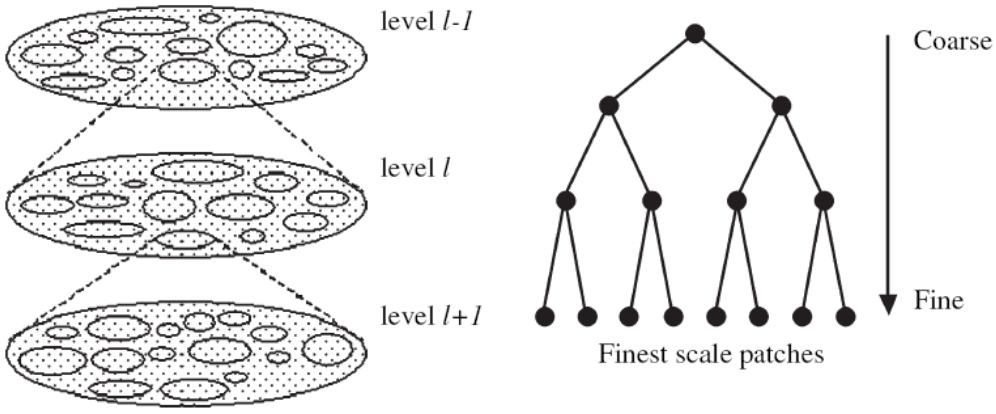
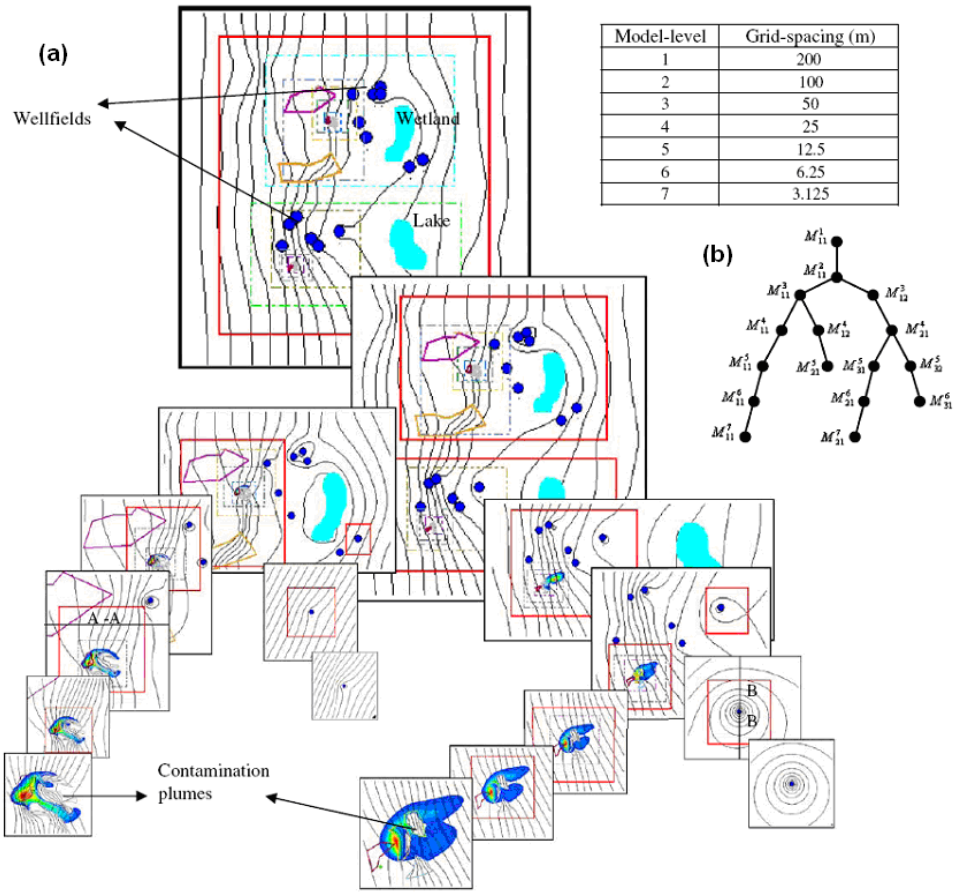


Figure 15-8 Hierarchical decomposition of a groundwater system (from Li et. al., 2006)
Recursive sub-divisions (left)
Hierarchical submodel network (right)

The concept is further illustrated in IGW modeling environment in **Figure 15-9.** It presents the final hierarchical solutions that characterize a heterogeneous system in response to various sources/sinks including the well fields and surface-water bodies. The solutions capture both the regional and local dynamics and provide important information for integrated management. The example in **Figure 15-9** shows a total of 16 submodels in 7 levels, zooming into 4 focused-areas. In general, the number of levels and submodels needed to achieve desired resolutions depends on what the modeling objectives are, how complex the problem is, and how powerful the computer is.



Model-level	Grid-spacing (m)
1	200
2	100
3	50
4	25
5	12.5
6	6.25
7	3.125

Figure 15-9 Example of hierarchical modeling
(a) Hierarchical patch solution in the model
(b) Hierarchical patch network for modeling levels

This window is the same as described in **Section 15.5.1**. Additionally, the options to select the computational layers to be involved are offered in this layer as well.

15.11.2. Submodel Naming Convention

A submodel drawn within another submodel will appear in the LHP of the AE (see **Section 4.1.1**) at the 'Layer / Submodel' level. It appears the same as a submodel drawn in the Working Area, but will retain the name of its parent submodel, a –x (where 'a' is the model name and 'x' is a number). This indicates that its boundary conditions are associated with the submodel it is named after. For example, a submodel drawn in the 'Model 1' window will by default be titled 'Model 1 - 1' to indicate that it is associated with 'Model 1'. If a second submodel is defined in 'Model 1', it will be named as 'Model 1 - 2'. If a submodel is then defined in 'Model 1 - 2', it will be named as 'Model 1 - 2 - 1'. And so on. Names of submodels as they appear in LHP of AE for Model 1 are shown in **Figure 15-10**.

If the name of a parent model is changed, it will not affect the names of any previously defined subordinates; only those defined after the name change will be reflected. The ' - x' number will retain the correct index regardless of the associated name, and will not reset even in the event of subordinate models being deleted.

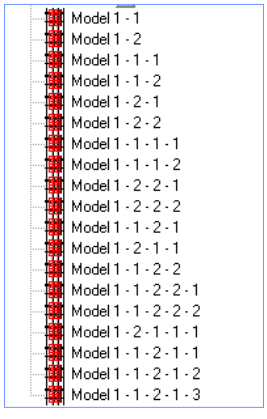


Figure 15-10
Submodel names in LHP of AE

i The user can over-write the name of any submodel in the LHP. Clicking once on the submodel name will highlight it, clicking on it a second time (i.e. clicking on it again) will allow the user to edit its name.

15.11.3. Adding Submodels in a Submodel

IGW Version 5.0P submodels function in a hierarchical setting that allows for the nesting of submodels within other submodels. There is no limit to the number of 'levels' of submodels that may be defined.

To define a submodel (child model) inside an existing submodel (parent model), the existing submodel must be open in a separate window. The user can define child model within the window of parent model in the same manner as explained in **Section 15.1**. Using the 'Create Submodel' button puts the cursor in 'draw mode'. The user may define a child model in the existing parent model window by defining its shape (same as defining a zone – **Section 7.1**). After discretization, the shape of child model will turn rectangular as the nodes of the polygon are snapped to the nearest nodes in the parent model grid.

As soon as a submodel polygon is created in the in the existing submodel window, it becomes the active feature. If the user opens the AE, it will already have the new submodel selected in LHP. The corresponding RHP will be available for adding/editing the attributes in the new submodel.

User can add more than one child models in any parent model at the same level. There can be overlapping boundaries between the child models in any parent model.

! The user has the option to modify the boundaries of a submodel in the IGW main window Working Area as explained in **Section 15.8**. However, the user must make sure that the boundaries of any child model polygon must stay entirely inside of its parent model polygon. Any mistake in this regard may cause an error and the model may not run.

A submodel will not show in a separate window unless the model is discretized. This implies that the model has to be discretized before a new hierarchical level can be added.

15.11.4. Visualizing Hierarchical Structure

Hierarchical structure can be visualized in *IGW Version 5.0P* in two ways.

One, by selecting 'Submodel polygon' in the 'Conceptual Features and Texts' area of 'Display Options for Model 1' window. This will show all submodel polygons in the working area and user can visualize the nested structure of submodels as shown **Figure 15-11**. This visualization allows the user to see the hierarchical structure to some extent as the submodel rectangular boundaries are seen by dot-dash lines. However, it is difficult to see, in this visualization, how many submodels exist at each hierarchical level, and the exact parent-child relationship.

A better visualization of the hierarchical structure is given by the treemap view. By selecting '**Show Treemap**' 

in the '**Display Options for Model 1**' window, a schematic Treemap of nested submodel hierarchy opens up in a separate window as shown in **Figure 15-12**.

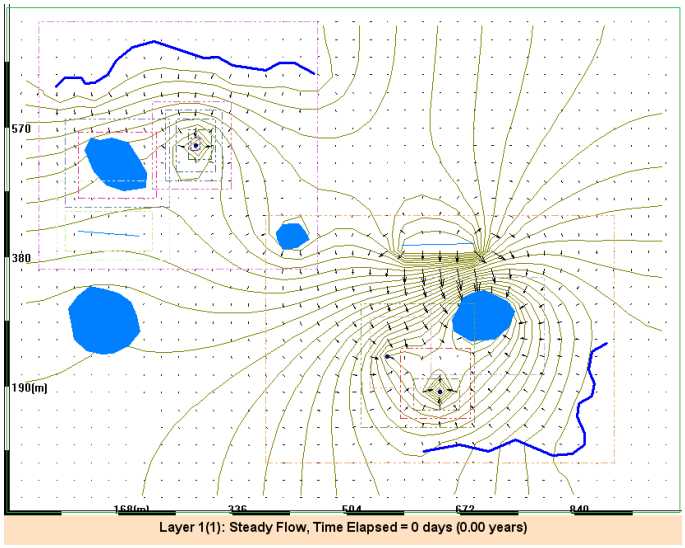


Figure 15-11 Nested submodels seen in the working area

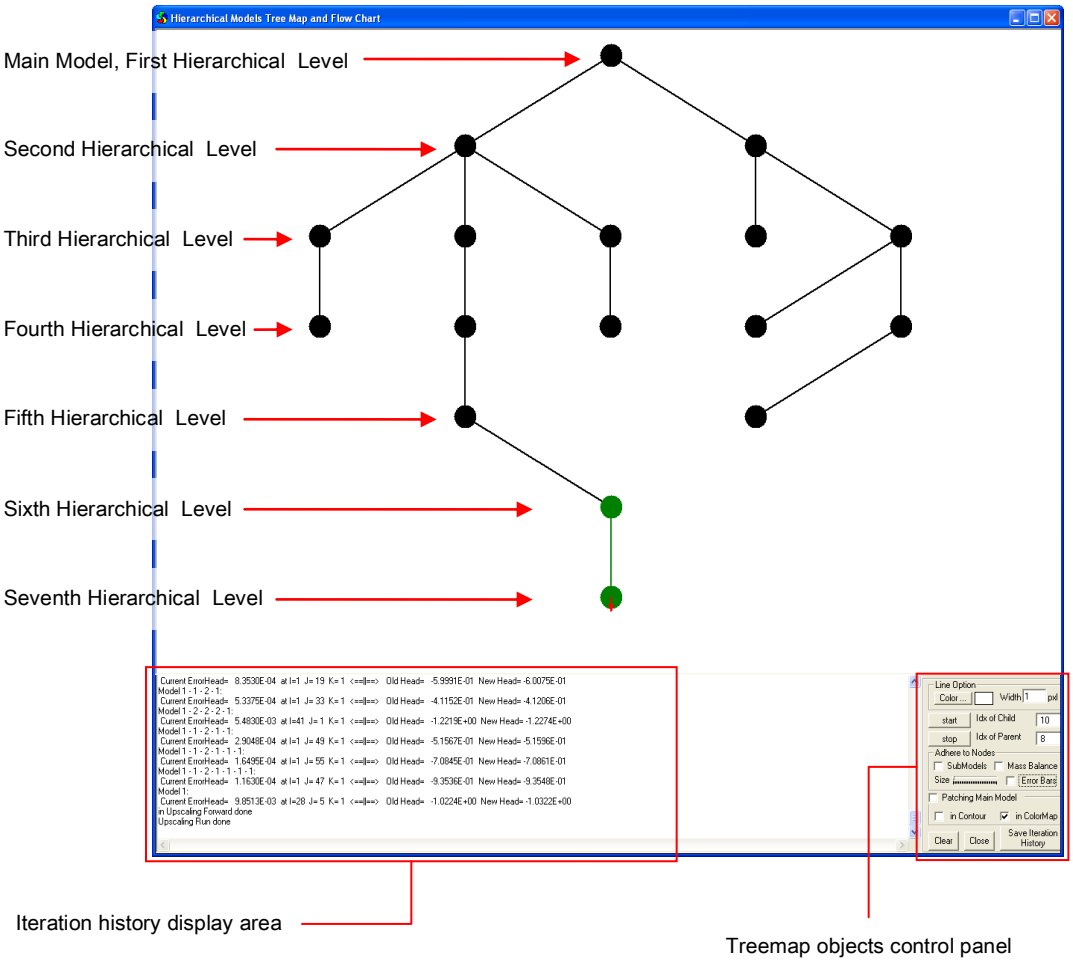


Figure 15-12 Hierarchical Models Tree Map and Flow window

Parent and child models are represented by nodes at their respective levels, connected by lines showing their interrelationships at various levels. When the model is running, this window exhibits a dynamic display. At any given time, a set of parent-child nodes and the line joining

them change color, with an arrow displayed along the line. The color indicates which parent-child model is being currently assessed by the software. The direction of arrow along the line indicates which way the information is flowing i.e., whether the upscaling operation is in progress or the downscaling.

Another dynamic display in this window is the '**iteration history**' which is continuously updated in the bottom left area of the window.

A **Tree Map Objects Control** panel is located in the bottom right corner of the window. **Figure 15-13** shows the panel in detail.

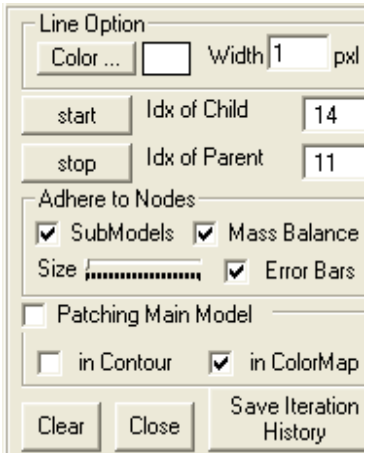


Figure 15-13 Tree Map Objects Control Panel

The '**Line Option'** area allows the user to choose the color and width of the lines connecting the nodes. The same color is applied to the nodes too.

Below this area are start and stop buttons. When the model is running, the user can use these buttons to start/stop hierarchical simulations at any time.

The '**Adhare to Nodes**' area allows user to show the sub models, mass balance of sub models and error of convergence. By default, 'Error Bars' is checked. **Figure 15-14** shows the default view of '**Hierarchical Models Tree Map and Flow Chart**' widow. The error bars appear horizontally by default. Clicking once on any error bar makes it vertical. A text line showing the magnitude of error also appears with every error bar. With the top node representing the main model, the number of iterations performed to reach the convergence criteria is also given by another bar.

User can hover the cursor inside the window. As soon as the cursor touches a node, a text display pops up to the right side of the cursor giving information about the submodel being represented by that node.

By checking the '**SubModels'** box, all submodel windows already open are snapped to the right side of their respective nodes. The user can increase or decrease the size of submodel windows in the tree map by using the sliding button next to '**Size'** in the '**Adhare to Nodes**' area. **Figure 15-15** shows submodel windows in the Tree Map. The display options for any submodel window can be separately chosen by right clicking inside the window and selecting 'Display Options' from the pop-up menu. The display options interface for any submodel is exactly the same as for the main model in the working area. User can refer to **Chapter 19** for details on display options.

By checking the '**Mass Balance**' box, bar graphs showing mass balance for respective submodels appear next to their nodes. Mass balance for submodels is explained in detail in next section (**Section 15.11.5**).

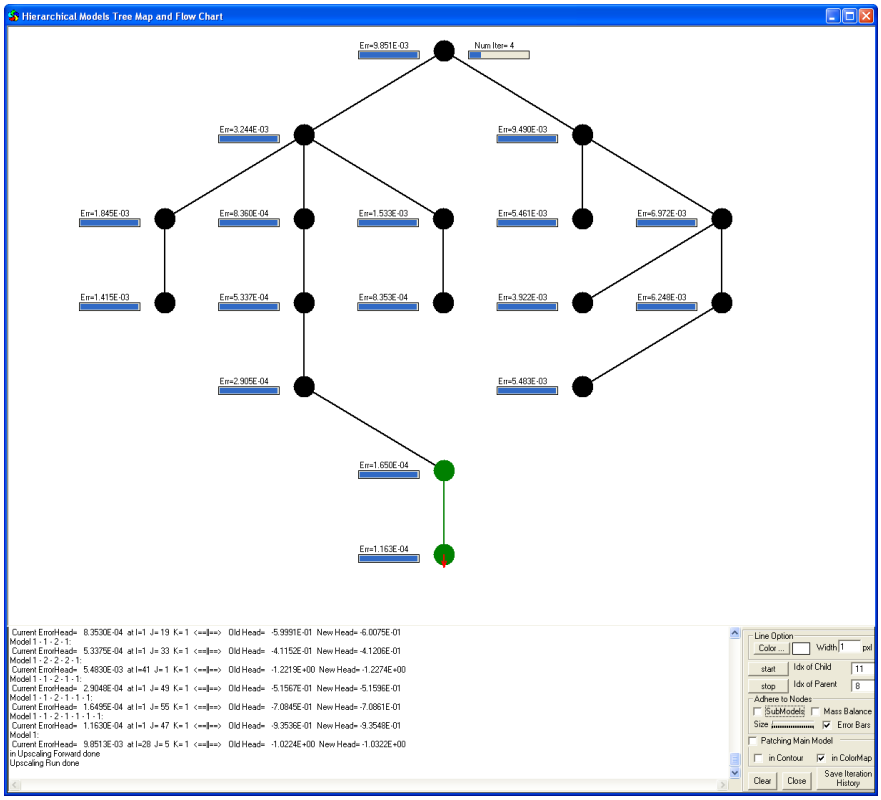


Figure 15-14 Default view of Hierarchical Models Tree Map and Flow Chart window

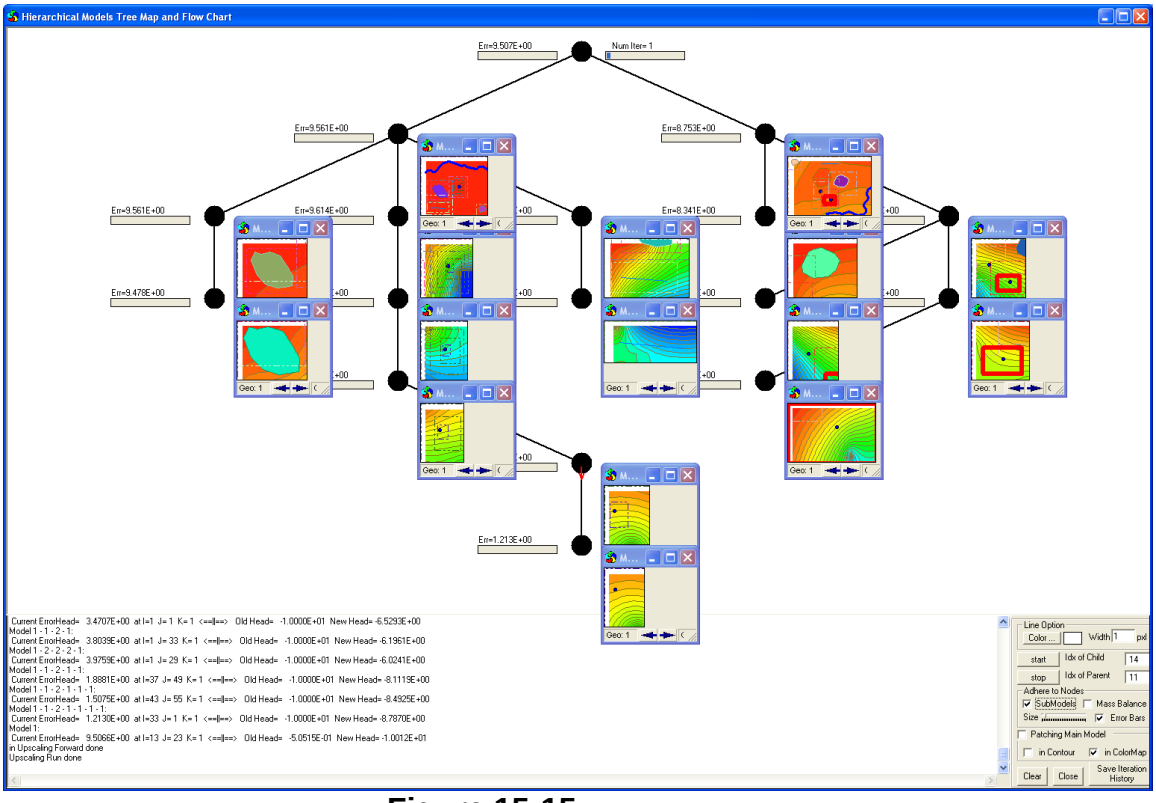


Figure 15-15 Submodel windows

The '**Patching Main Model**' area allows user to 'patch' the refined submodel results into the main model. By selecting different view options, the user can visualize how will the fine details from sub model fit into the main model. **Figure 15-16** illustrates how sub model patches can be compared and visualized in mail models.

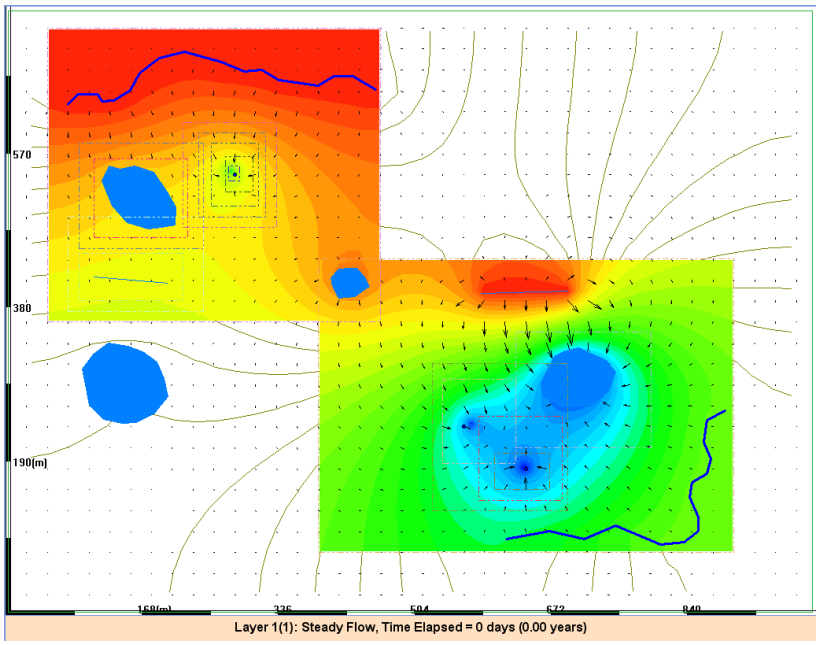


Figure 15-16 Submodel patching in the main model

15.11.5. Mass balance for Submodels

By checking the '**Mass Balace'** box in the Tree Map Objects Control Panel (**Figure 15-13**), the user can observe mass balance for every submodel in the hierarchical structure. However, before checking this box, the user has to open the mass balance windows for the submodels using the following steps.

> **!** Mass balance option for sub models is currently available in *IGW Version 5*.0P for models with only one geological layer and one computational layer. This feature will be extended to multilayered models in the later versions of *IGW*.

(1) Select the main model 'domain polygon' in the AE and check the Zone Budget box (**Section 7.6.2.4**)

(2) Discretize and run the model (**Section 15.6**).

(3) Select Water Balance for the zone in the TPS (**Section 14.2** and **Section 4.1.3**). Zone water balance for the main model will appear in a separate window as shown in **Figure 15-17**.

Notice that this mass balance has a component named '**SubM**'. This component pertains to all the conceptual model features inside the submodels within the main model. Water balance for all the conceptual features inside the submodels are lumped in the SubM bar. However, by double clicking any where in the grey area of this window the user can revert it to the normal water balance display for the main model as shown in **Figure 14-3**. Double clicking in normal water balance display will toggle the display back to the submodel-specific water balance.

Also notice that the SubM bar is stacked with different colors. Each color represents a different submodel at the next hierarchical level. Two colors in SubM bar in **Figure 15-17** indicates that there are two submodels at the next hierarchical level.

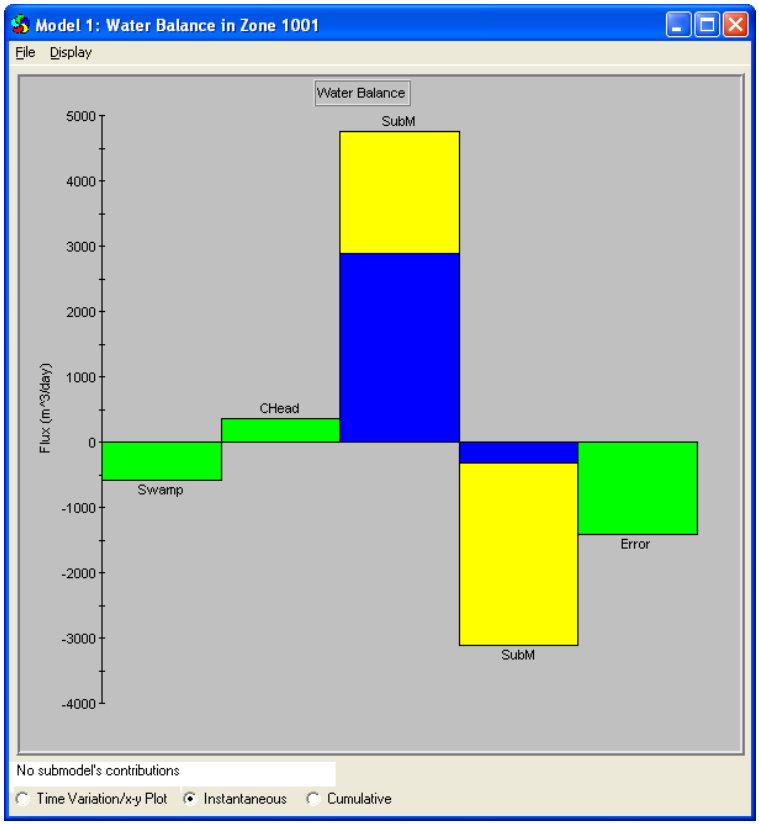


Figure 15-17 Main Model Water Balance

(4) Click once in each color in the **SubM** bar. Each click will open a separate water balance window for the submodel at the next hierarchical level. **Figure 15-18** shows water balance for the next hierarchical level.

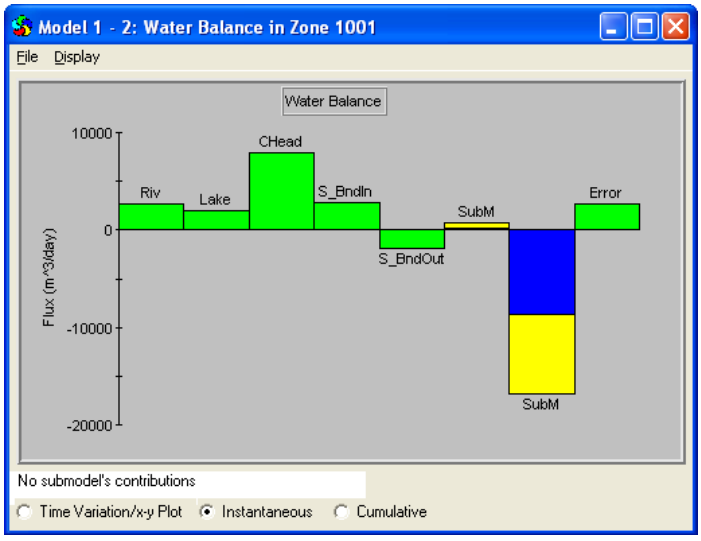


Figure 15-18 Water balance at first hierarchical level

Notice that in **Figure 15-18** there are water balance components with titles '**S_BndIn**' and '**S_BndOut**'. These components represent boundary fluxes across the sub model boundary. Notice that the **SubM** component is still present, which means that next level of hierarchical models exist. This **SubM** component represents all conceptual components contained in the submodels for the next hierarchical levels.

(5) Click once on the **SubM** bar again and the next hierarchical level water balance will appear in a separate window.

(6) Keep clicking in each color of the **SubM** bars till the last hierarchical level is reached. No **SubM** component will be shown in the water balance window when last hierarchical level is reached.

(7) Check the Mass Balance box in the Tree Map Objects control panel (**Figure 15-13**). All mass balance windows for sub models will appear next to their respective nodes as shown in **Figure 15-19**.

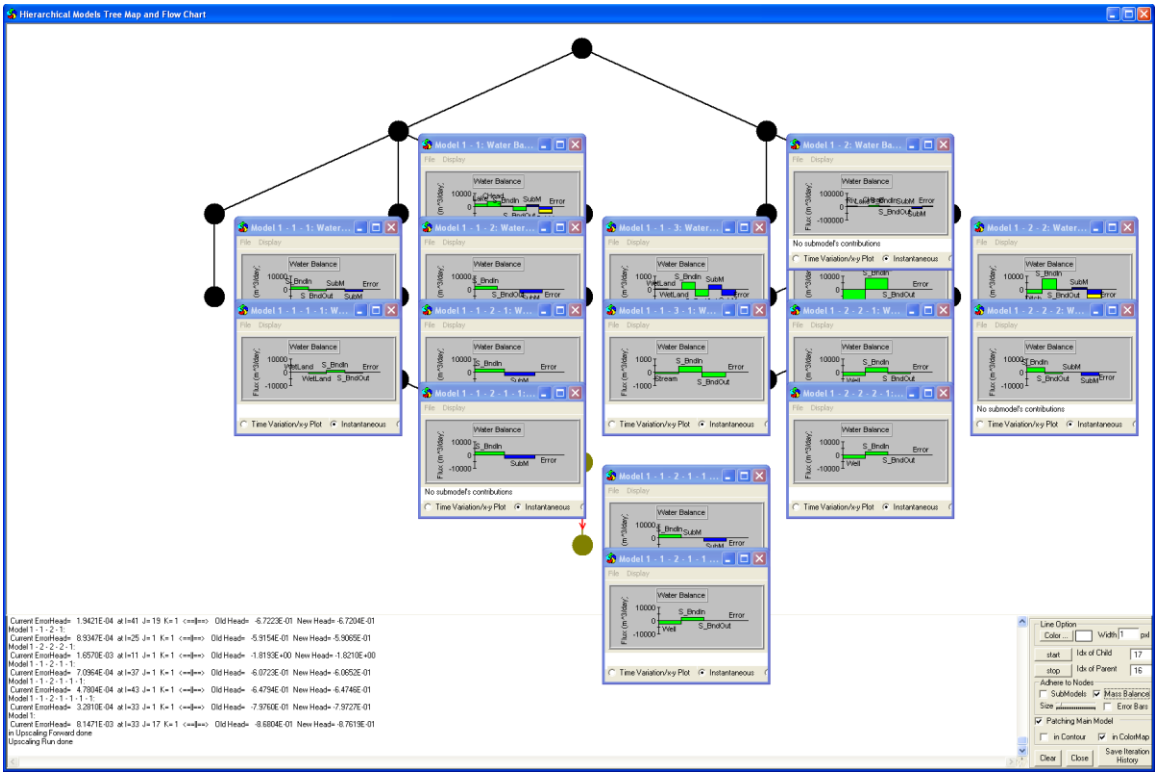


Figure 15-19 Water Balance for Submodels

Chapter 16 CROSS-SECTIONS

A cross-section is a localized model that gives more detail along a specific cross-section of the Working Area. Cross-sections use the parent model solution as starting and boundary conditions.

A typical cross-section from *IGW Version 5.0P* is shown in **Figure 16-1**. Cross-sections[24] show conceptual features in the model, such as wells, rivers, and geological layers etc. They also show modeling results such as velocity vectors, static head contours and static water line (blue dotted line at the top). Geological layers appear in different colors in the cross-sections while the computational layers are separated by dotted lines (yellow dotted lines between the layers). User has options to change the appearance as well as vertical exaggeration of the cross-section. The active computational layer in the cross section is bounded by a thick red line.

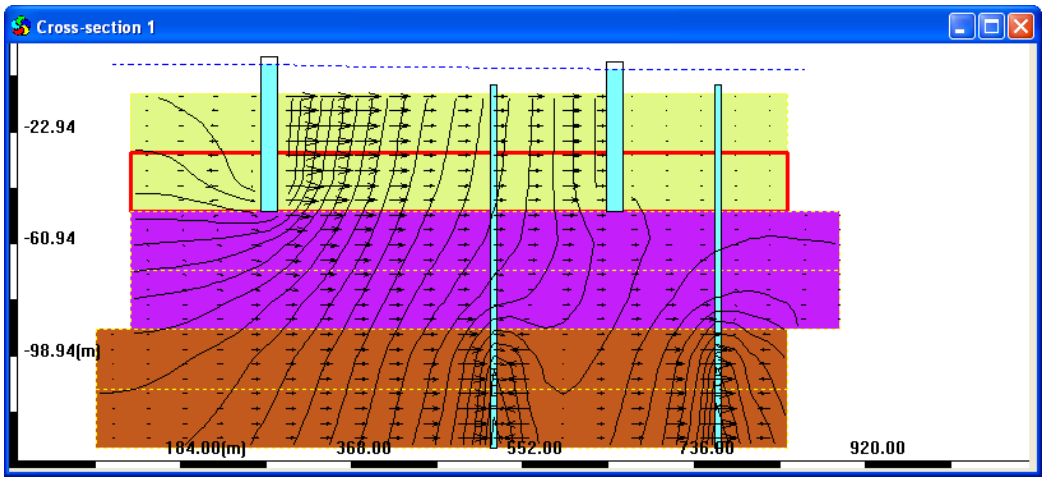


Figure 16-1 A typical cross-section

The following sections describe the implementation and functionality of cross-sections.

R A series of line segments called Poly-linear profile models can be created. The entire defined section is projected into a two-dimensional window and appears as a simple cross-section. Currently, IGW Version 5.0P does not indicate any three-dimensional attributes in the profile model window; so for complex cross-sections keep in mind what it is that is displayed.

16.1. Defining Cross-Sections

The first step in defining a cross-section is clicking the 'Define a Cross-section' button. This puts the cursor in 'draw mode'. The user may now define a cross-section in the Working Area by simply clicking the mouse at points along the desired series of line segments (same methodology as defining a polyline – **Section 8.1**).

Alternately, the user may type in the coordinates for each line segment endpoint (in the VCI – see **Section 3.13**) instead of clicking the mouse at the desired location. This method is not limited by the resolution of the screen/mouse relationship and allows for a more precise development of cross-section features.

When a cross-section is defined in the software it becomes the active feature.

[24] Refer to **Chapter 13** of the ***IGW Version 5.0P Tutorials*** document for examples of defining and viewing a profile model.

After a cross-section has been created, the cursor is still in 'draw mode' and the user may continue to add cross-section s as desired.

> **i** Cross sections are independent of model layers. The user can draw cross section in any model layer and it will show in all model layers in the Working Area.

16.2. Selecting Cross-Sections

To select a cross-section in the Working Area first click the 'Select Cross-section' button and then click the cursor on the desired cross-section. The cross-section becomes highlighted in red therefore indicating that is currently selected. Alternately, the desired cross-section may be selected in the AE (see **Section 4.1.1**).

16.3. Redefining Cross-Sections

A cross-section that has been defined in the software can be redefined by placing the cursor in 'Node Edit' mode (see **Section 3.16**). The user may change the shape of the polyline in either the Working Area or any submodel windows by: 1) moving the existing vertices, and/or 2) creating new vertices.

To move an existing vertex (line segment endpoint), click and hold the mouse above the black square that corresponds to the desired vertex, drag the cursor to the desired vertex location, and release the mouse button. To create a new vertex (i.e. make one line segment into two), click and hold the mouse above the blue crosshair symbol (one exists between each vertex) nearest to the desired location of the new vertex. Drag the cursor to the desired vertex location and release the mouse button.

These steps may be repeated as many times as necessary until the desired cross-section shape is achieved.

16.4. Setting Cross-Section Attributes

Cross-section attributes are set in the AE (see **Chapter 4**). After accessing the AE, the first step is to select the desired cross-section in the LHP (see **Section 4.1.1**). Doing this brings up the 'Cross-section' RHP (see **Section 4.1.2**). A sample of the RHP for 'Cross-section 1' is shown in **Figure 16-2**. The cross-section attributes displayed in the AE are discussed in the following subsections.

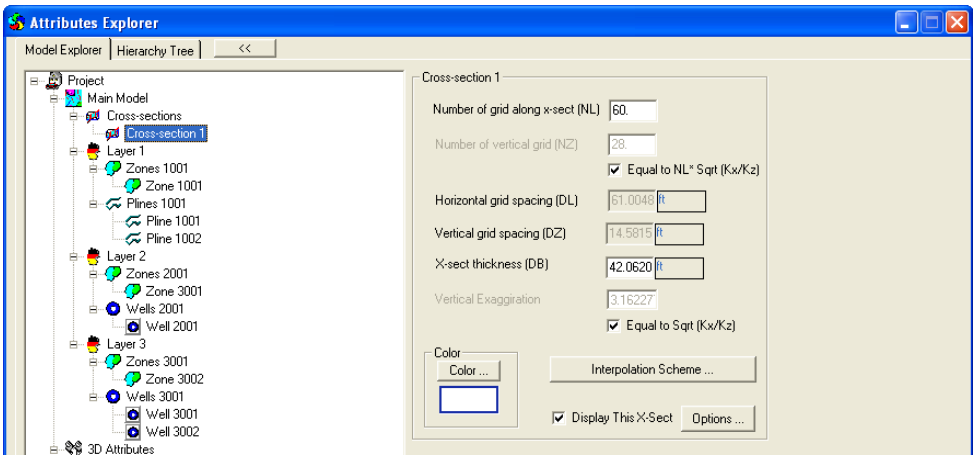


Figure 16-2 RHP for a Cross-Section

NUMBER OF GRIDS ALONG X-SECT (NL)
This field indicates the number of grids along the cross-section.

NUMBER OF VERTICAL GRID (NZ)
This field indicates the number of nodes along the entire length of the cross-section. The default value/entry is 28. The user may change this by entering a number in the appropriate field, however at the present time this feature is inactive.

NZ – EQUAL TO NL * SQRT (KX / KZ)
This field indicates the number of nodes in the z-direction to be included in the cross-section . By default the associated 'EQUAL TO NL * SQRT (KX / KZ)' box is checked and the software automatically determines this number from:

$$NZ = \left(\frac{TOPE - BOTE}{DL\left(\frac{1}{\sqrt{\frac{Kl}{Kz}}} \right)} \right) \qquad (16.5.1)$$

where $TOPE = top\ elevation\ of\ the\ aquifer\ [L]$,

$BOTE = bottom\ elevation\ of\ the\ aquifer\ [L]$,

$Kl = conductivity\ along\ the\ length\ of\ the\ cross\ section\ [L/T]$

The user may uncheck this box and enter a desired number in the field.

i This number is set to prevent the occurrence of ill-conditioned matrix systems in the solution. However, when modeling very large areas, the calculated NZ will be too small (DZ will be too large) to provide enough details.

HORIZONTAL GRID SPACING (DL)
This field indicates the computational (real-world) extent of each cell along the length of the cross-section. It is determined by:

$$DL = \frac{LENGTH}{NL} \qquad (16.5.2)$$

where $LENGTH = entire\ length\ of\ the\ cross\ section\ [L]$

VERTICAL GRID SPACING (DZ)
This field indicates the computational (real-world) extent of each cell in the z-direction. It is determined by:

$$DZ = \frac{TOPE - BOTE}{NZ} \qquad (16.5.3)$$

X-SECT THICKNESS (DB)

This field indicates the display extent of the 'thickness' (looking through) the cross-section. The value is centered about the cross-section. The user may change the displayed thickness of the cross-section by entering a number in the appropriate field.

VERTICAL EXAGGERATION

This field indicates the vertical exaggeration of the cross-section display. By default, the associated box 'Equal to sqrt (Kx/Kz)' is checked and the software automatically enters the AnisF number ($\sqrt{Kx/Kz}$) in the box. The user may uncheck the box and enter a desired number in the appropriate field.

> **i** The AnisF number ensures that the velocity vectors and head contours will be orthogonal (for homogenous, anisotropic media).

COLOR AREA

The color area shows a preview of the cross-section color. There is also a button the user may click to open the 'Color' window and change it. The user may also click in the sample color outline area to open the 'Color' window.

INTERPOLATION SCHEME WINDOW

Clicking the 'Interpolation Scheme' box opens the 'Profile Model Solver' window (shown in **Figure 16-3**). In this window, the user may specify the 'SOR Relaxation' number (1.86 by default), the 'Max. Number of Iteration' number (2000 by default), and the 'Error' number in meters (0.0001 m by default - see **Section 13.3**). These parameters are explained in **Section 13.1.1** and **Section 13.1.2** (SOR and 'Inner Iteration (Matrix Solution) Area' subsections).

> **i** The profile model always uses SOR as the solver method.

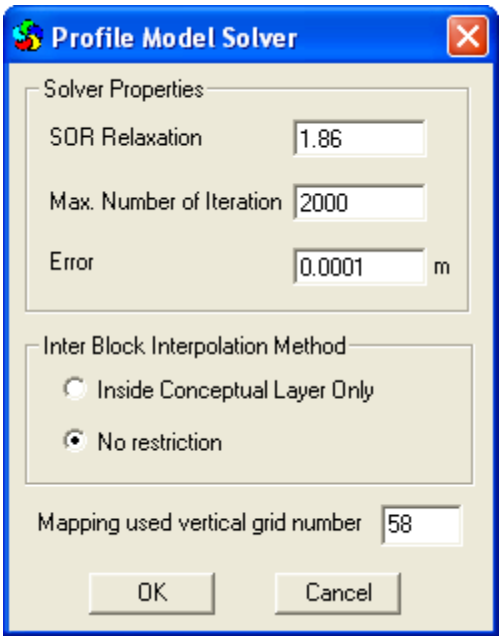


Figure 16-3 Profile Model Solver

16.5. Setting Cross-section Display Options

Checking the box next to the 'Display This X-Sect' button activates the 'Options…' button (this box will become unchecked whenever the associated cross-section window is closed). Clicking the button opens the 'Profile Model Display Options' shown in **Figure 16-4.** It and allows the user to modify the cross-section display. The settings are discussed in the following subsections.

TOP BOUNDARY OF DISPLAY BOUNDING BOX

In this area, the user may choose to either specify ('Given Value' – 30 is the default entry) or allow the software to automatically set ('Automatically') the drawn height of the cross-section.

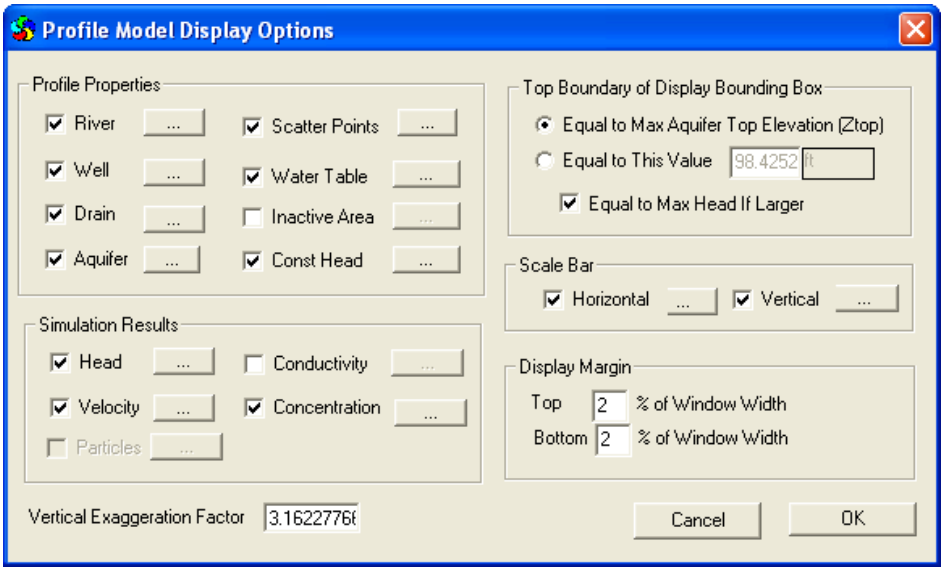


Figure 16-4 Display Options for Profile Model

SIMULATION RESULTS AREA

In this area, the user may choose to activate or deactivate visualization for 'Head', 'Velocity', and 'Conductivity'. 'Head' and 'Velocity' are active by default. Clicking the '…' button next to 'Head', 'Velocity', or 'Conductivity' opens up a separate window for further refinement of the display of these features. These settings affect only the cross-section at hand but the window interfaces are similar to those encountered when setting overall display options through the 'Main Model -- Draw Option' window: ('Draw Option – Profile Head': **Figure 19-7**; 'Velocity Draw Option': **Figure 19-8**; and 'Draw Option – Profile Conductivity': same format as **Figure 19-7**).

PROFILE PROPERTIES AREA

In this area, the user may choose to activate or deactivate visualization for various model features. All of these are active by default. Clicking the '…' button next to a feature opens the 'Draw _____' window (where "_____" represents the feature name) in which the display can be further refined. The window is discussed in the following subsection.

THE DRAW ( ) WINDOW

An example of this window (for the river feature) is shown in **Figure 16-5**.

In this window, the user has the option of activating or deactivating visualization of the outline ('Draw Contour Line'), the area fill ('Area Filling'), and the title

('Show Title') for the feature at hand. All three are active by default. Associated with each is an area in which the user may further refine the display of these.

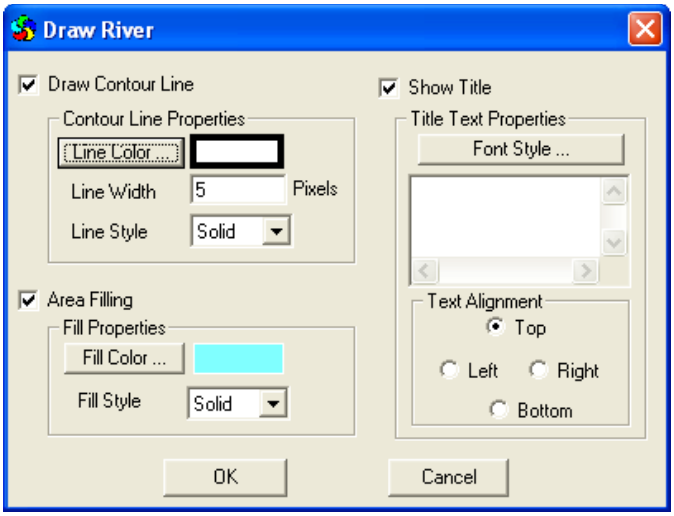


Figure 16-5 Drawing Window

The height of the riverbank above river stage can be specified by entering the percentage in the appropriate field ('Dike is ____% above River Stage' – the default value is 10%).

SCALE BAR

The user may select whether to activate (checked) or deactivate (unchecked) the scale bars in the horizontal and vertical directions. The attributes of the scale bars may be set by clicking the '…' button the appears to the right of 'Show Scale Bar in Horizontal Direction' or 'Show Scale Bar in Vertical Direction' as desired. Clicking the button opens the 'Horizontal Scale Option' or 'Vertical Scale Option' respectively. These windows are the same as those discussed in **Section 19.1**– '**Show Conceptual Model and Labels Area**' subsection – '**Horizontal Scale Option**' and '**Vertical Scale Option**' subsections.

DISPLAY MARGINS FIELD

The 'Top Space' (15 by default – as percentage) and 'Bottom Space' (10 by default – as percentage) fields allow the user to specify the amount of white space that should appear at the top of the window and the bottom of the window, respectively. These numbers should be entered as a percentage of the width of the window.

VERTICAL EXAGGERATION BAR

This function allows the user to exaggerate the vertical distance in the model for better overall clarity. Default value is $\sqrt{10}$, which is the square root of default anisotropy (K_x/K_y).

16.6. Detailed Cross-Section Information

The following subsections briefly describe some details about IGW Version 5.0P cross-sections that may be of interest to the user.

16.6.1. Boundary Conditions

In *IGW Version 5.0P*, cross-sections use constant head values as the boundary conditions at the terminal ends of the profile. For confined aquifer the top and bottom of the aquifer are considered no-flow boundaries. For unconfined aquifers, the values at the top are set to constant head levels

based upon the water table. These head values are taken from the window in which the particular cross-section was drawn. For instance, a cross-section drawn in the Working Area will use the main model for its boundary conditions while a cross-section drawn in a submodel window (see Error! Reference source not found.) will use that submodel for its boundary conditions. The exact alues assigned to the boundary nodes are determined by linear interpolation between each node in the parent model.

i A profile model drawn within a submodel will appear in the LHP of the AE (see **Section 4.1.1**) at the 'Layer / Submodel' level. This is the same as a profile model drawn in the Working Area, but will retain the name of its parent submodel with a *–n* (where *n* is a number) following it. This indicates that its boundary conditions are not associated with the main model, but instead with the submodel it is named after. For example, a profile model drawn in the 'Submodel 110' window will be titled 'Profile Model 110 -1' by default, to indicate that it is associated with Submodel 110.

! If the name of a parent model is changed, it will not affect the names of any previously-defined subordinates. Only those defined after the name change will be reflected. The *– n* number will retain the correct index regardless of the associated name, and will not reset even in the event of subordinate models being deleted.

16.6.2. Parameter Interpolation

The parameters associated with each node in a particular cross-section are taken from the parent model. The values are assigned based on linear interpolation between each node in the parent model.

16.6.3. Grid/Streamline Orientation

IGW Version 5.0P allows for the formulation of cross-sections that do not follow the grid orientation. Also, the software treats cross-sections in such a way that they do not have to be drawn along streamlines (due to the treatment of cross flow discussed in **Section 16.6.4**).

16.6.4. Cross Flow

The profile view is an actual model representing the cross-section. Any net cross-flow into the profile slice is treated as recharge for the cross-section, therefore maintaining an accurate representation[25]. The total cross-flow for a given cell in the parent model is evenly distributed throughout the vertical extent of the cross-section.

16.7. Cross-Section Window

For each cross-section defined, a separate window titled 'Cross-section *n*' (where *n* is a software-assigned number) will appear (see **Figure 16-1**) when the model is discretized (**Chapter 12**). The windows will cascade from the upper-left-hand corner of the window, corresponding to the cross-section that appears above it in the LHP of the AE (the location of the initial window will be in the upper left-hand corner of the monitor), see **Section 4.1.1.** The spacing of existing open windows and the required sizes of the newly opened windows may affect the initial display location of the newly opened windows.

[25] This cross-flow is estimated from the main model and is more accurate when the profile model is not located near sharp gradients (such as would be present near a well).

Right-clicking the mouse in a cross-section window opens a menu from which the user may select 'Refresh', 'Draw Option', or 'Export Data'. Selecting 'Refresh' is the same as clicking the 'Refresh' button. Selecting 'Draw Option' opens the 'Cross-section Draw Option' window discussed in **Section 16.4.** Selecting 'Export Data' begins the process of exporting data contained in the cross-section. The process is the same as for the main model and is described in **Section 23.4** (note that 'Concentration' is not selected by default for cross-section data export).

Note that the CAT (**Section 3.12**) will function with the cursor in the cross-section window the same as it functions with the cursor in the Working Area (except the 'Y' display lists the 'Z' value).

! Cross-sections may be present when performing Monte Carlo simulations. However, the cross-section solver will not function during Monte Carlo simulations and hence the display window will only show the stratigraphy of the default realization.

i In *IGW Version 5.0P*, cross-sections may be present when modeling multiple scenarios, and when particles and concentration plumes are visible in the cross-section windows.

Closing a cross-section window simply turns off its visualization and computation. This is sometimes desirable, as it increases Working Area visibility and decreases necessary computational power without deleting the submodel. It can be restored at a later time (as opposed to having to redefine it and reset its attributes) by checking the box next to the 'Visualizing Cross-section' button in the associated **RHP** of a cross-section (see **Section 16.4**).

Chapter 17 GRID BASED OPERATIONS

Before performing any Grid-Based Operations in *IGW Version 5.0P*, please bear in mind some of the important basics of numerical modeling explained below.

When the user discretizes a model, all the conceptual features in the model are mapped on to the grid nodes. The geometry of the grid is defined by the user in the process of model discretization (**Chapter 12**). The model grid in *IGW Version 5.0P* is a 3D object, with rows, columns and layers. An **array** is a set of values for a certain parameter in the model such that every value in the set corresponds to a particular grid node in the model.

Figure 17-1 shows the visualization of a 3D grid for a model. A 3D array can be visualized with a number (certain parameter value) sitting on every node of the 3D grid.

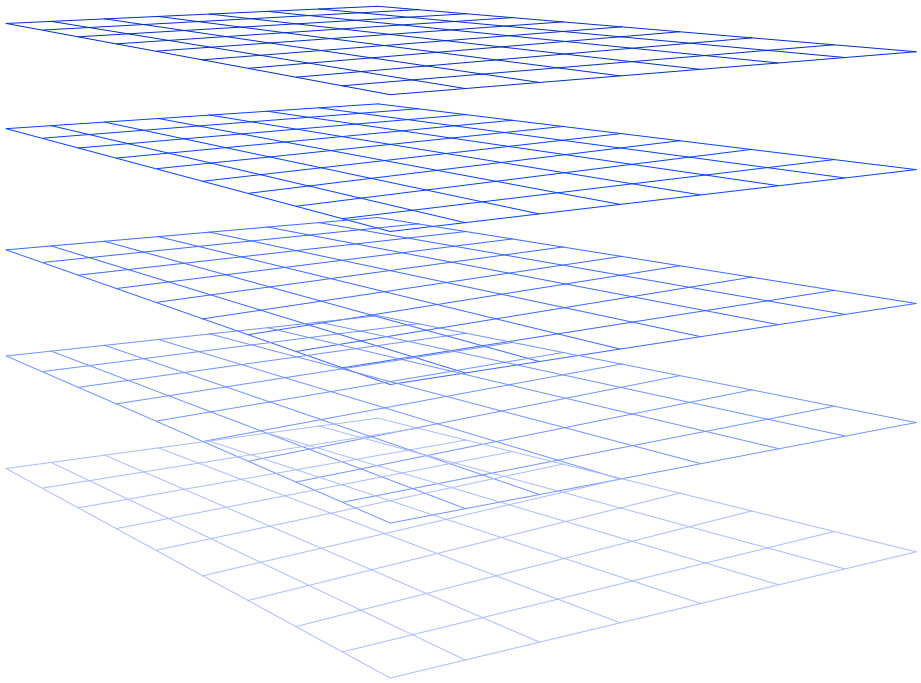


Figure 17-1 Visualization of a 3D grid

The arrays in the model can be categorized in two broad groups, viz., input parameter arrays and output parameter arrays. The input parameter arrays would primarily include values for initial conditions, boundary conditions and physical properties of the hydrogeological medium. The output arrays contain values of parameters such as final hydraulic heads, velocities and solute concentrations. Grid-Based Operations in IGW Version 5.0P allows the user to perform many operations on these arrays. The operations may include addition/subtraction of arrays from each other, addition of constant values in arrays, and multiplication of an array with a constant etc. Other operations may include saving arrays in text files, importing and assigning arrays from text files, copying array layers to Excel® spreadsheets etc.

The usefulness of Grid-Based Operations can be highlighted with the following example. Suppose in a multilayered model, the user wants to do a sensitivity analysis for the effect of change in hydraulic conductivity on the contaminant transport. Using Grid-Based Operations, all the user needs to do is select the hydraulic conductivity array and multiply it with a desired factor, then simply run the model and see the effect. Without Grid-Based Operations, although the user can change the hydraulic conductivity by the same desired factor in '**Multiplier For Sensitivity Analysis**' window (RHP for Zones in AE; **Section 4.1.2.4**), but then the model has to be discretized before it can run. In more complicated processes,

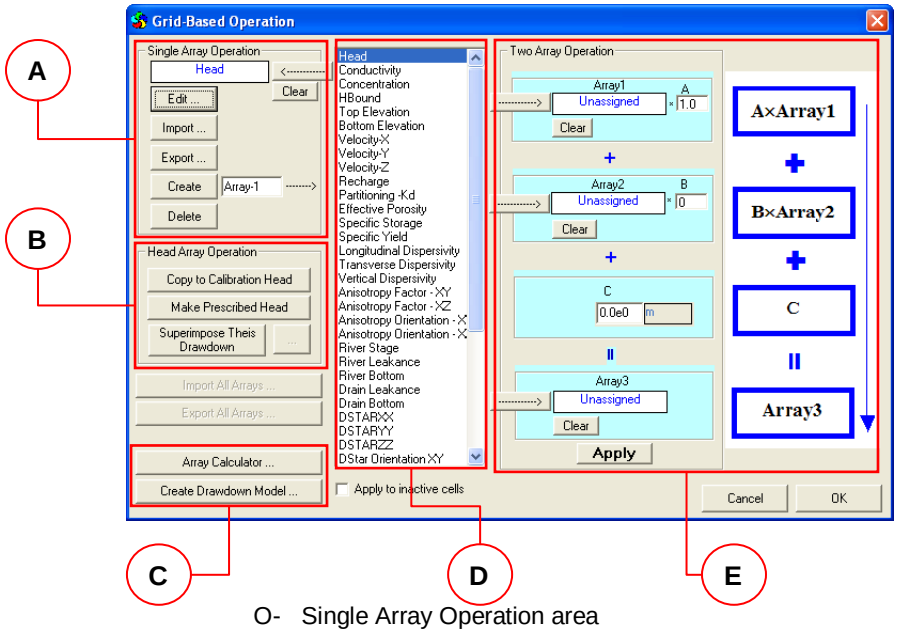
discretization may take longer time. Also, the process resets the whole model based on the conceptual settings which may not be desirable in certain situations while performing sensitivity analysis.

R If you want to see the effect of change(s) made (by using Grid-Based Operations) in any of the listed arrays in the model. DO NOT discretize the model – just do one of the following for visualizing the effects:

- When change is made in an input parameter array, run the model without discretization and observe the change.
- When any modification is directly made in an output parameter (e.g. Head), just refresh the display to observe the change.

! If you discretize the model, all the changes made in model arrays using Grid-Based Operations will be undone. Discretization will replace the values in all arrays based on the conceptual model.

The **Grid-Based Operation** is a calculator for performing array manipulations as mentioned above. It is also used for preparing arrays for IGW and for other simple model array calculations. Grid-Based Operations provide the user with a set of tools through which 3D arrays in the model can be manipulated in a variety of ways. In *IGW Version 5.0P*, Grid-Based Operation tools are consolidated in the Grid-Based Operations window (**Figure 17-2**). The window can be accessed by clicking the Grid-Based Operation button in IGW main window or choosing 'Grid-Based Operation' from the **Utilities** menu, or by right-clicking anywhere in the working area and selecting 'Grid-Based Operations' from the popup menu..



O- Single Array Operation area
P- Head Array Operation area
Q- Array Calculator and Create Drawdown Model buttons
R- Attributes Array List
S- Two Array Operation area

Figure 17-2 Grid-Based Operation Window

17.1. Attribute Array List

In the middle of the **Grid-Based Operation** window, the attribute arrays of the model are listed. These attributes are related to both input parameters (e.g. aquifer parameters, boundary conditions, sources and sinks) and output results (e.g. head, velocity and concentration distribution). The user also can create new arrays using the '**Create**' button in the **Single Array Operations** area. The new arrays are appended at the end of this list. The vertical sliding bar to the right of the Attribute Array list can be used to see all arrays in the list.

17.2. Single Array Operation Area

Single Array Operation toolbox contains several buttons for single array operations. The user can edit, import, export, create and delete an array using these buttons. The buttons in this area explained below:

17.2.1. Assigning Arrays

Prior to using any single array operation (except for creating a new array), it is necessary to specify an array from the array list on which the operation will be performed. To assign an array for single array operation, the user has to first highlight an array from the Attributes Array List by single mouse click, and then press <---------- (assign button). The selected array shows up in the space as can be seen in **Figure 17-3**. Edit, import, export and delete functions can now be preformed on the assigned array.

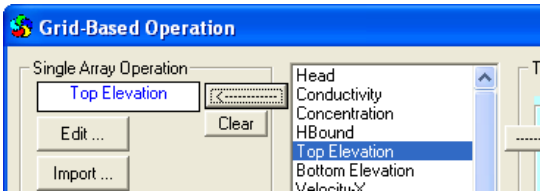


Figure 17-3 Assigning an array for single array operations

17.2.2. Editing Single Arrays

Clicking on the Edit ... button opens the 'Data Table – (`array name`)' window as shown for the Head array in **Figure 17-4**. The editing tools are given on the right side in this window.

Switch Array Layer field allows the user to choose the array layer for editing using the arrow buttons. After the desired layer number comes up in the field, the user has to click on **Go** button to get the layer in the edit area.

> **!** The layer numbers in the **Switch Array Layer** field are based on number of computational layers in the model. If a model has three geological layers, each divided into three computational layers, the field will show the numbers from 1 to 9. The layer numbers increase from top to bottom.

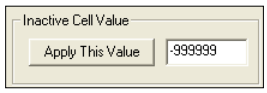
To change values in the cells, the user will first select the cell(s) in the table, then type the desired value for the selected cell(s) in the **Set Value for Selected Cells** field and finally click on the '**Set Value for Selected Cells'** button. The value typed in the field shows up in (all of) the selected cell(s). However, for the edited values to take effect, the user must click on '**Apply Changes**' button in the bottom right of the window.

> **i** For changing values in a single cell in the table, the user can also double click in the cell and edit the value right there.

'Do Linear Interpolation for Selected Cells' button is not active in this window.

The user also can save the current array layer value to Microsoft Excel® by clicking on the **Save Current Layer to Microsoft Excel** button. One click on this button will open an Excel® spreadsheet and all values will be automatically tabulated in it. The user can then save the Excel® spreadsheet just as any other spreadsheet is saved in Excel®.

IGW assigns its own values to the inactive cells. The User can assign a value of his/her choice to the inactive cells in the model in the **Inactive Cell Value** area by first typing the desired value in the field and then clicking the '**Apply This Value**' button. This feature is helpful especially when the user wants to process the array data in other software.



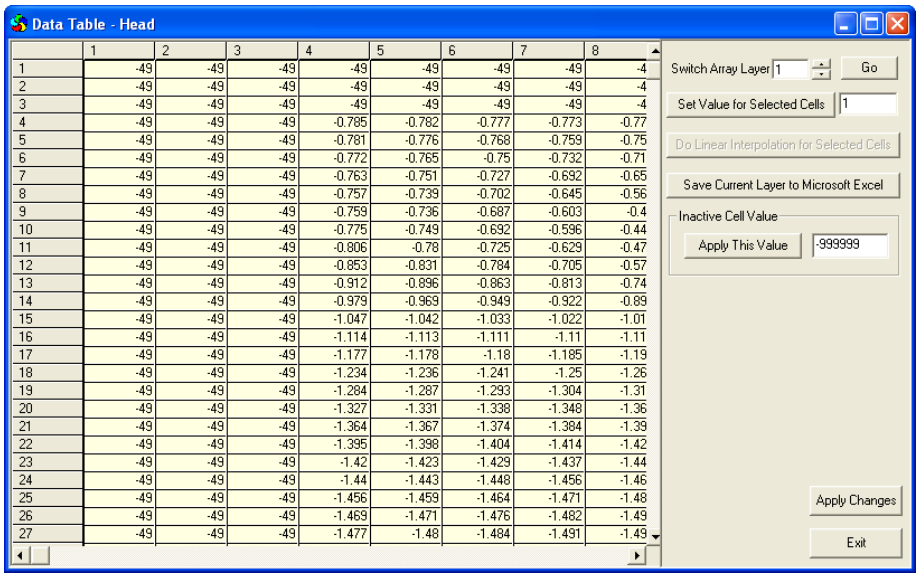


Figure 17-4 Data Table window for Editing Arrays

To close the Data Table window, click the **Exit** button. If **Apply Changes** button is not clicked before clicking the Exit button, the window closes without making any change in the original array.

> **!** The window closes on clicking **Exit** without warning/asking the user whether or not he/she wants to **Apply Changes**.

17.2.3. Importing Single Arrays

'Open' window appears on clicking 

button. The user can browse to the location of a saved *.txt file which contains the desired values for the selected array in the specific format, as shown in **Figure 17-5**. Using the correct file format, the data from the *.txt file can be imported into the currently assigned array.

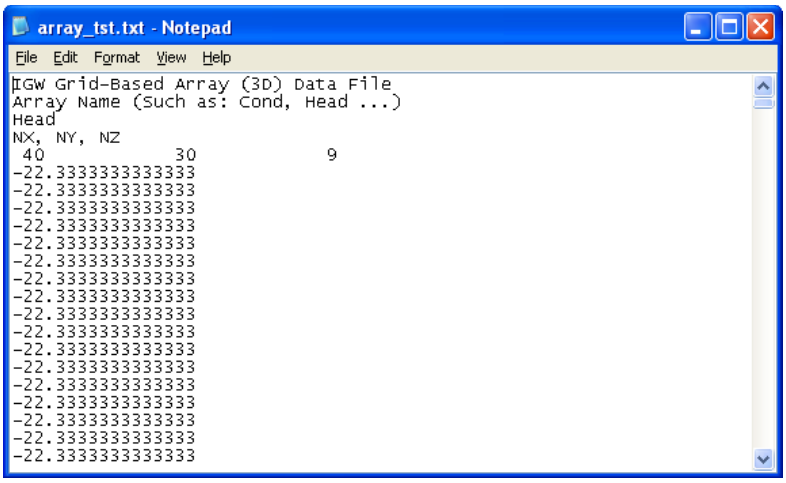


Figure 17-5 Import file format for *.txt file

The specific things to note in the format of the *.txt file are given below:

- The first and second lines in the file are text entries containing general information.
- The third line is also a text entry. It contains the 'name' of the array to which this file will be assigned. The name has to be exactly the same as that of the assigned array in the top field, else nothing will be will be imported into the assigned array.
- The fourth line contains 3 text entries, NX, NY and NZ separated by commas. These represent number of columns, rows and layers in the IGW model.
- The fifth line contains 3 numeric entries corresponding to the text entries in the fourth line, defining the array dimensions in the X, Y and Z dimensions.
- From sixth line onwards, the values for every cell of the array are tabulated as a single column. The first value represents the bottom left cell of the deepest layer in the model.

17.2.4. Exporting Single Arrays

Clicking the Export ... button allows the user to save the assigned attribute array from the grid-based distribution as a *.txt file. 'Save As' window appears. The user can save the file in a desired location.

A valid name must be supplied as no default name is given. The text file created using this feature is in the format explained above and can be imported back into an array using the **Import** function.

> **i** If the saved file has to be imported in an array which has a different name, the user should first open the file in a text editor and change the array name in the third line before opening the file using **Import** function.

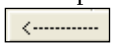
17.2.5. Creating a Temporary Array

Selecting this allows the user to create a temporary array file. The user must enter a name in the window to the right of the **Create** button and then click the **Create** button. The name of the temporary file will appear at the bottom of the **Attributes Array List**.

The user can create as many temporary arrays as he/she requires. If the **Grid-Based Operation** window is closed and opened again in the same session, the user-created array will still appear in the **Attributes Array List**. If the model is discretized, all temporary arrays disappear. Shallow discretization, however, will not remove the temporary arrays from the list. If the model is saved and closed, and then opened again, the temporary arrays will not appear in the **Attributes Array List**.

When a temporary array is created, it only contains the 3D structure of the array. It is not populated with any values. To assign values in a temporary array, the user can either use the **Import** function to upload values from a text file or use the **Two Array Operation** tools. Once populated, the temporary array can be edited using the Edit function, and can be saved in a text file using the Export function.

Populating a temporary array from a text file will involve the following steps:

- Create the temporary array using the Create function.
- Select the temporary array from the Attributes Array List.
- Using  button, assign the temporary array to the Single Array Operation area.
- Click Import and select the required file to populate the temporary array. Please be sure that the array name in the text file (line 3) matches exactly with the name given to the temporary array.

Populating a temporary array using the **Two Array Operation** tools is explained in **Section 17.6**.

17.2.6. Deleting Temporary Arrays

Selecting this button will delete user-defined temporary arrays.

> **i** Only user defined arrays can be deleted. If Delete is clicked while an attribute array is assigned, the message 'You can only delete user defined arrays!' will popup.

17.3. Head Array Operation Area

The Head Array Operation area is shown in **Figure 17-2**. Its functions are performed only (and automatically) with the existing Head array. User does not need to assign or select the Head array prior to performing these functions.

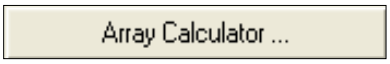
Clicking on **Copy to Calibration Head** button assigns the values in the Head array to the Calibration Head array.

Clicking on **Make Prescribed Head** button assigns Head array values as prescribed head in the model. This option is used when the stress on the hydraulic head is not changed during the analysis.

Selecting this feature allows the user to simulate drawdown in the model (i.e. in a well) using the Theis Equation. Information about this function can be found in the ***IGW Version 4.7 Reference Manual***.

17.4. Array Calculator

Clicking on the '**Array Calculator**' button opens an Array Calculator interface as shown in **Figure 17-6**.



This interface is not fully functional in *IGW Version 5.0P*. The operations/buttons shown on this interface will become functional in later versions of *IGW*.

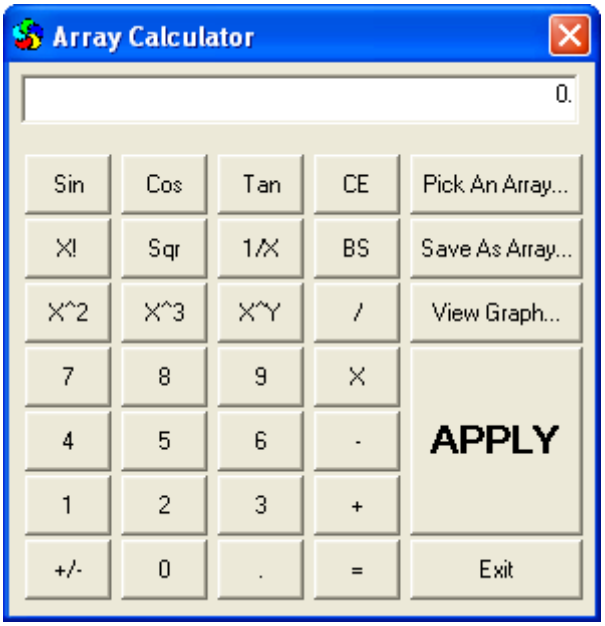


Figure 17-6 Array Calculator Interface – not functional in *IGW Version 5.0P*

17.5. Drawdown Model

In many groundwater modeling situations, the user may need to analyze the drawdown caused by pumping, recharge or river stage *changes* etc. **Create Drawdown Model** button makes it very convenient to calculate *net change* in the system due to *change* in any of the above mentioned stresses, without having to run the model sepatrately for pre and post conditions and then doing the difference to find the net change.

i In *IGW Version 5.0P*, creating a drawdown model option is only applicable for single layered confined aquifer systems

Selecting this button opens **Calculate Drawdown** window shown in **Figure 17-7**. The window gives the user the options for selecting boundary conditions for the drawdown model.

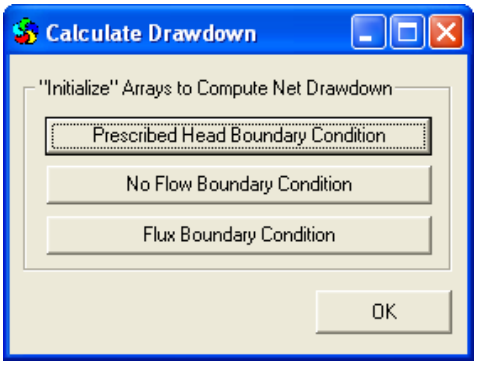


Figure 17-7 Calculating Drawdown

When user clicks on either of the first two (Prescribed Head or No-flow) boundary conditions, a message pops up, cautioning the user that 'rediscretization will be needed to recover this after change'. By clicking 'Yes', the 'Done' message pops up. The user may then run the model (without discretization). After running the model the Head array is updated to the net-draw down in the model and can be observed/saved using the Grid-Based Operations. The drawdown results can also be visualized in the Working Area using the display options. The user can experiment with different boundary types to observe their impact on the calculated drawdown.

> **i** Flux Boundary Condition is not available in *IGW Version 5.0P* to calculate drawdown. However, the user can select Recharge array from the Attributes Array List and add recharge flux at the boundaries equal to the required boundary flux to create flux boundaries.

17.6. Two Array Operations Tools

Two Array Operation area is highlighted in 'Grid-Bsed Operations' window shown in **Figure 17-2**. The left half of this area contains the operational tools while the right half only shows the general formula of operation(s). The formula is reproduced below:

$$Array1 \times A + Array2 \times B + C = Array3$$

where:

Array1, *Array2* and *Array 3* could be any arrays selected from the Attributes Array List. *A*, *B*, and *C* are any real numbers. Default values for $A = 1$, while $B = 0$ and $C = 0$.

Using the above formula, the Two Array Operation can be used to perform many operations.

From the examples given below, the user can get a fair idea of how to implement a Two Array Operation. The formula for the operation is simple and the interface is quite intuitive. The user can select arrays from the Attributes Array List and assign them in the appropriate fields corresponding to *Array1*, *Array2* and *Array3*. The user can also enter the desired values of constants *A*, *B* and *C*. To complete any operation, user must click the Apply button.

17.6.1. Copying Values from one Array to Another

If the user wants to copy, for example, values from one existing array (say, Head array) to an other existing array (say, Calibration Head array), it can be done as follows:

- Select the Head Array from **Attributes Array List** and use ----------> button to assign it as *Array1* in the **Two Array Operation** area. Do not change the value of *A*.
- Do not assign any array as *Array2*.
- Select 'Calibration Head' array from the **Attributes Array List** and use the ----------> button to assign it as *Array3*.
- Click the Apply button. 'Done' message will pop up advising the user that the operation has been completed.

This operation assigns values from Head array to Calibration Head array. **Figure 17-8** shows the Two Array Operation area for the above mentioned steps.

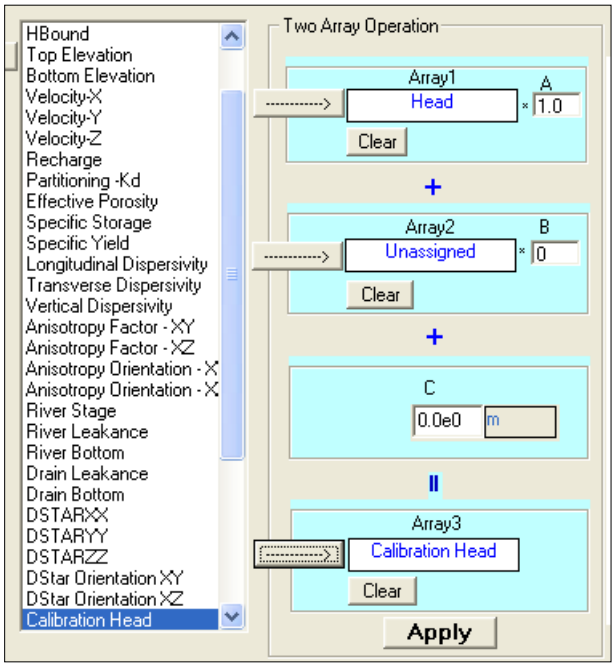


Figure 17-8 Two array operations

In order to assign values from an existing array to a new array created by the Create function, the user can follow the same steps as mentioned above. In that case, the existing array will be assigned as *Array1* and the newly created array will be assigned as *Array3.*

17.6.2. Addition and Multiplication by a Constant

Say, for example, the user wants to simulate a drought scenario by lowering the recharge rate by a constant number (0.001 m/day) all over the model area. This scenario will require the a constant number (-0.001) added to the Recharge array. To do so:

- Select 'Recharge' from the **Attributes Array List** and assign it as *Array1*
- Enter value (-0.001) for *C*.
- Again select 'Recharge' from **Attributes Array List** and assign it as *Array3*.
- Click the Apply button. . 'Done' message will pop up advising the user that the operation has been completed.

If, for example, the user wants to reduce the recharge rate to 80% of existing one. This scenario will require the a constant number (0.80) to be multiplied by the Recharge array. To do so:

- Select 'Recharge' from the **Attributes Array List** and assign it as *Array1*

- Enter value (0.8) for *A*.
- Again select 'Recharge' from **Attributes Array List** and assign it as *Array3*.
- Click the Apply button. . 'Done' message will pop up advising the user that the operation has been completed.

Figure 17-9 shows the Two Array Operation area for adding a constant to an array.

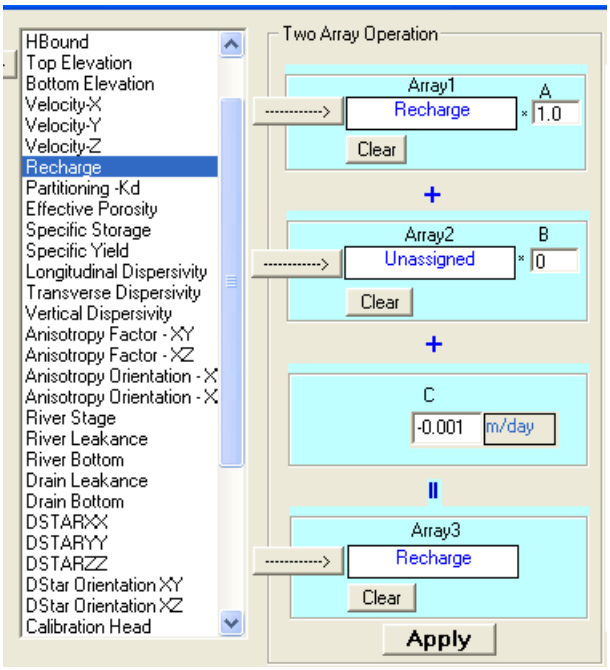


Figure 17-9 Two Array Operation – Adding constant to an array

17.6.3. Addition and Subtraction of Arrays

For adding two arrays, assign one of the arrays as *Array1* and the other as *Array2*. For values of constants, enter *A*=1, *B*=1 and *C*=0. Assign the an array as *Array3* which will contain the result. Click apply.

> **R** To make sure that an operation is successfully performed, assign the array in *Array3* filed to the **Single Array Operation** area, and click **Edit** to see if it contains the desired result.

For subtracting one array from the other, assign the array from which the other array is to be subtracted as *Array1* and the one being subtracted from the first as *Array2*. For values of constants, enter *A*=1, *B*=(-1) and *C*=0. Assign the an array as *Array3* which will contain the result. Click apply

> **R** Before performing a **Two Array Operation**, it is a good idea to create a new array using **Create** function. This new array can be assigned as Array3 in the operation.

17.6.4. Backing up an Array

The user can utilize Grid-Based Operations backing up any of the attribute arrays in the list. For example, hydraulic head can be backed up by using the following steps:

- Create a temporary file name (say, **head-backup**) in which the hydraulic head array will be copied;
- Highlight the temporary file name in the **Attributes Array List**, and assign it to the *Array3* field;
- Highlight the Head array in the **Attributes Array List** and assign it to the *Array1* field.
- The transfer of the Head array to the **head-backup** array is completed by selecting the Apply button.
- The user can then select **head-backup** array from the **Attributes Array List** and assign it to the **Single Array Operation** area.
- Using the **Export** function, the user can save the **head-backup** array in a file.

!	In order to implement the changes in grid based operations, there is no need to discretize the model. It is important to understand that all changes related to the grid-based operations will be eliminated if the user discretizes the model.

R	Grid-based operations are very useful for sensitivity analysis. In order to observe the effect of a parameter (e.g. hydraulic conductivity) on the groundwater model, the user is recommended to perform array operations on that particular parameter while they are keeping the other parameters constant.

Chapter 18 STOCHASTIC MODELING

IGW offers the ability to perform stochastic modeling. Stochastic modeling is a way to examine the many possibilities of heterogeneity that share the same geo-statistical structure. Using specified statistical parameters, possible aquifer formations are randomly generated and solutions obtained for each.

Number of realizations generated in a stochastic modeling process is one of the key factors to make better estimates of probability distribution of various modeling out comes. The larger the number of realizations, the better the probability estimates would get. Taking full advantage of computing power within a network, *IGW Version 5.0P* gives the user the option/capability to simultaneously employ all (or a number of) machines and/or processors available in a net work. This is termed as 'parallel computing'. Parallel computing features of *IGW Version 5.0P* are discussed in detail in **Section 18.7**.

The following sections discuss the implementation and methodology associated with **IGW** stochastic modeling.

18.1. Prerequisites

In order to perform meaningful stochastic modeling, there needs to be at least one model parameter that has been defined either as:

1) A random field (associated with a zone – defined in the **AE**), or as
2) A set of scatter points (associated with a zone) with simulation selected to define the random field. When scatter points are used to define a random field, the user has the option to perform conditional or unconditional simulations (please see **Section 18.4**)

> **i** A greater number of parameters can be associated with scatter points (as many as can be defined for a zone). Scatter point implementation is discussed in **Section 7.7** and **Chapter 20**.

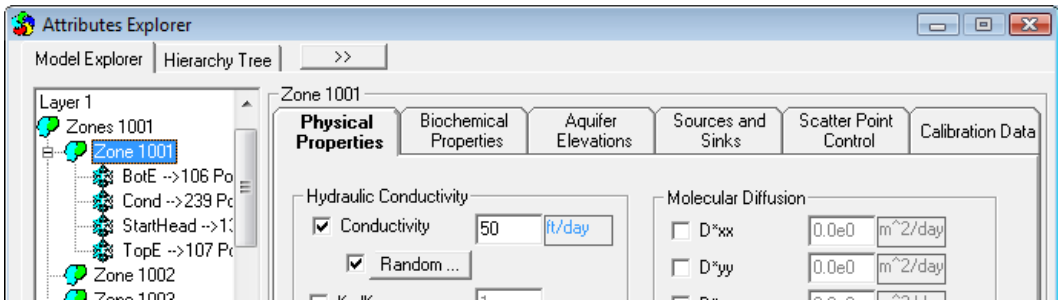
When either of the aforementioned settings is implemented for a parameter, the software will be able to generate any number of statistically equivalent random field formations for each. This is the basis for single realization and Monte Carlo stochastic modeling.

18.2. Setting up a Parameter for Stochastic Simulation

> **i** Currently, random fields can only be specified for hydraulic conductivity. Random fields for effective porosity, the partition coefficient, and recharge will be developed in later versions of IGW. These are discussed in the appropriate subsections of **Section 7.6.1**.

Follow these steps to setup a parameter as random field:

- Select the zone within which there exists a random field i.e., hydraulic conductivity;
- Open 'Physcial Properties' tab in the **AE** window;
- Check the 'Conductivity' box and enter the mean value for conductivity in the placeholder;



- Check the box before the 'Random' button and then click on the 'Random' button; this will open the 'Option of Unconditional Random Field (Attr.)' window as shown in **Figure 18-1**. Here, the user can specify correlation scales (LambdaX, LambdaY etc.), theoretical variance, rotation angle and nugget. The user can also select the algorithm and model type to generate random field. Click 'OK' when finished with appropriate selections have been made and required values entered.

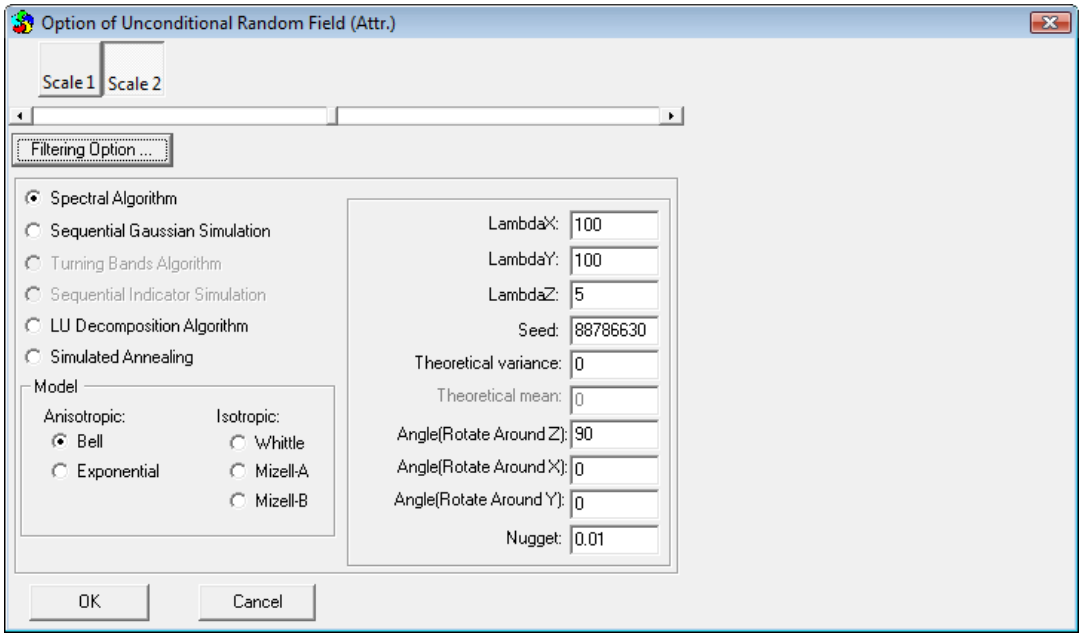


Figure 18-1 Option of Unconditional Random Field (Attr.) window

Going through the above steps will set up hydraulic conductivity for stochastic simulations. To run the stochastic model, the user will discretize the model and go to the 'Solver Settings' as explained in the next section.

18.3. Solver Settings

The **Stochastic Model** tab of the **Solver Settings** window (**Figure 13-7**) is the interface used to activate stochastic modeling. It can be accessed through 'Numerical Solver Setting' button and clicking on the **Stochastic Model** tab. The user may select the modeling method as either **Single Realization** or **Monte Carlo Simulations**.

i	The model must be discretized before selecting the solver setting.

18.3.1. Select Simulation Methods

In this area, the user selects the stochastic modeling method. Available methods are discussed in the following subsections.

SINGLE REALIZATION

This is the default selection. When this is selected, the software operates in one of two ways. If no parameters are defined statistically (as described in **Section 18.1**), the software will operate in mean (deterministic) mode. However, if one or more parameters have been assigned a random field or determined from a scatter point based simulation, then the software will operate in single realization mode.

In single realization mode, the software generates only one model realization. In this realization, each statistically defined parameter is assigned an independent random variable field instead of being assigned a spatially averaged value (as would be the case in mean mode). Note that there is no difference in software interface or function to distinguish the modes from each other and that the specialized stochastic displays discussed in **Section 18.5** are not applicable to this mode.

MONTE CARLO SIMULATIONS

The Monte Carlo mode is an extension of the single realization mode. If this method is selected, the software sequentially generates any number of model realizations that are statistically equivalent. Based on these multiple realizations, the user may examine a variety of parameters (related to the entire set) such as probabilities, mean values, and correlations of solution parameters through the advanced stochastic modeling display options discussed in **Section 18.5**.

Clicking the Options ... button opposite to **Monte Carlo Simulations** in '**Model 1: Solver Settings**' window (**Figure 13-7**) opens the **Monte Carlo Simulation Settings** window (**Figure 18-2**).

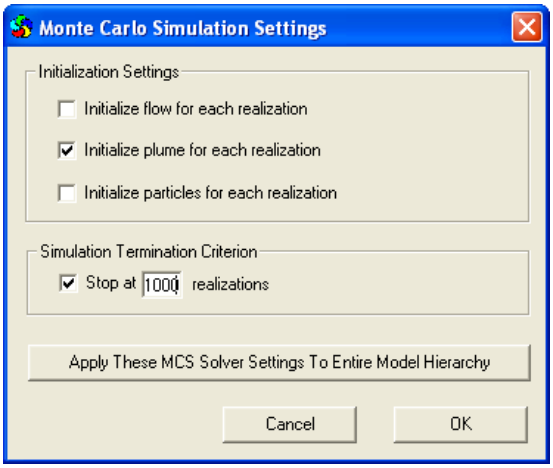


Figure 18-2 Monte Carlo Simulation Window

Clicking the **OK** button closes the window and sets the changes in the software. Clicking **Cancel** closes the window and ignores any changes. Other options in this window are discussed as follows:

INITIALIZATION SETTINGS

These options allow the user to manually initialize any combination of the plume, flow or particles for each realization.

SIMULATION TERMINATION CRITERION FIELD

In this field the user may specify the total number of model realizations to be generated and solved. This field is only active when the '**Stop at**' box (see below) is checked. If

unspecified, the software will continue to generate and solve model realizations until it is stopped by the user.

'STOP AT' CHECK BOX

Checking this box specifies the software to terminate the Monte Carlo simulation process when the number of realizations specified in the **Realizations** field is reached. When the box is unchecked, the software will keep generating realizations until the Stop/Pause Model button is pressed.

APPLY THESE MCS SOLVER SETTINGS TO ENTIRE MODEL HIERARCHY

This feature will only work with models having multiple layers. Clicking the button will allow the software to apply all settings entered, for all layers in the model.

18.3.2. Running Monte Carlo Simulations

After selecting the desired simulation settings, the user can click 'OK' at the bottom of '**Model 1: Solver Settings**' window (**Figure 13-7**). This will set the model to run in stochastic mode.

User can start the Monte Carlo simulations by pressing 'Run Model Forward' button. The simulations will stop when the specified number of simulations in the 'Stop At' box is reached and/or by clicking the 'Pause/Stop' button.

All model features should be added before running the model. This is because the Monte Carlo simulation is based on multiple statistical realizations of the same conceptual model, and therefore no changes should be made to the model during the stochastic modeling process. *IGW Version 5.0P* protects against this by disabling most of the buttons on the Button Palette (**Figure 18-3**) as soon as the model is set into stochastic mode.

If changes need to be made to the model, the user will first have to return the model solver to single realization. This is done by clicking the **Numerical Solver Settings** button to open the **Solver** window (**Figure 18-4**) then selecting **Single Realization Simulation**, and then clicking the **OK** button.

Figure 18-3 Buttons Palette during stochastic simulation mode

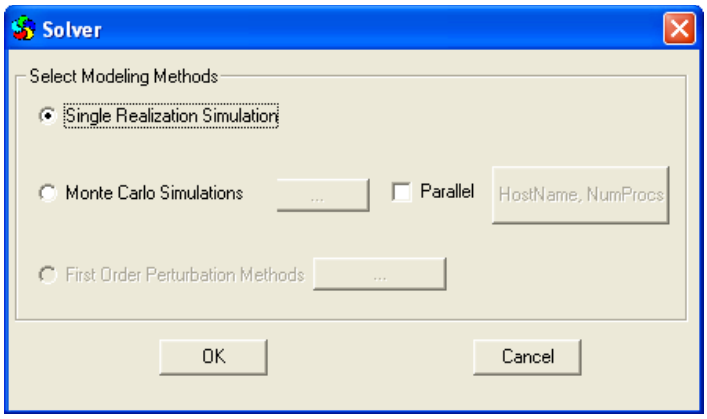
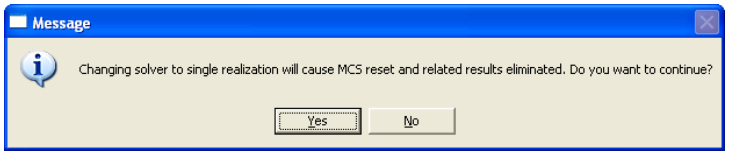


Figure 18-4 Solver Window



Getting out of Monte Carlo Simulations mode will discard all the results of the previous Monte Carlo simulation. As soon as the user clicks OK in the Solver window, a warning message appears to warn the user.

Once the model is out of Monte Carlo simulation mode, changes can be made and then Monte Carlo simulation reinitialized.

18.4. Conditional and Unconditional Simulations

Statistical parameters for the random field can be inferred from data. The data is generally available at specific point locations and can either be entered in the model as scatter points (**Section 7.7**) or directly imported from GIS files (**Section 22.5**). Please also refer to **Section 7.7.6.2** and **Sections 20.3.2.3** and **20.3.2.4** for unconditional and conditional simulations as a means of interpolation of data.

18.4.1. Unconditional Stochastic Simulations

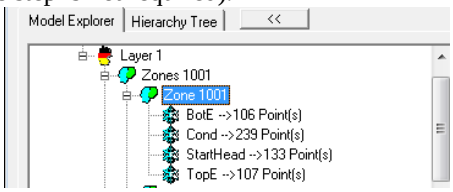
An unconditional simulation is one in which the simulated field is not constrained to pass through the (known) data points, but honoring the covariance of the variogram (instead of the explicit data values). The **Unconditional Simulation** procedure generates a spatially-correlated random field based on sample statistical parameters. This procedure will generate new values for the locations corresponding to the measured values.

Unconditional simulations are similar to the one explained in Section 18.2, except that instead of the user entering the statistical parameters in 'Option of Unconditional Random Field (Attr.)' window, the software automatically calculates and assigns these parameters from the data available at scatter points.

To perform unconditional simulations, follow these steps:

- Select the zone within which scatter points are added with random data attribute, i.e, hydraulic conductivity;

- Open the AE window and right click on the selected zone and click 'Switch List' so that the scatter points appear as shown on the right (if AE window already show the point in this format then this step is not required).



- Select 'Cond -- > *nnn* Point(s)'
- The corresponding RHP will appear in the AE as shown in **Figure 20-11**.
- Check the 'Unconditional Simulation' button and choose a 'Simulation Method' in the list.
- In the 'Variogram Model' area, if 'User specified' button is checked and then the 'Options' button is clicked, the 'Random Field Options' window (**Figure 18-6**) will appear. Here the user can manually specify statistical parameters.
- In the 'Variogram Model' area, if 'Infer from Data' button is checked and then the 'Edit' button is clicked, the 'Variogram' window (**Figure 18-7**) will appear. Here the user can select/built a variogram model based on data. When satistifed with variogram model, click OK. Statistical parameters of the variogram model are automatically assigned to the unconditional simulation. Please refer to **Section 20.4** for more on variogram modeling.
- Before running the unconditional simulation, discretize the model and then follow the procedure given in **Section 18.2** .

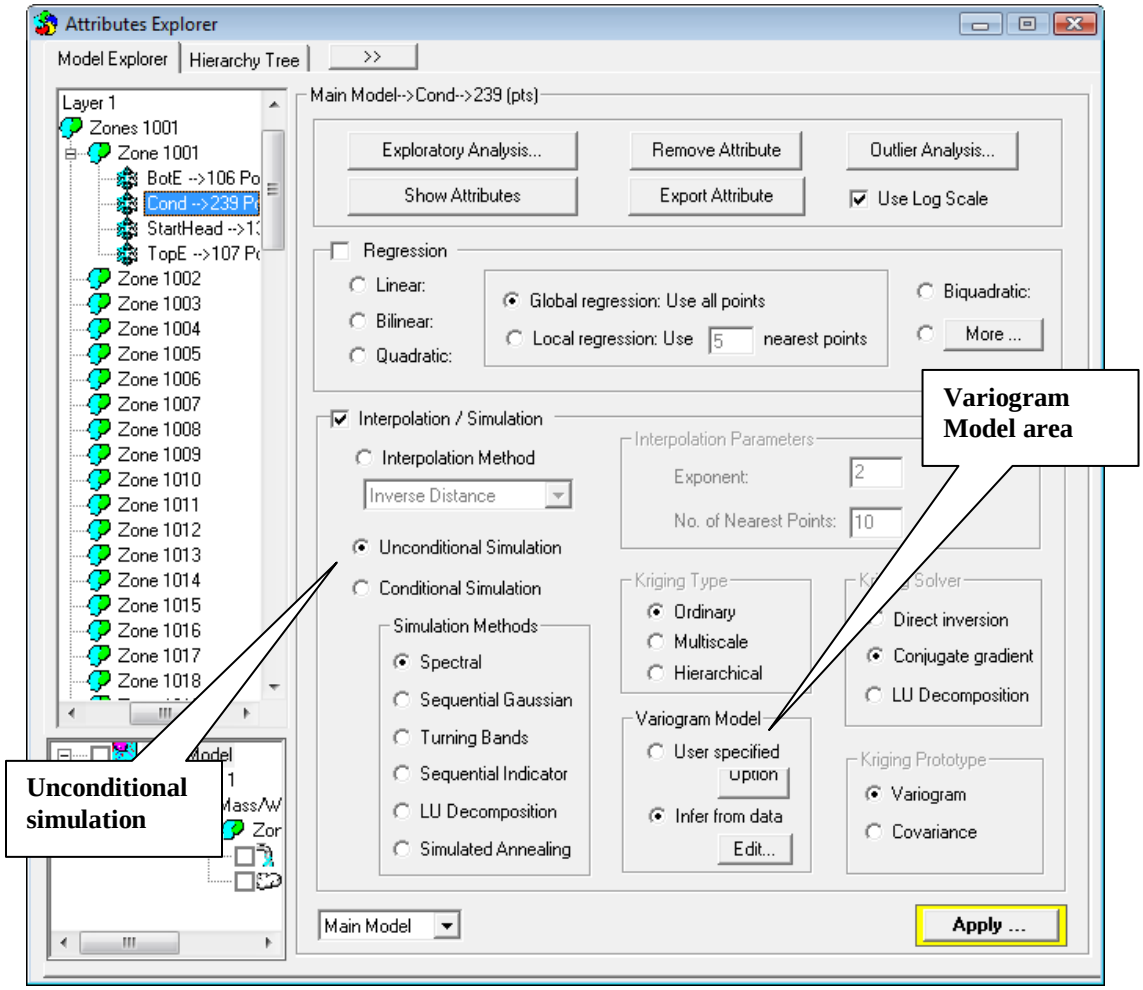


Figure 18-5 Options for unconditional simulation

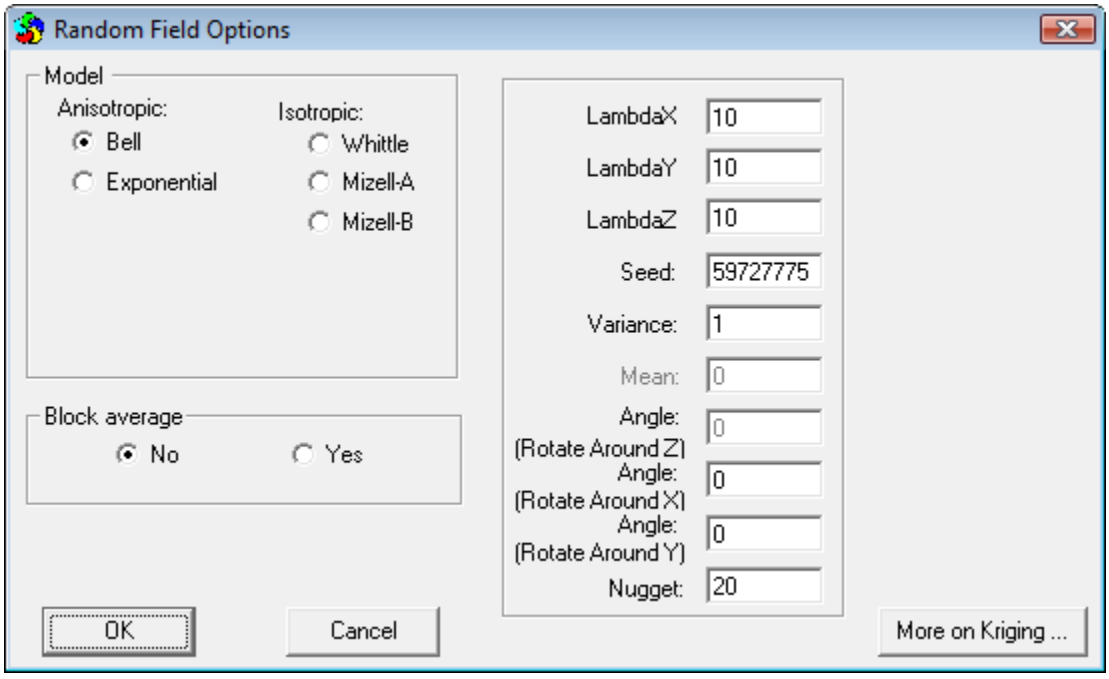


Figure 18-6 Random Field Options window

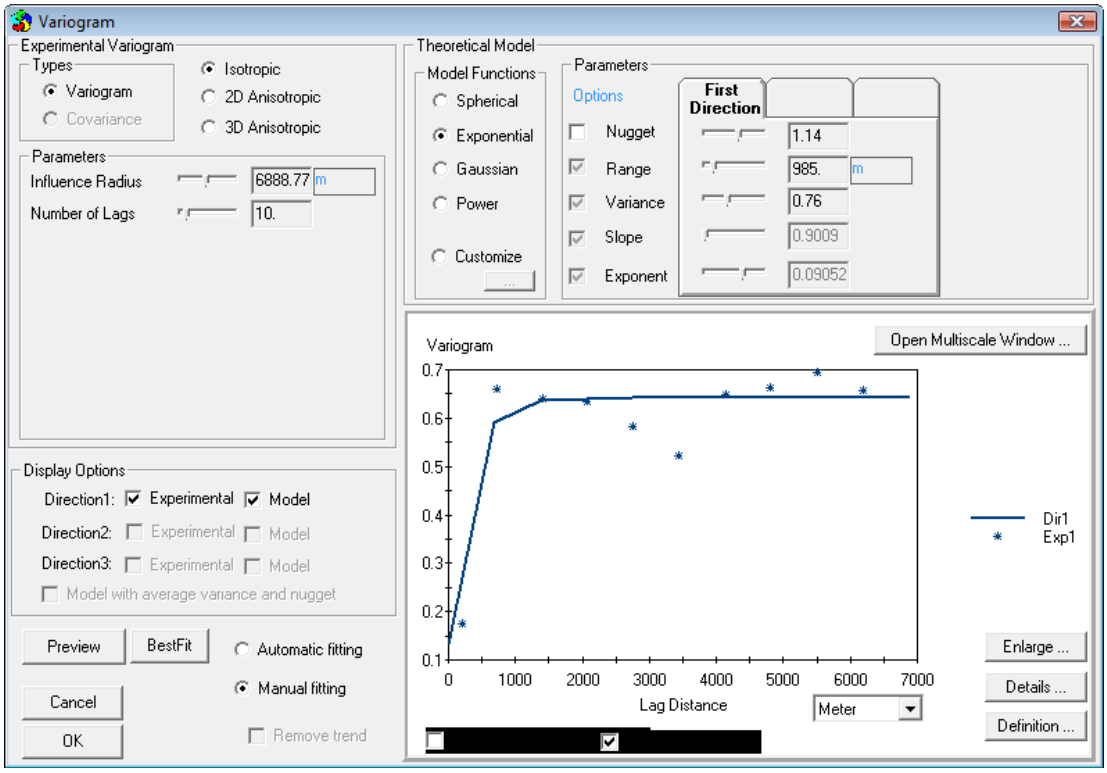


Figure 18-7 Variogram window

18.4.2. Conditional Stochastic Simulation

A conditional simulation is the one in which simulated field is forced to pass through known data points. This honors both the data and the statistics. The **Conditional Simulation** procedure generates a spatially-correlated random field based on sample statistical parameters. This

procedure is similar to the **Unconditional Simulation** method, except the values for the locations corresponding to the measured values are held equal to those values when the field is generated.

There are two options available in the **Simulation Methods** area when this procedure is selected:

1) **Spectral Algorithm** (the default), and
2) **Sequential Gaussian Simulation**.

18.5. Display Options/Monitoring

There are a number of display and monitoring options that are available with respect to stochastic modeling in *IGW Version 5.0P*. These options allow the user to visualize real-time stochastic simulations and results as the simulations go. The options are discussed in the following subsections.

18.5.1. Main Model Display Options

After desired stochastic modeling mode has been selected (see **Section 18.3**), clicking the **Set Display Options** button opens the display settings for the main model. These options are exactly the same as discussed in **Section 19.1.** These options apply to the model display in the main Working Area of *IGW Version 5.0P* interface.

The user should note that when the display options are accessed in Monte Carlo Simulations mode, the '**Monte Carlo Simulation Results**' area becomes active. This area has two buttons, viz., 'Means and Variances …' and 'Realizations …' as shown in **Figure 18-8**. 'Realizations…' button is not active in *IGW Version 5.0P*. 'Means and Variances…' button is discussed in the next section.

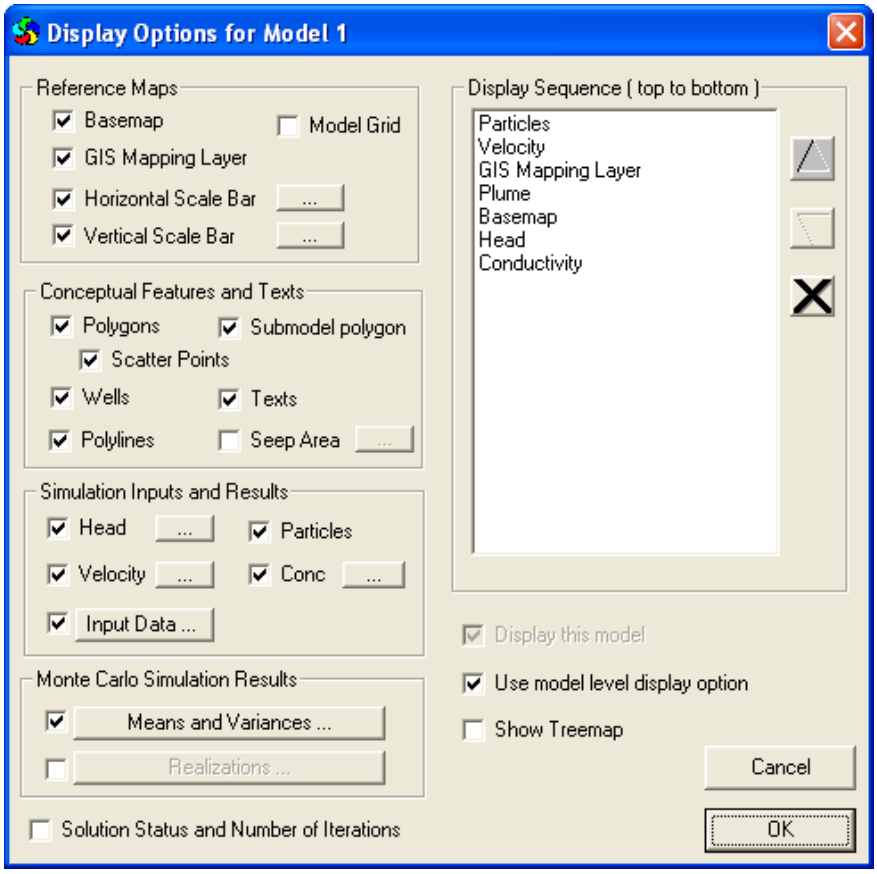


Figure 18-8 Display Options for Monte Carlo Simulations

18.5.2. Stochastic Display Options

Stochastic simulations in *IGW Version 5.0P* allow user to observe statistical nature of various processes as the realizations go. The user can observe mean spatial patterns of processes in the model areas. The user can also observe specific processes at selected points (monitoring bores) and across selected linear objects (polylines).

R It is a good idea to set all the display options before starting the Monte Carlo simulations, i.e., before running the model. However, the user can also pause the simulations any time and set/reset the display options.

18.5.2.1. Observing Spatial Processes

By checking the box to the left of 'Means and Variances..' button in 'Display Options' window in **Figure 18-8** and clicking at the button opens the '**MCS Means & Variances**' window as shown in **Figure 18-9**.

i The specific parameters that are available in this window are set in part based on the selections made in the **Select Model Parameters** area on the **Stochastic Model** layer on the **Solver Engine** window (see **Chapter 13**).

A check mark indicates that visualization is activated for that specific parameter. The user may select or deselect these as desired. By default **Mean Velocity**, **Capture Zone**, **GIS Mapping Layers**, **Mean Concentration**, and **Mean Head** are active.

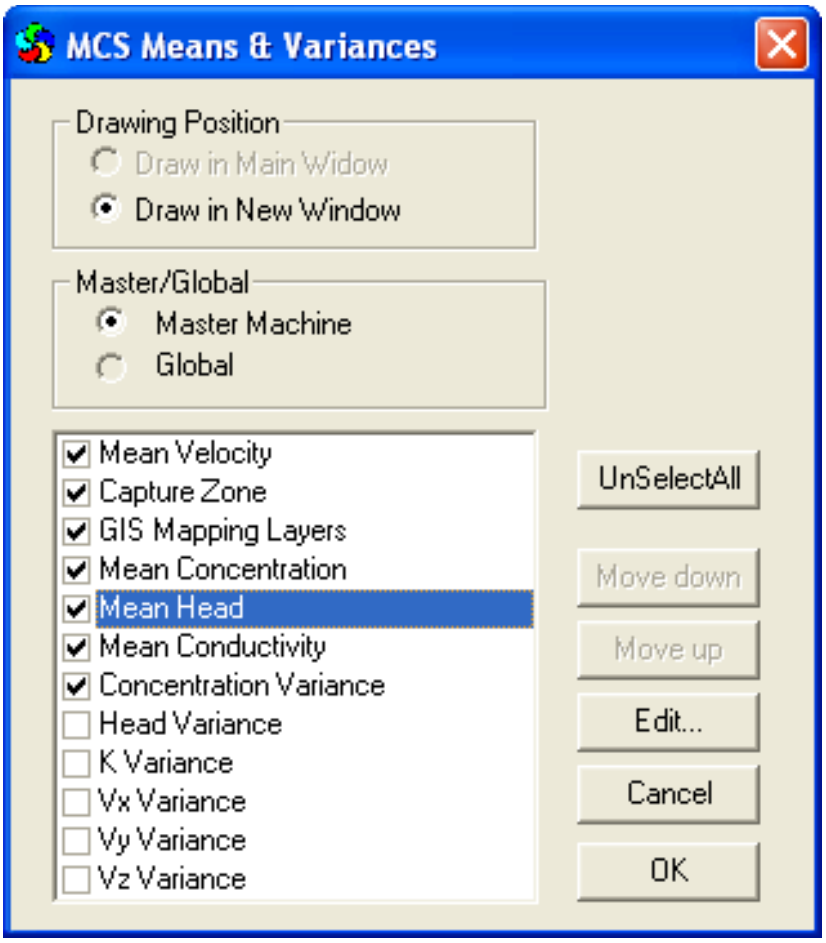


Figure 18-9 Visualizing Variance and Other Model Parameters/Results

In the **Drawing Position** area, the user currently may only choose to display these parameters in a new window (**Draw in New Window** – the default).

By single clicking on any of the parameters in the 'MCS Means & Variances' window, the parameter will be highlighted. Clicking on the 'Edit' button to the left will open the 'Display Options' window for that parameter as shown in **Figure 18-10**. This window allows the user to select the display configuration for the selected parameter. The functionality of this window is explained in **Section 19.4.1**.

'Move down' and 'Move up' buttons on this window are not active in *IGW Version 5.0P*. The selected parameters are displayed in layers. The sequence/order of layers is the same as it appears in the window (e.g. Mean Velocity is always displayed at the top, followed by Capture Zone and so on). If some representations are hidden beneath another, the user can deselect the top layers to see the bottom ones.

After the user is finished with the selection of parameters in 'MCS Means & Variances' window, the user can click 'OK', and then 'OK' again in the 'Display Options' window for main model.

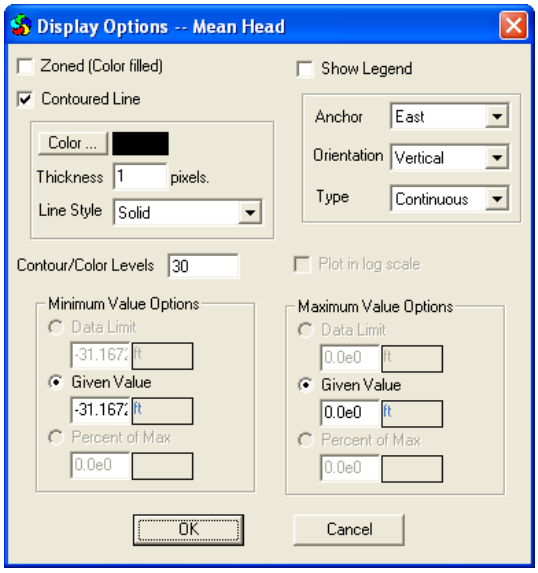


Figure 18-10 Display Options for spatial parameters

The user can start the model simulations after the display settings are done. As soon as the first realization is complete, the model display also appears in a separate window called the '**MCS Field Statistics'** window as shown in **Figure 18-11**.

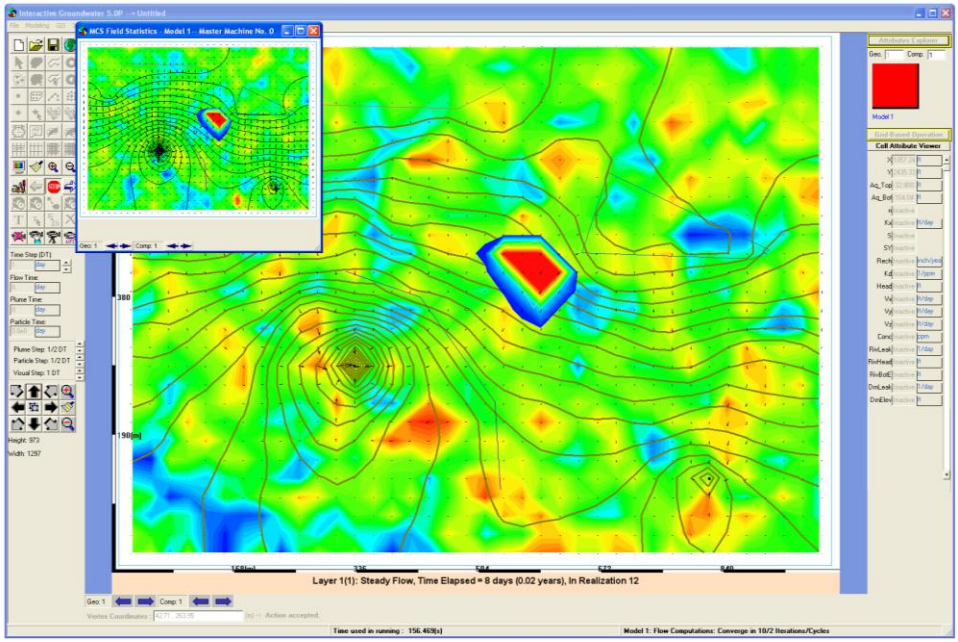


Figure 18-11 MCS Field Statistics window and the main model display

The display in 'MCS Field Statistics' window is updated after every realization. For example, in **Figure 18-11**, the main model displays the results of 12th simulation whereas the 'MCS Field Statistics' window displays the mean of these 12 realizations for velocity, concentration, head and conductivity etc.

Display in the 'MCS Field Statistics' window can also be modified by the user by simply right clicking anywhere within the window and selecting 'Draw Options' from the pop-up menu as shown in **Figure 18-12**. This will lead the user back to 'MCS Means & Variance' window in which user can select a parameter and click edit to change draw options (**Figure 18-9** and **Figure 18-10**).

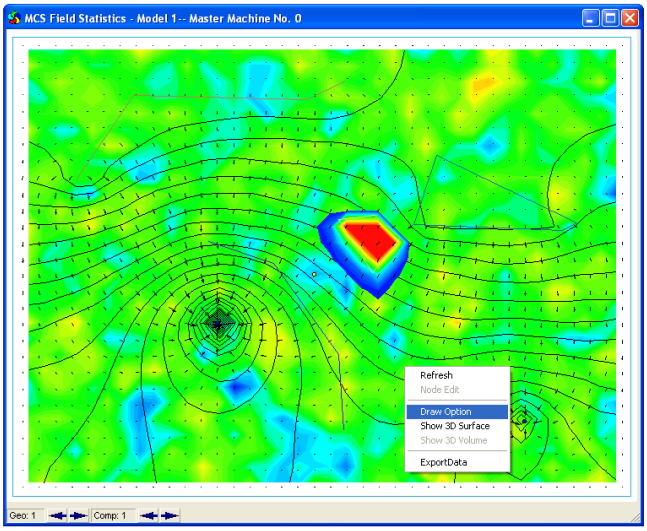


Figure 18-12 Draw options for MCS Field Statistics window

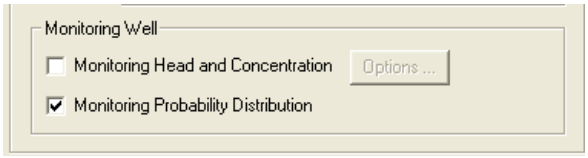
18.5.2.2. Observing Point Processes

To observe stochastic processes at a point the user can define a monitoring well at that point. *IGW Version 5.0P* allows to user to observe stochastic nature of hydraulic conductivity (lnK), head (Head) and concentration (Conc) at the monitoring well location (please also refer to **Section 9.4** for information on monitoring wells).

To observe stochastic processes at a monitoring well, follow these steps:

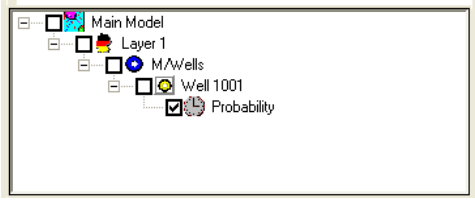
(1) Define a monitoring well at the desired location of interest in the model (see 'Monitoring Well' in **Section 9.4**).

(2) Select the 'Monitoring Probability Distribution' box in the wells' RHP of AE.



(3) Discretize the model (see **Chapter 12**).

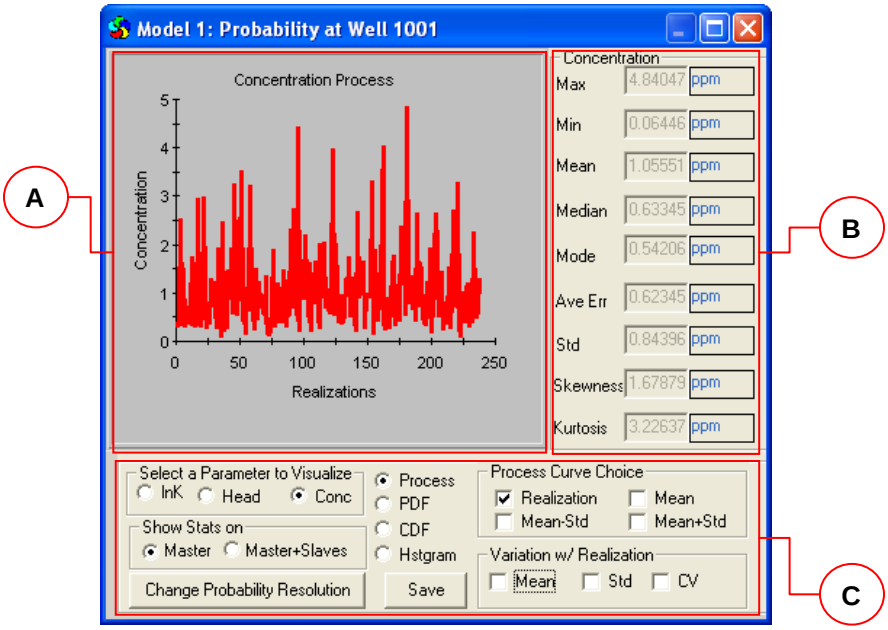
(4) Check the 'Probability' box in TPS (see **Section 4.1.3**).



(5) Click 'Numerical Solver Settings' button  , open **Stochastic** tab (**Figure 13-7**) from the '**Model 1 : Solver Settings**' window and select '**Monte Carlo Simulations**'.

(6) Run the model.

After second realization, '**Model 1 : Probability at Well xxxx**' window appears as shown in **Figure 18-13**. This window keeps updating after every realization till the simulations are stopped.



A – Graphical Display Area

B – Statistical Parameter Area

C – Display Options Area

Figure 18-13 Window for monitoring stochastic process at point locations (monitoring wells)

The window in **Figure 18-13** can be divided into three main areas as shown. These areas are discussed below.

GRAPHICAL DISPLAY AREA

Based on the display options selected in the Display Options area, this area shows the selected process graphically. The graphical representation is dynamically updated at the end of each realization. Right anywhere in this area will open the '2D Char Control Properties' window (**Figure 14-5**) which is explained in **Section 14.3**.

STATISTICAL PARAMETERS AREA

For the parameter selected (lnK, Head or Conc) in the Display Options area, the Statistical Parameters area displays the listed statistics for that parameter. The name of the parameter is also displayed at the top of the area. The statistics are updated dynamically at the end of each realization.

These statistical parameters in this area include max, min, mean, median, mode, average error, std deviation, skewness and kurtosis. These statistical parameters are defined in **Appendix E-II-I** (except average error (**Avg Err**) and **Kurtosis**).

Ave Err is the average error or average difference between each value in a set and the mean.

Kurtosis is a measure that describes the tails for a given distribution. It is defined as:

$$kurtosis = \frac{\sum_{i=1}^{N}(x_i - \mu)^4}{(N-1)\sigma^4} - 3 \qquad (18.3.2.2.2)$$

$$where\ N = total\ number\ of\ data\ in\ a\ set,$$
$$x_i = a\ datum\ in\ the\ set,$$
$$\mu = mean\ of\ data,\ and$$
$$\sigma = standard\ deviation\ of\ data.$$

DISPLAY OPTIONS AREA

The display options area is subdivided into more areas. These are explained below:

Select a Parameter to Visualize allows the user to choose between lnK, Head or Conc. Only one of the stochastic processes can be selected at a time.

Show Stats on allows the user to choose whether to display stochastic process from the Master machine only or include the Slaves machines as well. Master and Slave machines are employed in parallel computing and are discussed in **Section 18.7**.

Clicking on the Change Probability Resolution button opens the 'Subdivisions between Min/Max' window as shown in **Figure 18-14**. This window allows the user to choose the resolution or number of bars for the PDF, CDF and Histogram displays for the selected stochastic process in the Graphical Display area.

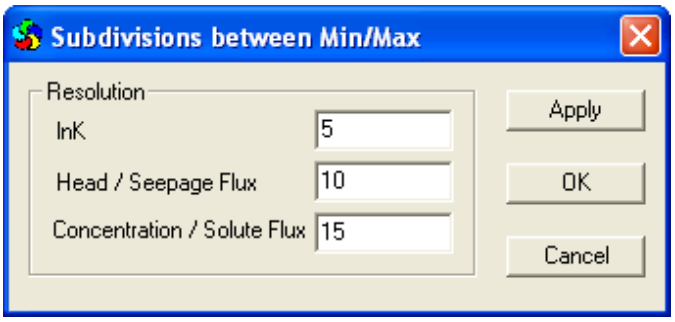


Figure 18-14 Probability resolution for PDF, CDF and Histograms for various parameters

In the middle of the Display Options area, the user can choose to display the Process, the PDF, the CDF or the Histogram. Only one of these displays can be selected at a time. **Figure 18-13** shows the process for concentration. PDF, CDF and Histogram for concentration are shown in **Figure 18-15**. Also note that when PDF, CDF or Hstgram is selected, the **Process Curve Choice** and **Variation w/ Realization** areas become inactive.

The Save button allows the user to save the data generated in all the realizations for lnK, Head and Conc. The data is saved in a *.dat file format. The format is explained in **Section 23.6.1**.

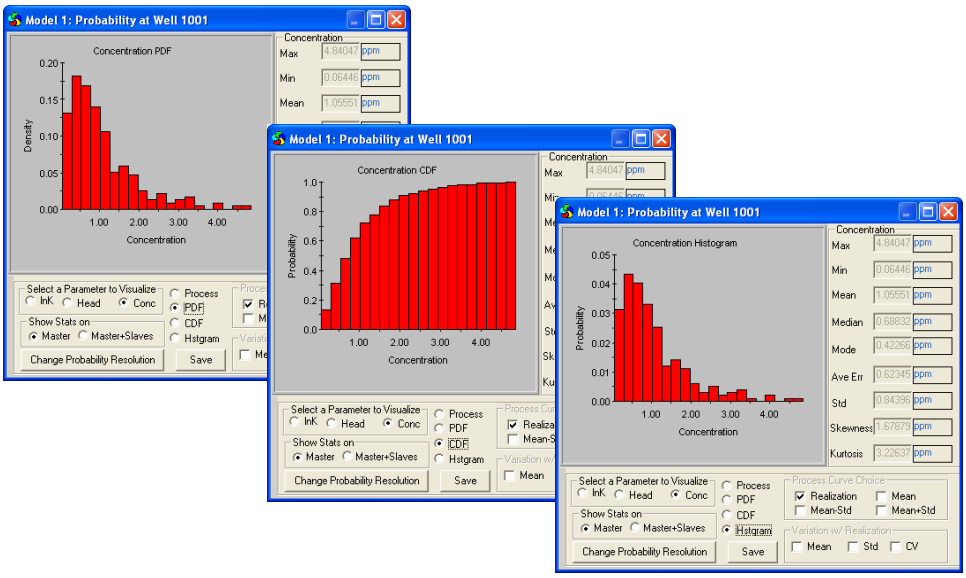


Figure 18-15 PDF, CDF and Histogram for concentration process

Process Curve Choice area allows the user to plot the stochastic process. It also allows to plot the dynamic display of mean value of a parameter at the end of last realization, along with the bounds around the mean at plus and minus of one standard deviation. In the Graphical Display area, one or all of the choices can be plotted simultaneously. **Figure 18-16** shows all three parameters (lnK, Head and Conc) with all of their process curves selected simultaneously.

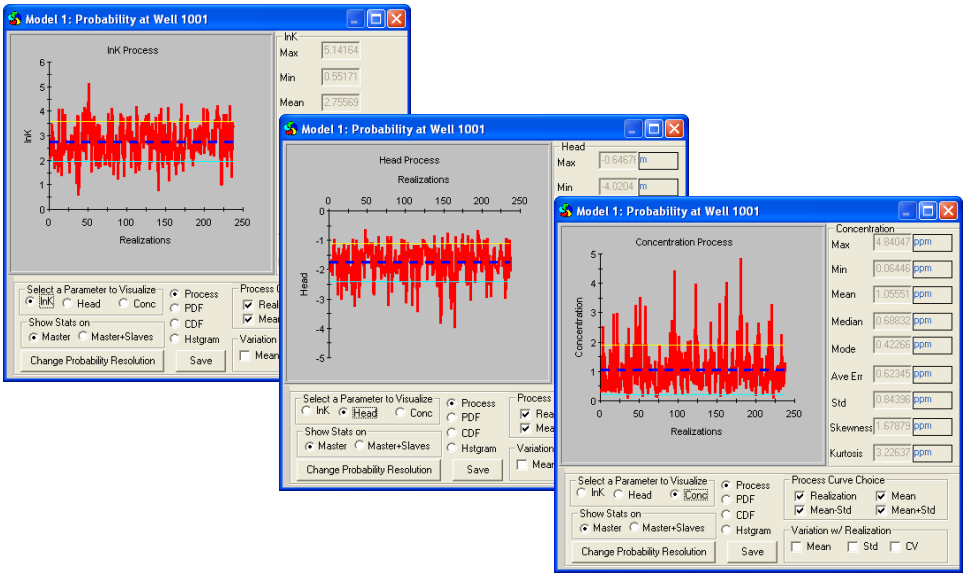


Figure 18-16 Process Curves for stochastic parameters

Variation w/ Realization area allows the user to observe the variation in **Mean**, standard deviation (**Std**) and coefficient of variation (**CV**) at end of each realization. Variation in mean and standard deviation can be plotted simultaneously. Also, the same plot can include all the curves selected in the **Process Curve Choice** area. When **CV** box is checked, no other curve can be seen except that of the **CV** in the **Graphical Display** area.

The choice of plots in Variation w/ Realization area are useful in determining whether or not enough of realizations have been generated to have statistically more meaningful modeling results. This aspect is discussed further in **Section 18.7**.

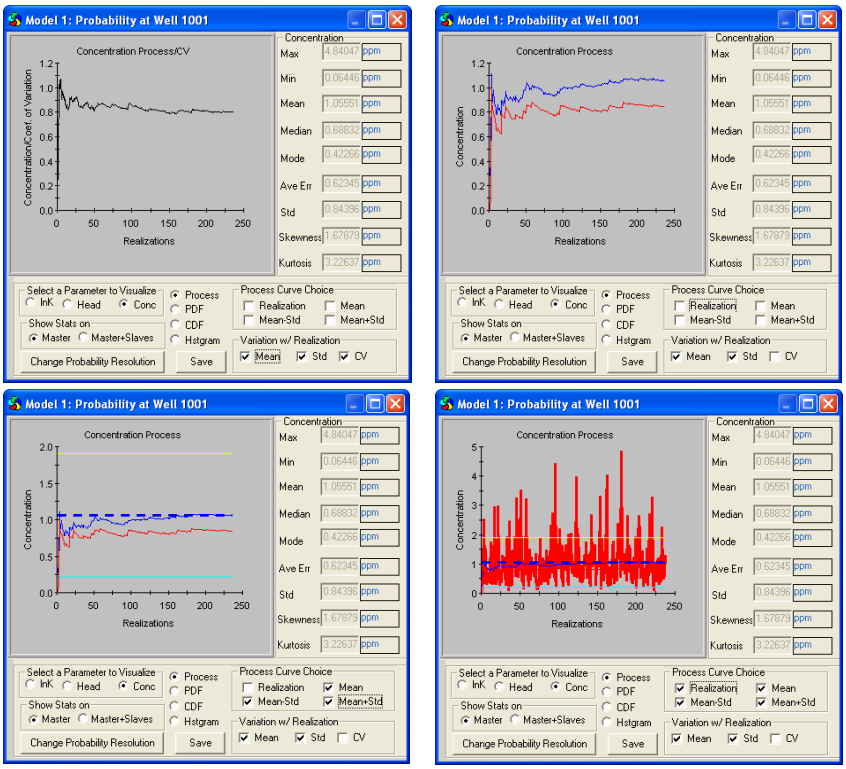


Figure 18-17 Various displays of stochastic processes

Figure 18-17 shows different plots for Conc parameter. Please note the selection of check boxes in **Display Options** area and relate them to the curves on the plots.

18.5.2.3. Observing Fluxes Across Polylines

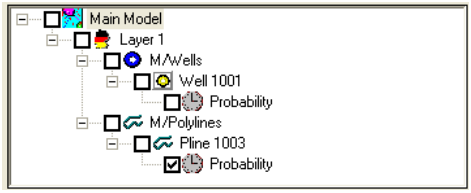
To observe stochastic nature of fluxes across a line, the user can define a polyline at the desired location in the model and assign its attributes as explained in **Section 8.5.4**. The line does not have to be a straight one. *IGW Version 5.0P* allows to user to observe stochastic nature seepage flux and solute flux across the line.

To observe stochastic fluxes across a polyline, follow these steps:

(1) Define a polyline and set its attributes for calculating fluxes across (**Section 8.5.4**).

(2) Discretize the model (see **Chapter 12**).

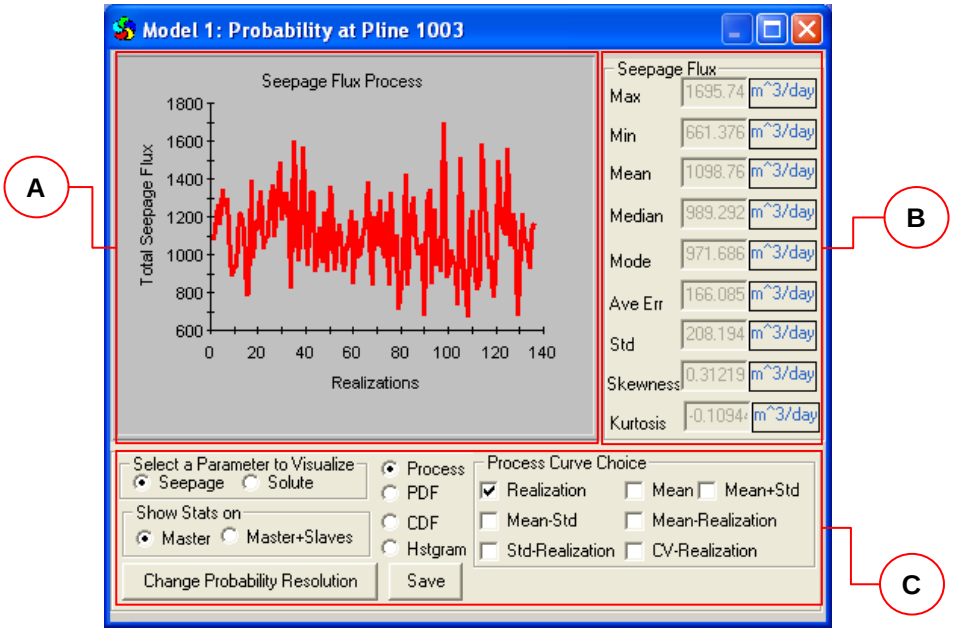
(3) Check the 'Probability' box in TPS for 'Pline xxxx' probability (see **Section 4.1.3**).



(4) Click 'Numerical Solver Settings' button  , open Stochastic tab (**Figure 13-7**) from the '**Model 1 : Solver Settings**' window and select '**Monte Carlo Simulations**'.

(5) Run the model.

After second realization, '**Model 1 : Probability at Pline xxxx**' window appears as shown in **Figure 18-13**. This window keeps updating after every realization till the simulations are stopped.



A – Graphical Display Area

B – Statistical Parameter Area

C – Display Options Area

Figure 18-18 Window for monitoring stochastic process at point locations (monitoring wells)

The window in **Figure 18-13** can be divided into three main areas as shown. These areas are discussed below.

GRAPHICAL DISPLAY AREA

Based on the display options selected in the Display Options area, this area shows the selected process graphically. The graphical representation is dynamically updated at the end of each realization. Right anywhere in this area will open the '2D Char Control Properties' window (**Figure 14-5**) which is explained in **Section 14.3**.

STATISTICAL PARAMETERS AREA

For the parameter selected (Seepage or Solute) in the Display Options area, the Statistical Parameters area displays the listed statistics for that parameter. The name of the parameter is also displayed at the top of the area. The statistics are updated dynamically at the end of each realization.

These statistical parameters in this area include max, min, mean, median, mode, average error, std deviation, skewness and kurtosis. These statistical parameters are defined in **Appendix E-II-I** (except **average error** and **kurtosis**). Avg Err and Kurtosis are defined in **Section 18.5.2.2**.

DISPLAY OPTIONS AREA

The display options area is subdivided into more areas. These are explained below:

Select a Parameter to Visualize allows the user to choose between Seepage or Solute. Only one of the stochastic processes can be selected at a time.

Show Stats on allows the user to choose whether to display stochastic process from the Master machine only or include the Slaves machines as well. Master and Slave machines are employed in parallel computing and are discussed in **Section 18.7**.

Clicking on the Change Probability Resolution button opens the 'Subdivisions between Min/Max' window as shown in **Figure 18-19**. This window allows the user to choose the resolution or number of bars for the PDF, CDF and Histogram displays for the selected stochastic process in the Graphical Display area.

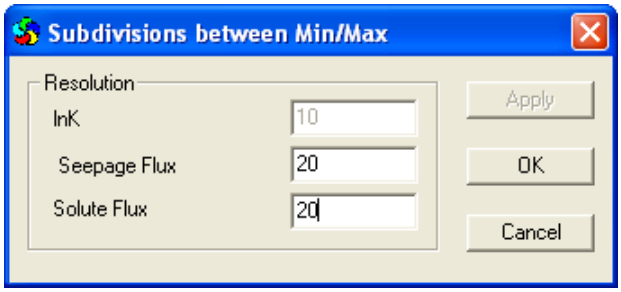


Figure 18-19 Probability resolution for PDF, CDF and Histograms for various parameters

In the middle of the Display Options area, the user can choose to display the Process, the PDF, the CDF or the Histogram. Only one of these displays can be selected at a time. **Figure 18-18** shows the process for concentration. PDF, CDF and Histogram for concentration are shown in **Figure 18-20**. Also note that when PDF, CDF or Hstgram is selected, the **Process Curve Choice** and **Variation w/ Realization** areas become inactive.

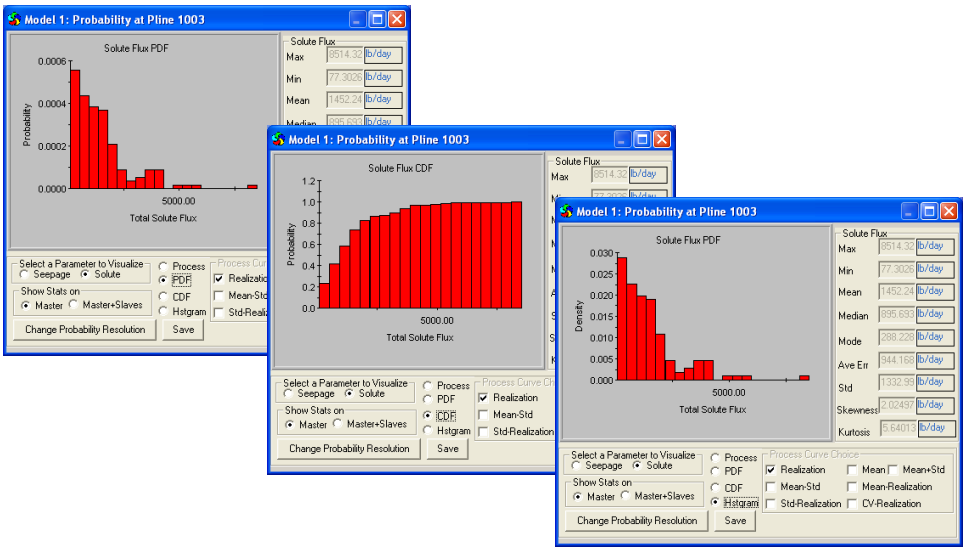


Figure 18-20 PDF, CDF and Histogram for concentration process

The Save button allows the user to save the data generated in all the realizations for Seepage and Solute. The data is saved in a *.dat file format. The format is explained in **Section 23.6.2**.

Process Curve Choice area allows the user to plot the stochastic process. It also allows to plot the dynamic display of mean value of a parameter at the end of last realization, along with the bounds around the mean at plus and minus of one standard deviation. In the Graphical Display area, these choices can be plotted simultaneously.

The **Mean-Realization**, **Std-Realization** and **CV-Realization** represent variation of mean, standard deviation and coefficient of variation, respectively, at end of each realization. All curves in Process Curve Choice area can be plotted simultaneously except **CV-Realization.** Also, once CV-Realization box is checked, no other curve can be seen except that of the **CV-Realization** in the **Graphical Display** area.

The plots of **Mean-Realization**, **Std-Realization** and **CV-Realization** are useful in determining whether or not enough of realizations have been generated to have statistically more meaningful modeling results. This aspect is discussed further in **Section 18.7**.

Figure 18-21 shows different plots for Solute flux across a polyline. Please note the selection of check boxes in **Display Options** area and relate them to the curves on the plots.

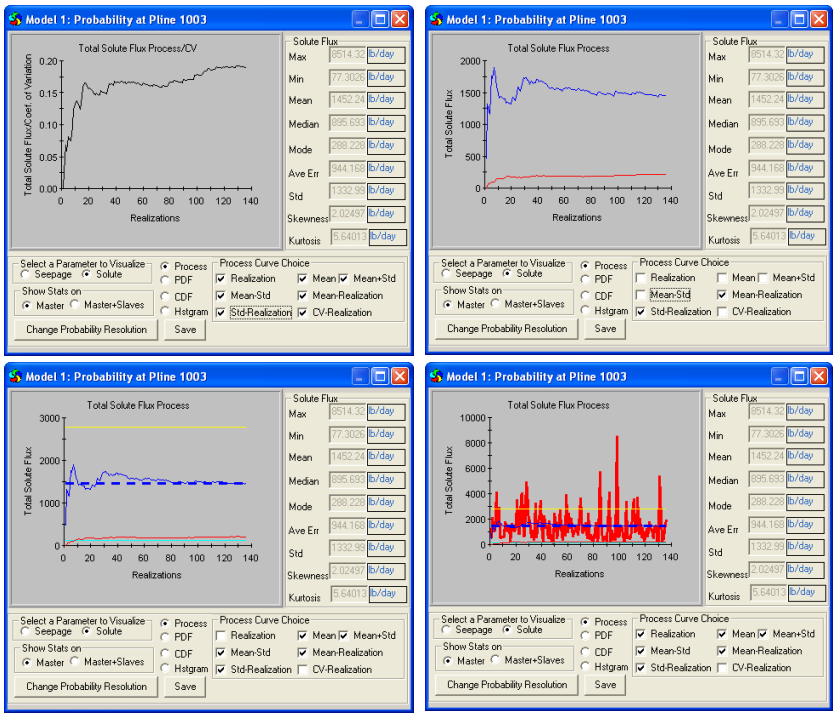


Figure 18-21 Various displays of solute flux stochastic process

> **i** Note that the probability results displayed in these windows include all realizations up to the present time. If only two simulations have been run, then the probability results are based on those two simulations only.

18.6. Stochastic Modeling Results

This section briefly describes the results obtained for each type of stochastic simulation.

18.6.1. Single Realization Simulation

In single realization mode, there are no extra interfaces to display model results. All solution data is presented in the same fashion as would be expected when operating in mean mode. It is important to note that a single realization is one out of an infinite number of realizations, and hence there is no measure of the likelihood of the given solution values being encountered.

18.6.2. Monte Carlo Simulation

The results for Monte Carlo simulations are continuously updated. The Working Area will show the current realization. The past realization windows will show the specified number of past realizations at the specified time state. All other windows display information based on all of the realizations generated and solved up to the present realization.

18.7. Parallel Computing

Parallel computing feature of *IGW Version 5.0P* allows the user to take advantage of maximum computing resources available in a network to perform a large number of realizations for Monte Carlo simulations. Within the network, one of the computers can act as the **Master** machine and others as **Slaves**. The Master can assign separate tasks to the slave machines within the network.

18.7.1. Prerequisites

To enable a net work for parallel computing in *IGW Version 5.0P*, open source Message Passing Interface (MPICH2) should be run on machines in the system which the user wants to employ for parallel computing. MPICH2 is an open source file and can be downloaded at:
http://phase.hpcc.jp/mirrors/mpi/mpich2/

Each machine should be given one of the names that appears in **Parallel Hosts and Tasks** window (**Figure 18-25**). Currently, *IGW Version 5.0P* code is customized to these 12 name (the names are based on the network where the software is being developed). To setup the network, follow these steps:

(1) Install MPICH2 on machines in the network that you want to use for parallel computing.

(2) Check connections by running `config.exe` of `MPICH2`.

(3) Map a network drive with the name "`S`" (network drive name other than 'S' is not supported by *IGW Version 5.0P* code)

(4) Put `IGW.exe`, `VBMPIEXE.exe` and related `DLL` files to `S` drive.

(5) Save the following script, shown in , as a text file at a desired location on all machines. The file should be named `IGWMPI3D.txt`.

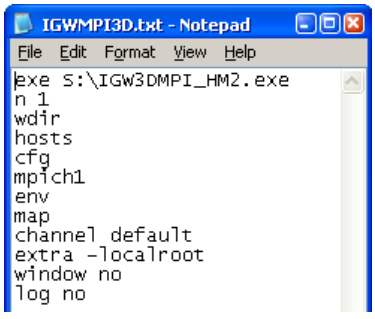


Figure 18-22 IGWMPI3D text file

With this setup, any machine on the network can be used as Master/Slave.

18.7.2. Starting a Parallel Computing Session

In order to start *IGW Version 5.0P* for parallel computing, the software has to start through WMPIEXEC program. This program becomes available after MPICH2 is installed. To being a parallel computing session, the user should follow these steps:

(1) Run WMPIEXEC.exe file on the Master machine. The file can be accessed from the program menu or a desk top shortcut. This will bring up the '**MPIEXEC wrapper**' window as shown in .

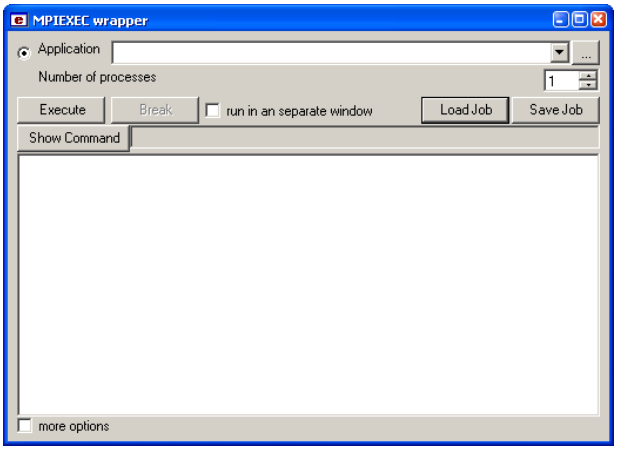


Figure 18-23 MPIEXEC wrapper window

(2) Click 'Load Job'.

(3) Browse to the file `IGWMPI3D.txt` saved on your machine, and click 'OK'.

(4) Click 'Execute' button in the **MPIEXEC wrapper** window.

This will open *IGW Version 5.0P* main window. The user can either create a model for Monte Carlo Simulations in the window as explained in previous sections, or just open a saved model for simulations. After discretizing and setting the model (including display and TPS options), the user can enter the Monte Carlo simulation mode as explained in **Section 18.3**.

(5) Before hitting the 'Run Model Forward' button, click  button. The **Solver** window pops up (**Figure 18-24**).

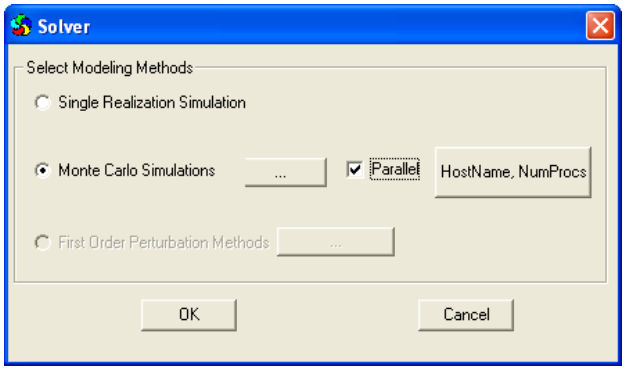
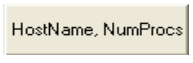


Figure 18-24 Selecting Parallel Computing option in Solver window

(6) Check box on the left of 'Parallel' and click 

button. Parallel Hosts and Tasks window pops up as shown in **Figure 18-25.**

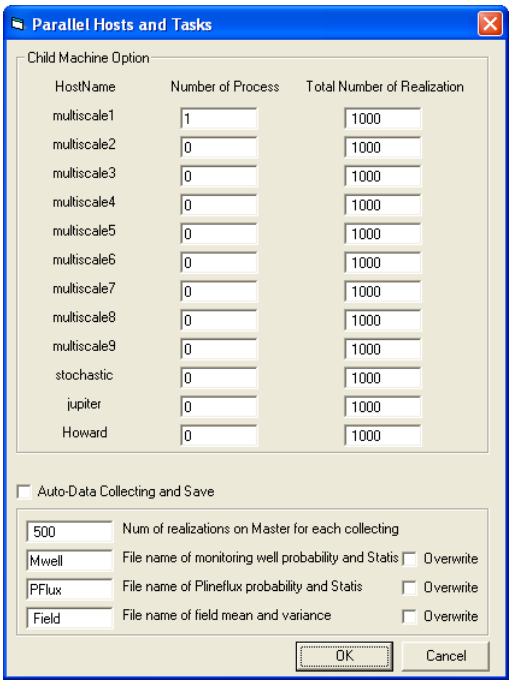


Figure 18-25 Parallel Hosts and Tasks window

Parallel Hosts and Tasks window is divided into two area. The 'Child Machine Option' area has three columns. The first column contains the names of all the machines available on the network. The second column contains the fields in which user can allocate/assign number of processes each machine will perform. The last column is number of realizations for each process a machine will perform. This number will instruct the machine to stop after that many number of realizations for each process. However, if the user desires to let the slave machines continue realizations as long as the user wants, then the '**Stop at**' box should be unchecked in **Monte Carlo Simulation** window (**Figure 18-2**) prior to accessing the **Parallel Hosts and Tasks** window. Although a number will still appear in the 'Total Number of Realizations' field, but it will not be effective.

(7) Before selecting the number of processes to be assigned to each slave machine in the network, get all desired slave machines in the network ready to receive jobs from the master machine. To get them ready, follow these steps:

- Go to each machine and open the Command Prompt.
- Go to root directory `C:\>` and type the following:
  `C: \>aaa`
  then press Enter.
- `C:\> echo off` is displayed in the Command Prompt window.
- Leave the window open. The machine is now ready to accept jobs from the Master machine.

(8) Come back to **Parallel Hosts and Tasks** window.

(9) Enter the number of processes/jobs to be assigned to each Slave machine in the network.

By default, one job is already assigned to the Master, i.e., the main model that is running in the machine. More number of child jobs can be assigned to master. At least one job is to be assigned to every slave machine that the user wants to use for parallel computing. A zero in second column against any machine would mean that that machine is not participating in parallel computing. This is sometimes desirable because not all machines on the network can always be available/spare.

(10) Select the **Auto-Data Collecting and Save** options for monitoring wells, polyline fluxes and fields. Checking the Auto-Data Collecting and Save box is always a good idea when

the user desires to run simulations over a longer time. The file locations for saving the data are setup in the S drive during the network setup stage.

(11) Click 'OK' button in **Parallel Hosts and Tasks** window.

(12) Click 'OK' button in Solver window.

(13) Click the 'Run Model Forward' button.

All machines selected in the network will start performing allocated tasks. The user will observe *IGW Version 5.0P* windows open in each slave machine. The model simulations can be observed in the Working Area for each job in each machine.

18.7.3. Stopping a Parallel Computing Session

The user can stop the parallel computing session by clicking the Pause/Stop button in the Master machine. This will stop simulations in all slave machines. However, before the simulations stop, a messages appears on the screen prompting the user to save simulations results.

If results are not saved before stopping the simulations, all the unsaved results from slaves machines are lost.

Simulations in any slave machine can be stopped by clicking the Pause/Stop button within the job window of that machine.

18.7.4. Observing Simulation Results

The user can observe simulation results for parallel computing for spatial fields, monitoring wells, and polylines exactly the way it is done in normal simulations as explained in **Section 18.5.2**. The only exception is that the user can now have the option to either select **Master** or **Master+Slaves** option in the '**Show Stats on**' area (please see 'C – Display Options area' in **Figure 18-13** and **Figure 18-18**).

Some parallel computing results for Monte Carlo simulations are illustrated below. These illustrations also highlight the advantages of parallel computing.

Figure 18-26 shows the Conc process and the variation of its Mean with realizations. One can see that after 300 realizations, the Mean (blue line) is still fluctuating.

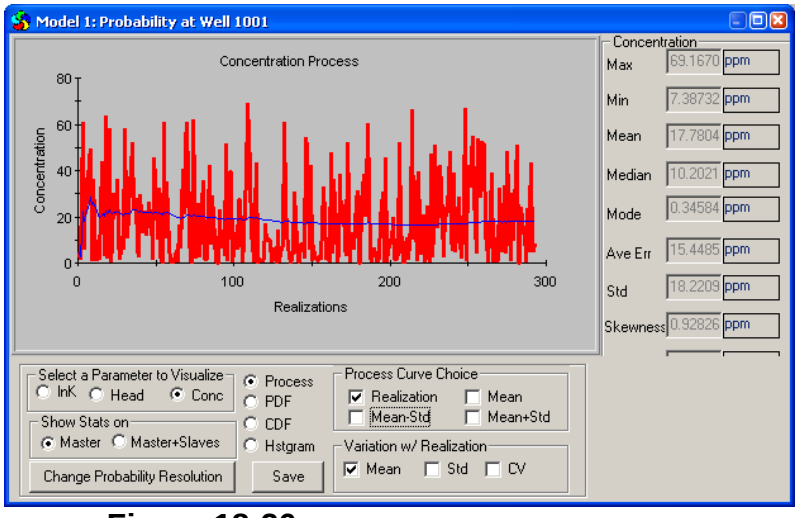


Figure 18-26 Master – Process and its Mean

However, when the display is switched in 'Show Stats on' area from **Master** to **Master+Slave**, one can see in **Figure 18-27** that the number of realizations close to 2700 and the mean is far more stable than in just 300 realizations.

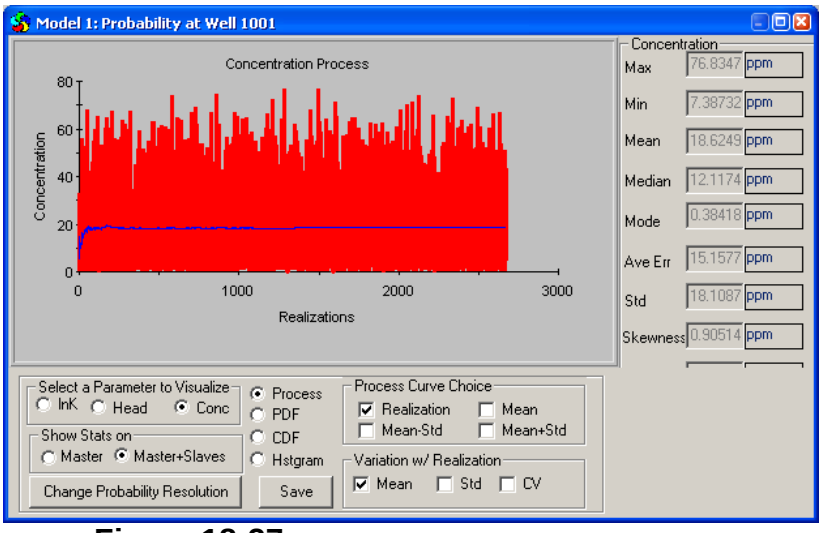


Figure 18-27 Master+Slave Process and its Mean

For more meaningful stochastic results, it is always a good idea to have a larger number of realizations. The decreasing fluctuation in Mean as the number of realizations increase is a good indication of whether or not enough of realizations have been generated. Depending on the number of machines and processors available in a network, the speed of generation realizations could be order of magnitude faster than would be possible on a single machine. The example above just illustrates the point.

The user can take advantage of those machines in the system which are faster with more number of processor by allocating them more jobs. This way, the computing power in the network can be optimized.

When the number of realizations is large, the graphical representation of Master+Slave results can become too cluttered and it may become difficult to observe whether or not the fluctuation in Mean (or other parameters) has stabilized. The user can change the x-axis display in the 'Graphical Display area' (**Sections 18.5.2.2** and **18.5.2.3**) to log scale using the '2D Chart Control Options' (**Section 14.3**). Comparison between **Figure 18-27** and **Figure 18-28** illustrates this point.

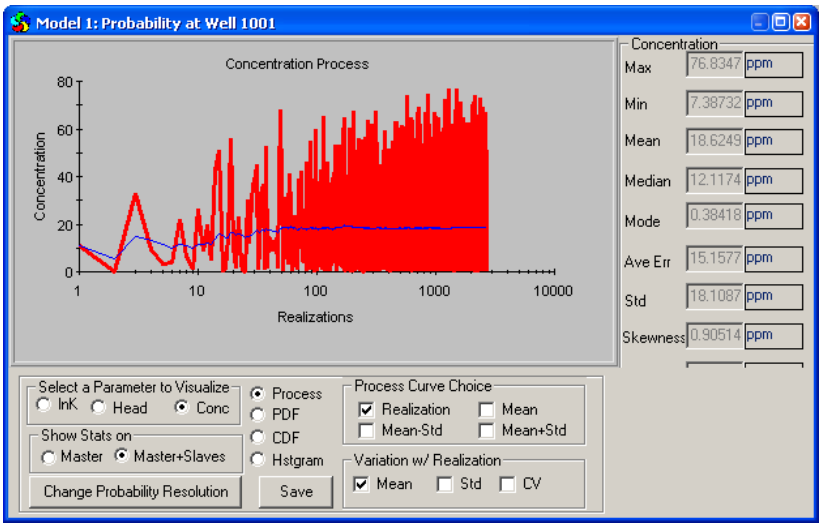


Figure 18-28 Master+Slave Process and its Mean with number of realizations on log scale

With larger number of realization, the distribution patterns of various stochastic processes become more discernable. **Figure 18-29** and **Figure 18-30** show the PDFs for observed heads at a monitoring well location. One can see that using only one (Master) machine cannot create a very clear distribution pattern as compared to parallel computing operations.

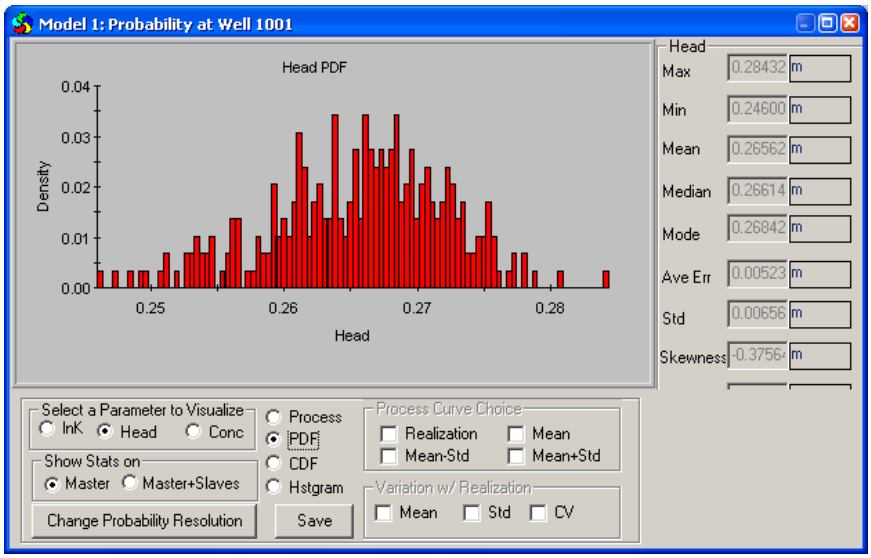


Figure 18-29 Head PDF with Master

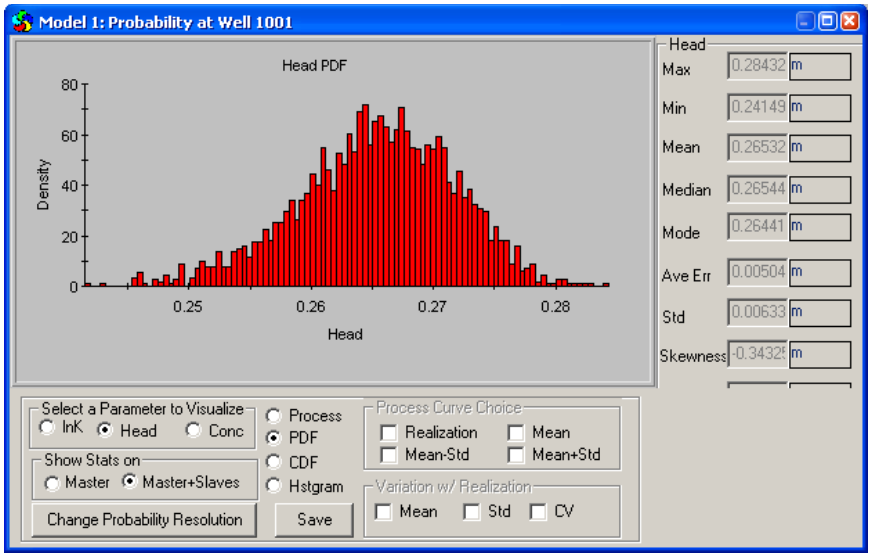


Figure 18-30 Head PDF with Master+Slaves

Finally, show the fluctuations in Coefficient of Variation for Master machine and Master+Slave. All these examples highlight the point that in order to have more meaningful stochastic simulations, we always need a larger number of realizations. For relatively more complicated models, it might take a very long time to complete the required number of realizations. Parallel computing features can improve the speed of generating realizations by orders of magnitude, depending upon the size and combined computing power of a network.

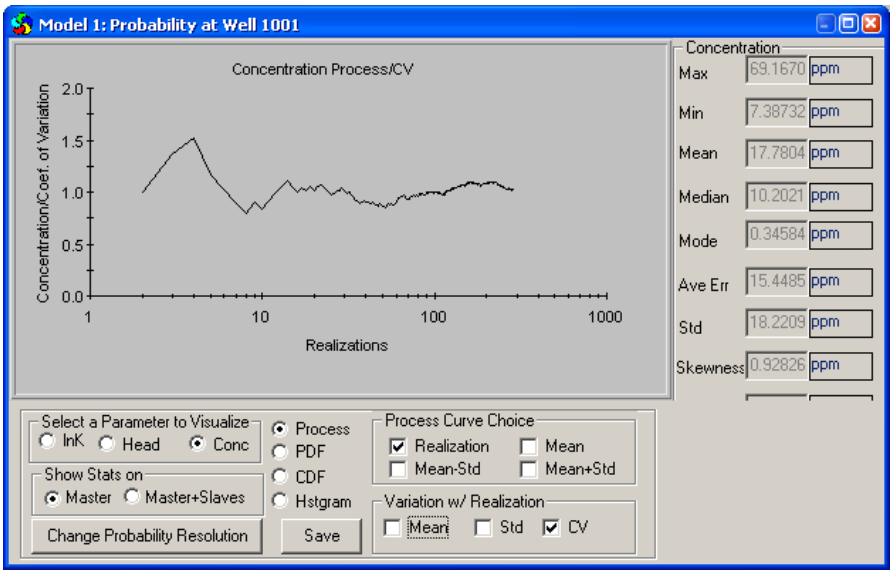


Figure 18-31 CV with Master

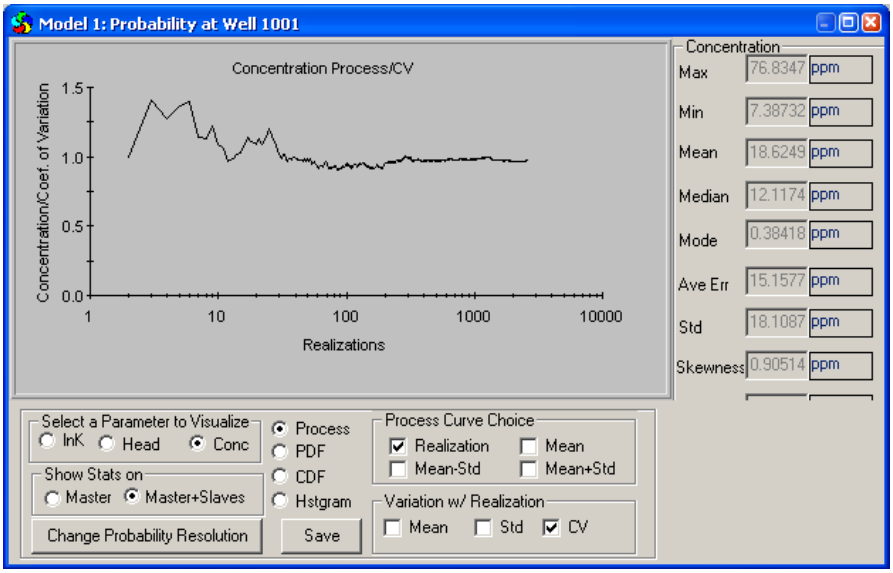


Figure 18-32 CV with Master+Slaves

Chapter 19 DISPLAY OPTIONS

IGW Version 5.0P provides the user with a variety of options concerning the visualization of modeling parameters and results in the Working Area. This chapter details those options.

19.1. Display Options Interface

The display options[26] are controlled through the '**Display Options for Model 1**' window (**Figure 19-1**) that can be accessed by clicking the '**Display Options**' button on the Button Palette (see **Section 3.4**). The window can also be accessed by clicking the 'Option…' button in the RHP of 'Main Model' in the AE, or from by selecting '**Display Options**' from the **Display** menu, or by selecting 'Display Options' in the pop up menu after right-clicking in the Working Area.

The window is divided in five areas (Reference Maps, Conceptual Features and Texts, Simulation Inputs and Results, Monte Carlo Simulations and Display Sequence) and some general options. All parameters in the window are discussed in the following sections.

When the desired options have been selected, the user selects 'OK' to set the options in the software or 'Cancel' to close the window and discard any changes.

> **i** This window sets the visualization attributes for the main model. Visualization attributes submodels and profile models (cross sections) are set in separate windows. Refer to Error! eference source not found. and **Chapter 16**, respectively, for further information.

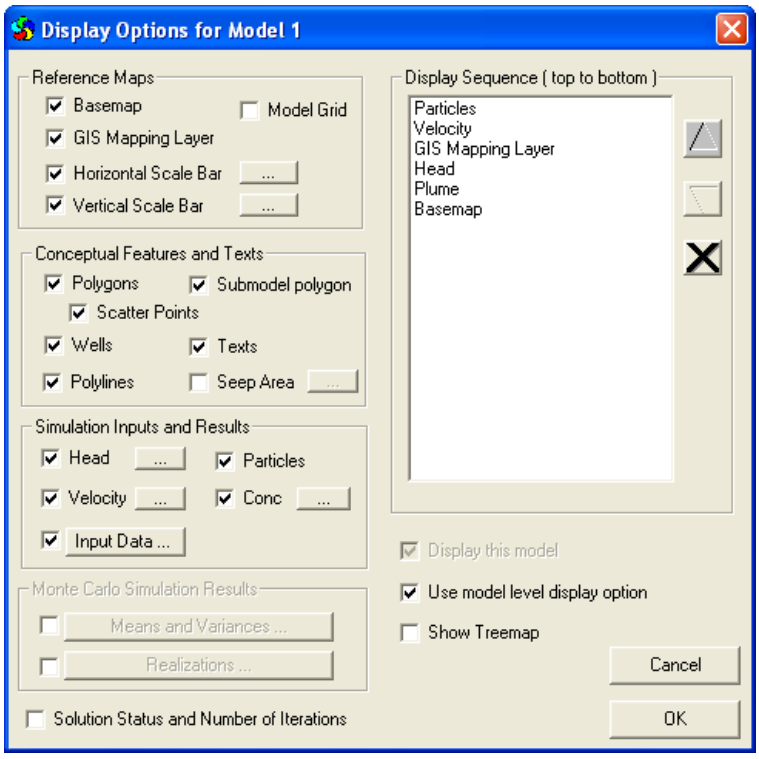


Figure 19-1 Drawing Options for the Main Model

[26] Refer to **Section 7.1** of the ***IGW Version 5.0P Tutorials*** document for examples of changing the options in this window.

19.2. Reference Maps Area

In this area, the user may choose to activate or deactivate visualization for the Basemap, GIS mapping layer, Model Grid, Horizontal Scale Bar and/or Vertical Scale Bar, by checking the respective boxes. By default, Model Grid is unchecked and all other features are checked.

There are option buttons ... for Vertical and Horizontal scales bars in this area. Clicking on these buttons will open the respective windows for Horizontal Scale and Vertical Scale as shown in **Figure 19-2**.

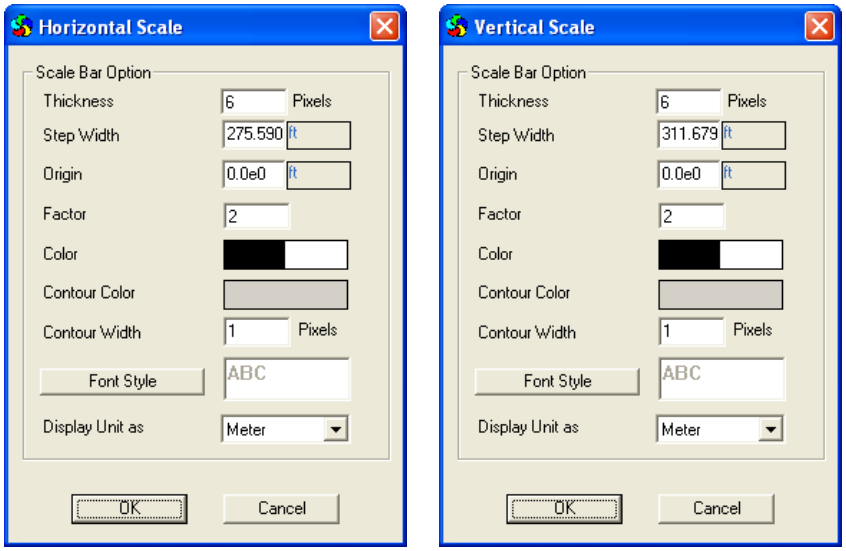


Figure 19-2 Horizontal and Vertical Scale windows

Both of these windows work exactly the same way. The user may adjust the **Thickness** (the default is 6 – as pixels), **Step Width** (the number is chosen by the software as the smallest integer that will yield at least 10 whole steps in the display), **Origin** (the default value is 0.0e0 – the default unit is feet [ft]), **Factor** (default is 2; the factor indicates how often the distance should be displayed on the bar 2 instructs the software to display the distance at every other step. The value associated with the origin is never displayed), **Color** (default is black and white – choosing a different color will only change the black part on the scale bar), **Contour Color** (default is grey) and **Contour Width** (the default is 1) by entering numbers in the appropriate field. These parameters are visually explained in **Figure 19-3**.

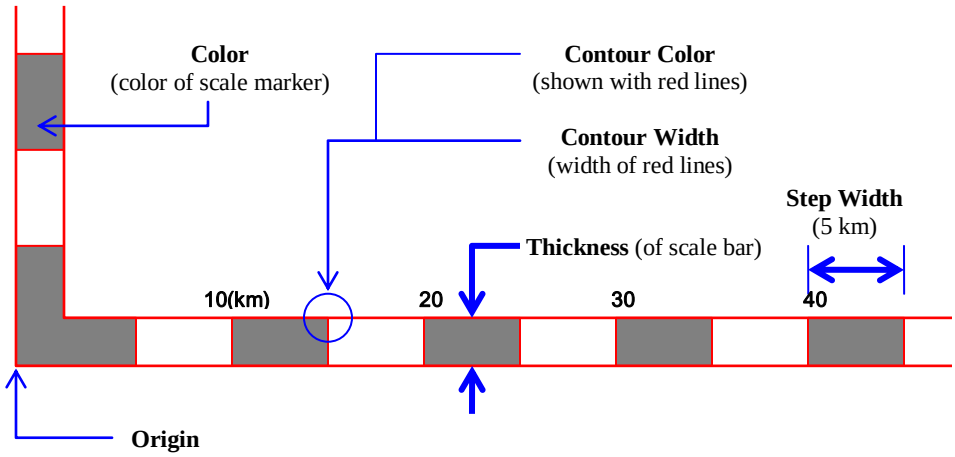


Figure 19-3 Scale parameters with some illustrative values/attributes

Clicking on '**Font Style'** this button opens Font window (**Figure 19-4**). This window pertains to the font of the labels on the scale bar, giving user the options to select font parameters.

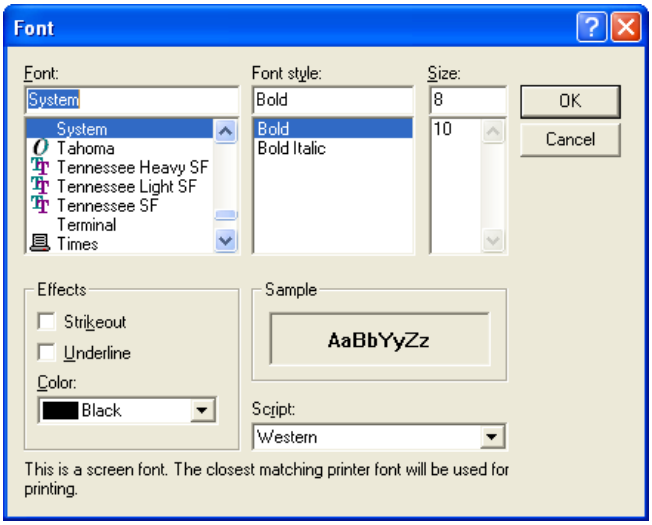


Figure 19-4 Font Window

Display Units are in meters by default. The user can also choose from feet, miles and kilometers. The unit of scale is displayed with the first label on the horizontal scale bar.

Clicking the 'OK' button sets the changes and closes the window. Clicking the 'Cancel' button discards any changes and closes the window.

19.3. Conceptual Features and Texts Area

In this area, the user may choose to activate or deactivate visualization of conceptual features, and text.

19.3.1. Displaying Conceptual Features in the Model Area

Conceptual features in the Model Area are presented by **Polygons**, **Scatter Points**, **Wells**, **Polylines**, **Submodel** polygons and **Seep Area** polygons. User can check or uncheck the box to display or hide any feature. Hiding any conceptual feature in the display options will not affect the model results. The user can also add annotations in the working area and choose to display or hide the annotated text using the Texts check box.

All conceptual features are displayed by default except **Seep Area**. Clicking the options button ... next to **Seep Area**, opens the '**Pattern Display Option**' window as shown in **Figure 19-5**.

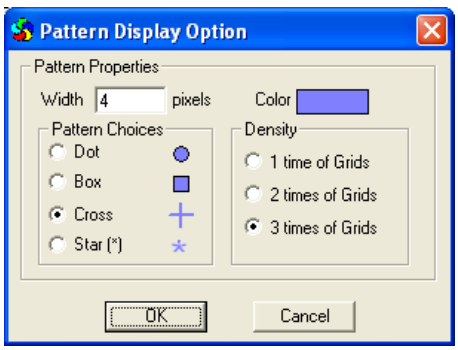


Figure 19-5 Pattern Display Options

i	The seep area visualization can be used to delineate a seepage area (such as a wetland) as a function of time.

In the 'Pattern Properties' area, the user may set the width in the 'Width' field (the default is 4 – as pixels), choose the pattern type in the 'Pattern Choices' area (the default is 'Cross'), select the density of the pattern display in the 'Density' area (the default is '3 times of Grids'), and/or select the pattern color by clicking on the sample color swatch to open the 'Color' window (in which the user subsequently selects the desired color).

19.3.2. Implementing Text into the Working Area

Text fields may be placed in the Working Area through use of the 'Add Text' button. User can click on the Add Text button and then click once anywhere inside the Working Area to create a *text field* at that point. User can click at more than one locations to create as many text fields in the model. After creating a *text field*, nothing shows up in the Working Area, however, all the text fields created by mouse clicks apprear in the LHP of AE.

The user can add/edit text by selecting a *text field* in the LHP (see **Section 4.1.1**). Doing this brings up the 'Text' RHP (see **Section 4.1.2**). A sample of the RHP for 'Text 1001' is shown in **Figure 19-6**.

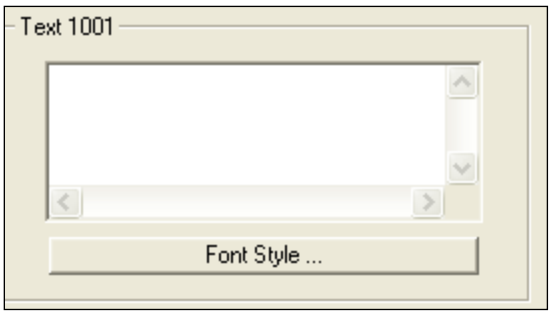


Figure 19-6 RHP for Assigning Text in Working Area

The user may enter/edit the desired text in the text box showing in the RHP. Font style may be adjusted by clicking the '**Font Style…**' button to open the '**Font'** window. This window is similar to the one in **Figure 19-4**. After adding the text, click anywhere outside the text box and the text will appear in the Working Area.

Once a text is visible in the Working Area, it may be selected by clicking the '**Select Text**' button, and then clicking the cursor on the text.

Using the **Text** check/uncheck box in the **Conceptual Features and Text** area, user can choose to display or hide the text in the working area. By default, this box is checked.

19.4. Simulation Input and Results Area

In this area, the user may choose to activate or deactivate visualization for 'Head', 'Velocity', 'Concentration', 'Particles', and 'Input Data'. All five are active by default. Clicking the '…' button next to 'Head', 'Velocity', or 'Concentration' opens up a separate window ('Display Option – Head', 'Velocity Display Option', and 'Display Option – Concentration', respectively) for further refinement of the display of these features. These interfaces are discussed in the following subsections.

19.4.1. Head Display Options

The 'Draw Option – Head' window is shown in **Figure 19-7**.

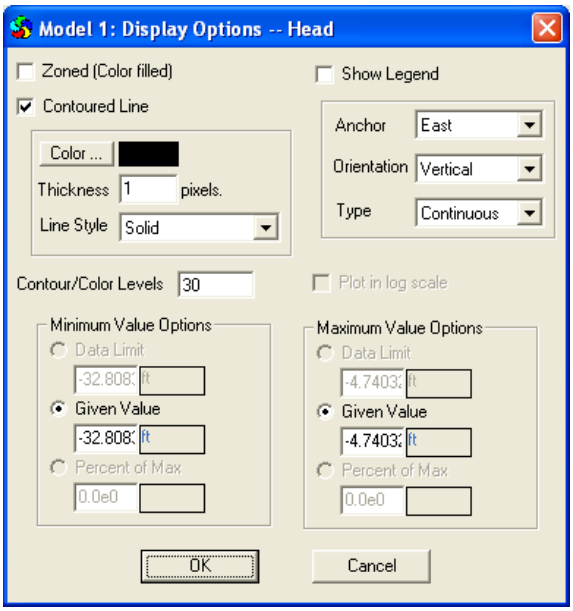


Figure 19-7 Display Options for Hydraulic Head

At the top of the window, the user may choose to activate or deactivate visualization for contour color fill ('Zoned (Color filled)'), the contour lines ('Contoured Line'), and the legend ('Show Legend'). Note that 'Plot in log scale' is not an active option at this time.

> **i** When any parameter is visualized as **Zoned** (color-filled), the contour associated with the lowest value range is set to be transparent. This prevents the entire basemap from being blocked out when these parameters are visualized.

The user may further refine the contour line display by clicking the 'Color…' button, or clicking on the color patch (to open the 'Color' window and subsequently select the desired color) and by setting the line thickness (in pixels) in the 'Thickness' field.

The number of contour levels can be specified in the 'Contour Levels' field.

In the '**Minimum Value Options**' and '**Maximum Value Options**' areas the user may specify the extreme values through which the data should be visualized. Selecting '**Data Limit**' (default setting) allows the software to automatically determine the value (which is displayed in the associated field). Selecting '**Given Value**' allows the user to specify the value in the appropriate field. Selecting '**Percent of Max**' allows the user to specify (in the appropriate field) a percentage (either as a decimal or percentage, depending on the unit selected) to determine the value (based on the maximum value determined by the software).

> **!** The user must UNCHECK the '**Use Model Level Display Option**' if choosing '**Given Value'** or '**Percent of Max'** to implement these values in the model display. If this box is left CHECKED (which is the default setting), the display will still use **Data Limit** option. Please see **Section 19.4.7**.

19.4.2. Velocity Display Options

The 'Velocity Draw Option' window is shown in **Figure 19-8**.

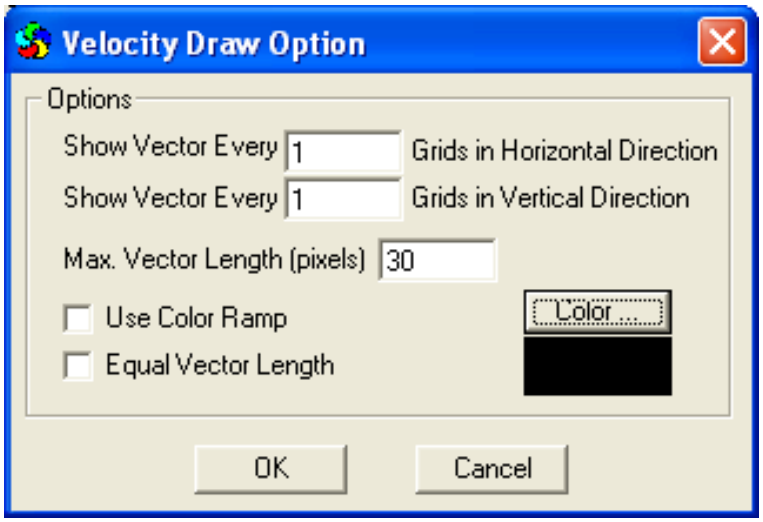


Figure 19-8 Display Options for Velocity Vectors

Near the top of the window, the user may change the spacing of the displayed velocity vectors (based on the grid) by adjusting the values for the horizontal and vertical directions. The default values for each are 1.

The user may specify the length (in pixels) of the vector associated with the highest velocity in the model by entering a value in the 'Max. Vector Length (pixels)' field. The default value is 30.

Checking the 'Use Color Ramp' box instructs the software to display the velocity vectors as a continuum of colors based upon the absolute value of the vector.

Checking the 'Equal Vector Length' instructs the software to draw each velocity vector at the 'Max. Vector Length (pixels)' value. This option is best used in conjunction with the color ramp (above).

The user may adjust the color of the velocity vectors by clicking the 'Color…' button to open the 'Color' window and subsequently choosing the desired color. The selected color is not applicable when 'Use Color Ramp' is checked.

Clicking the 'OK' button closes the window and sets the changes in the software. Clicking the 'Cancel' button closes the window and discards any changes.

19.4.3. Particle Display Options

Particle display options can be set in the AE (See **Section 10.2.1** and **Section 10.2.2**). Checking/un-checking the Particle box will display or hide the particles in the Working Area.

19.4.4. Concentration Display Options

This window is of the same format as the 'Draw Option – Head' window (see **Figure 19-7**).

19.4.5. Input Data Display Options

This button provides access to the 'Model Input and Data' window (**Figure 19-9**) which contains a list of other features / data that may be displayed in the Working Area. Each entry also has an associated options button ... used to refine its display in the Working Area.

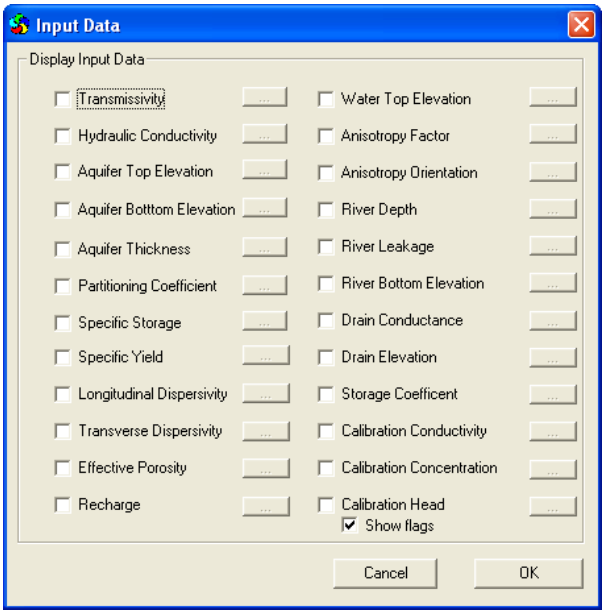


Figure 19-9 Input Data for Visualization

The user can check all the desired parameters from this window and set their display options using the ... button.

When '**Input Data …**' box is checked, in the **Display Options for Model 1** window, all the attributes checked in the '**Input Data**' window will be display in the Working Area. By default the '**Input Data …**' button is checked, but all buttons in **Input Data** window are unchecked. Consequently, none of the attributes from this window are displayed by default in the Working Area.

19.4.6. Solution Status and Number of Iterations

The **Solution Status and Number of Iterations** box at the bottom right corner of the window unchecked by default. Checking this box instructs the software to display more detailed message windows concerning the solution status as the software is solving the model.

19.4.7. Use Model Level Display

'**Use Model Level Display**' option is checked by default. When this is checked, the contour intervals and color ramps are calculated/assigned by the software based on maximum and minimum values of the attributes. If user decides to use a different range of values (than the maximum and minimum within the model solution/data) this option must be unchecked.

19.4.8. Show Treemap

Show Treemap is unchecked by default. This feature brings up the hierarchical structure of all the submodels in the main model. This option is elaborated in **Section 15.3**.

19.5. Monte Carlo Simulation Results Area

These settings are reserved for stochastic modeling and are discussed in **Section 18.3.1**. By default, these are unchecked and grayed out.

19.6. Display Sequence (Top to Bottom) Area

As soon as a feature is selected for display (e.g. Head, Basemap etc.), it appears at the bottom of the list in the '*Display Sequence*' field. This area shows the order (from top to bottom) in which the displayed features will be shown in the Working Area. Items may be selected by clicking on them and then moved up (up arrow button), or down (down arrow button) in priority, or deleted altogether (X button). Note that certain features (such as basemaps) may be displayed over the entire extent of the Working Area, and may block out other features (such as head contours). By adjusting the top to bottom sequence the user can see more in the model area.

19.7. Refreshing the Display

The '**Refresh Screen**' button may be used at any time to refresh the features in the Working Area. This button will not draw features that have not yet been discretized into the model.

19.8. Zooming In/Out

The 'Zoom in' button may be used to enlarge the Working Area within the model screen. The 'Zoom out' button may be used to shrink the Working Area within the model screen**.**

Chapter 20 SCATTER POINT DATA PROCESSING

Scatter points are discrete points that may be associated with zones to achieve greater resolution when data (such as bore hole logs) is available for one or more points.

Within **IGW**, the processing of scatter point data consists of three primary steps, **Exploratory Data Analysis**, **Outlier Analysis**, and **Interpolation/Simulation**.

The view of the left-hand pane, **LHP** is where the user selects the desired statistical interpretation for the defined scatter points. It can be accessed by right-clicking on any zone in the LHP and selecting **Switch List** from the drop-down menu appears (**Figure 20-1**).

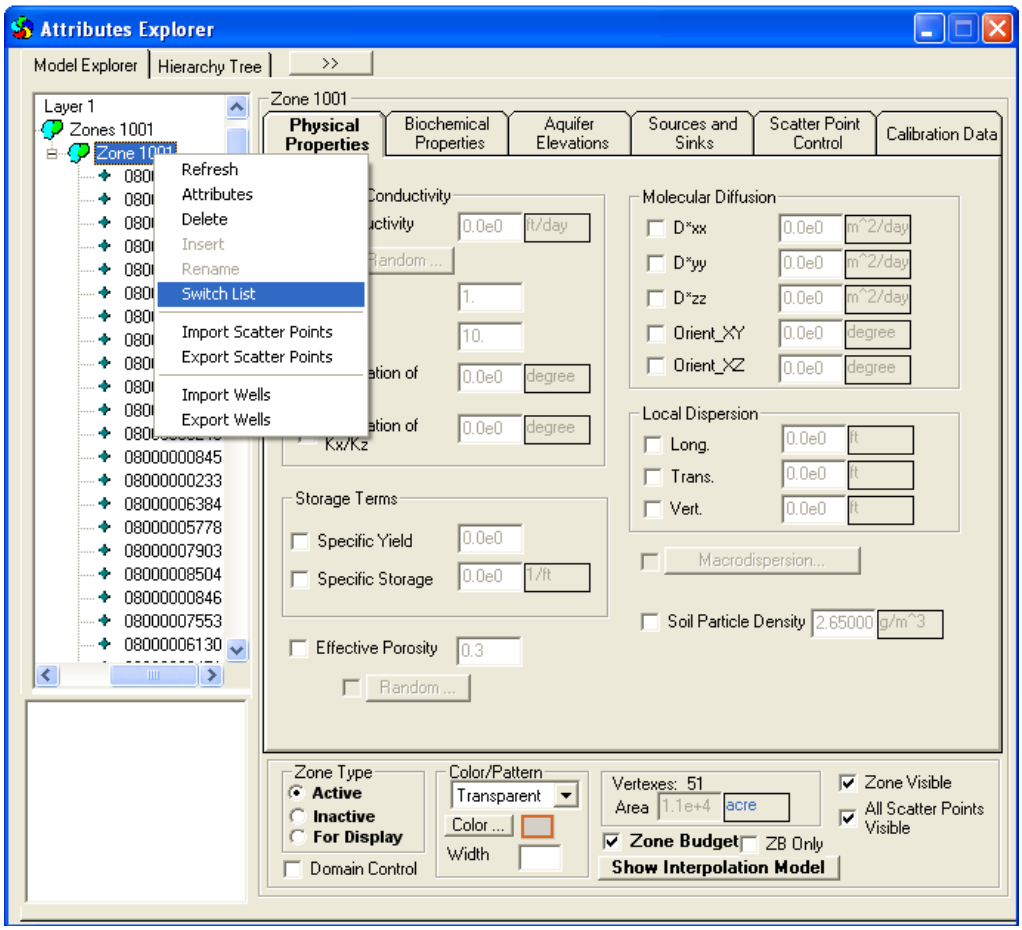


Figure 20-1 Scatter Points and Attributes Explorer

The **LHP** view differs in that the individual scatter points under a zone are replaced by entries for each parameter being specified by the scatter points. These entries also indicate the number of scatter points that have this particular parameter specified. Each entry has its own right-hand pane (**RHP**) in which the user may specify the conditions for statistical analysis of the group of scatter points.

20.1. Exploratory Data Analysis

Prior to interpolating scatter point data, it is advisable to examine the distribution of the data. This is accomplished using **Exploratory Data Analysis**. It is only possible to perform exploratory data analysis on scatter point data at this time. It is not possible to analyze attributes associated with polylines or polygons. Data analysis is accomplished by highlighting a particular scatter point attribute in the **Attributes Explorer** and selecting the **Exploratory Analysis** button (**Figure 20-2**).

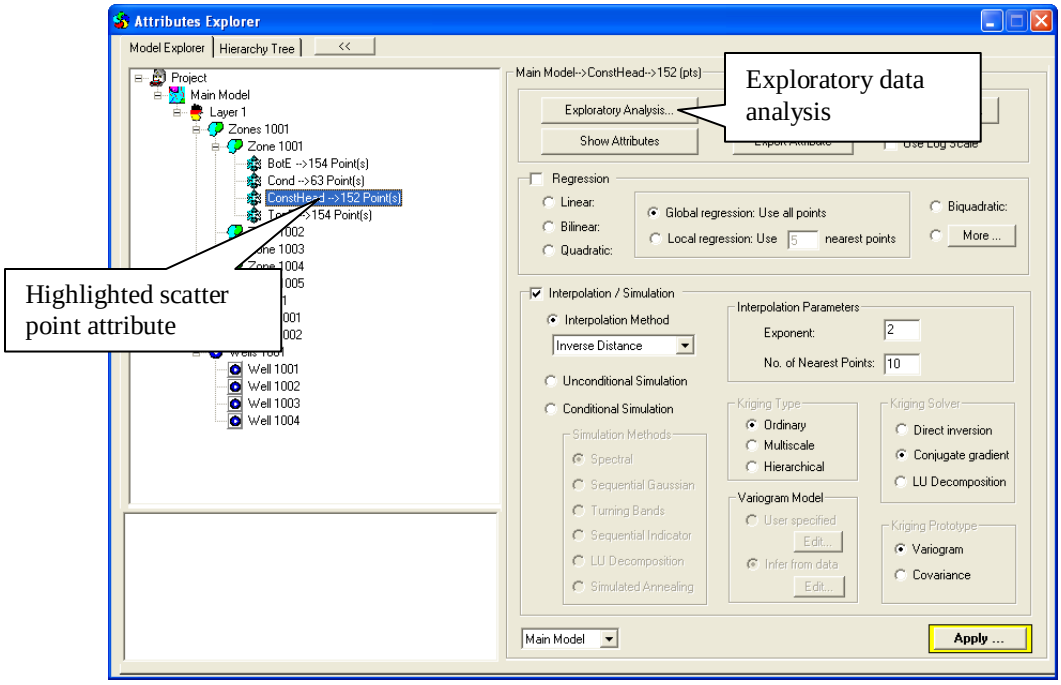


Figure 20-2 Exploratory Data Analysis

The **Exploratory Data Analysis** windows are shown from **Figure 20-3** through **Figure 20-6**. It is possible to display the scatter point data as a histogram (**Figure 20-3**), a probability density function (**PDF**) plot (**Figure 20-4**), a cumulative density function (**CDF**) plot (**Figure 20-5**), or an **h-scatterplot** (**Figure 20-6**). Different types of data distribution plots are displayed by selecting one of the tabs near the top of the window (**Figure 20-3**). The distribution statistics for the data set are displayed along the right-hand side of each window. Default length and time units for all plots are meters and days. It is currently not possible to change either unit for these plots.

In the bottom right-hand corner of each window are three graph parameter options that may be entered by the user. The first option, **Number of Intervals**, applies only to the **Histogram**, **PDF**, and **CDF** plots.

R

It is recommended to increase the number of intervals to more accurately display the distribution of the data set, as having only a few large intervals may give an erroneous depiction of the data distribution.

The last two options, **Scatterplot Lag h**, and **Scatterplot Tolerance,** apply only to the **h-Scatterplot**. Increasing the values for either parameter may present a better depiction of the data distribution.

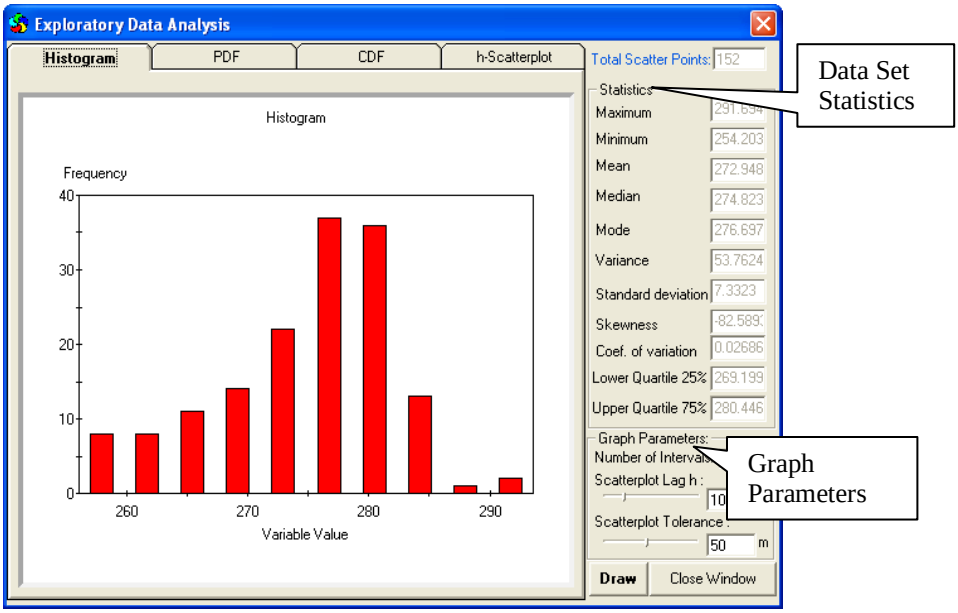


Figure 20-3 Histogram Analysis

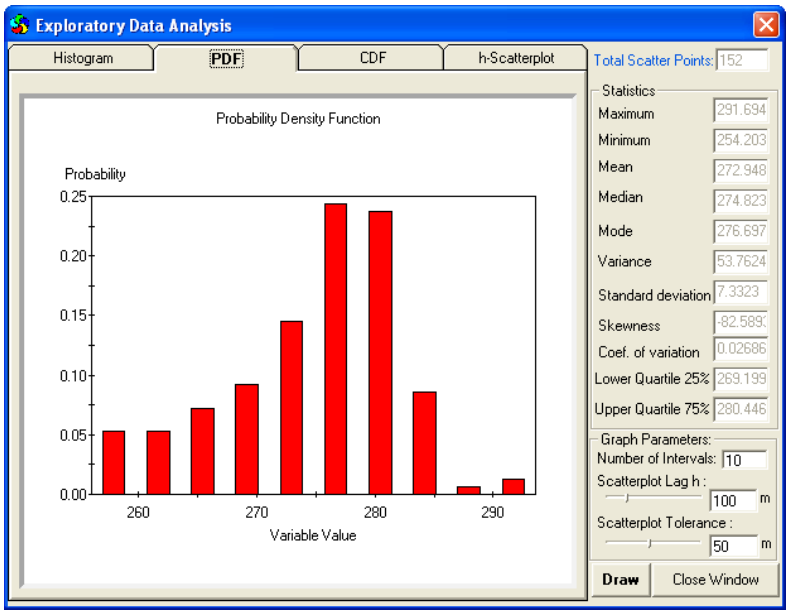


Figure 20-4 Probability Density Function

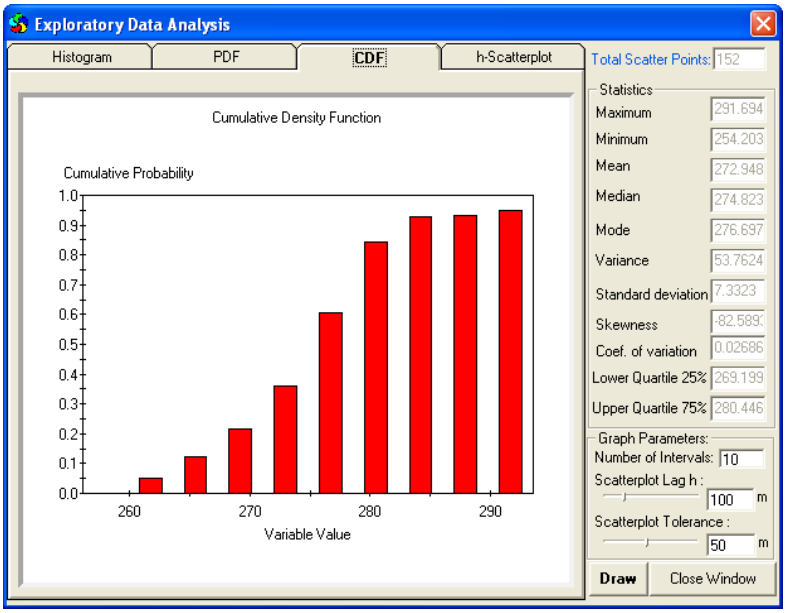


Figure 20-5 Cumulative Density Function

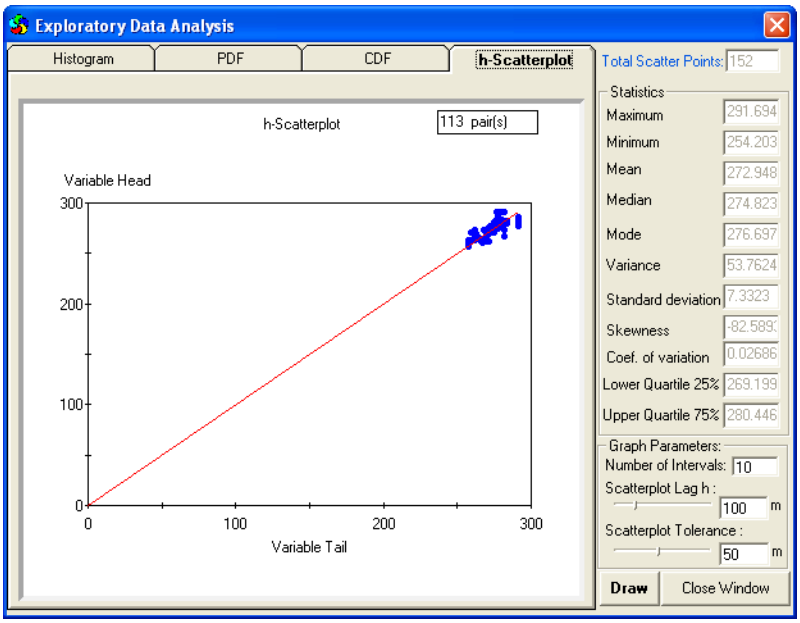


Figure 20-6 Scatterplot

20.2. Outlier Analysis

Following the examination of the data distribution in the **Exploratory Data Analysis**, it is generally necessary to perform an outlier analysis of the data and remove any possible outliers.

The first step is to check the **Regression** analysis check box (**Figure 20-7**). This is necessary since the outlier analysis fits a regression equation of the users choosing to the data. The user also selects the type of regression equation, e.g. **Linear**, **Bilinear**, **Quadratic**, **Biquadratic**, or a higher order (**More**), and whether to perform **Global** or **Local** regression. These options depend entirely on the data set.

R It is recommended that the user perform the outlier analysis using different regression options, to obtain the "best" regression fit to the data.

After selecting the **Regression** options, the user next selects the **Outlier Analysis** button. When processing hydraulic conductivity data, it is necessary to check the **Use Log Scale** option. Selecting this option forces **IGW** to process the logarithm of the estimated hydraulic conductivity values.

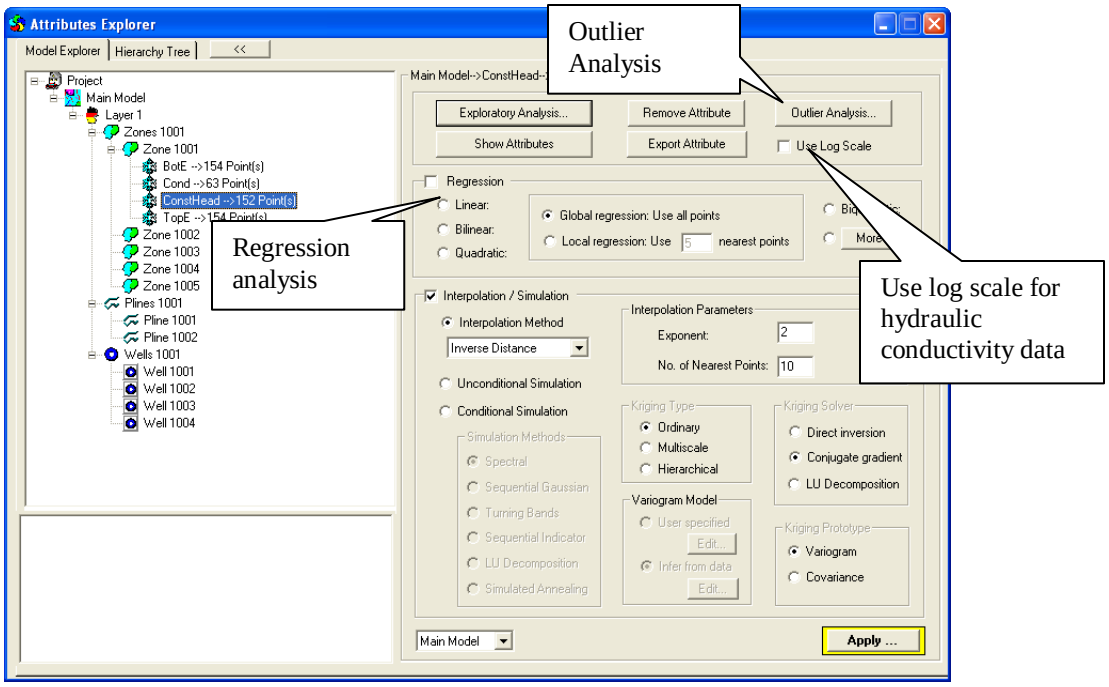


Figure 20-7 Regression-Outlier Analysis

The **Outlier Analysis** window is shown in **Figure 20-8**. The user selects the number of standard deviations beyond the mean to be used to define an outlier. The standard deviation is measured perpendicular to the regression surface that has been fit to the data. The user then selects the **Detect Outliers** button. The **Mean**, **Standard Deviation**, data ID (**Name**), and data value (**Value**) are displayed in the workspace.

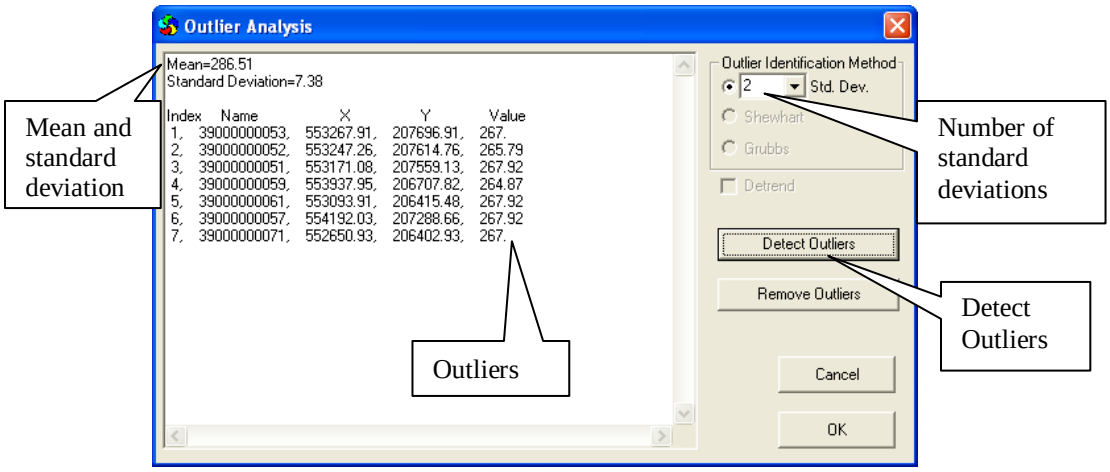


Figure 20-8 Outlier Analysis Window

If the user selects the **Remove Outliers** button, the message window shown in **Figure 20-9** opens. The user should click **Yes** to remove the outliers. Removal of outliers results only in the deletion of that particular scatter point attribute from the dataset. All other values for other attributes at that data location are not removed. Finally, the user should click the **OK** button to complete the outlier analysis.

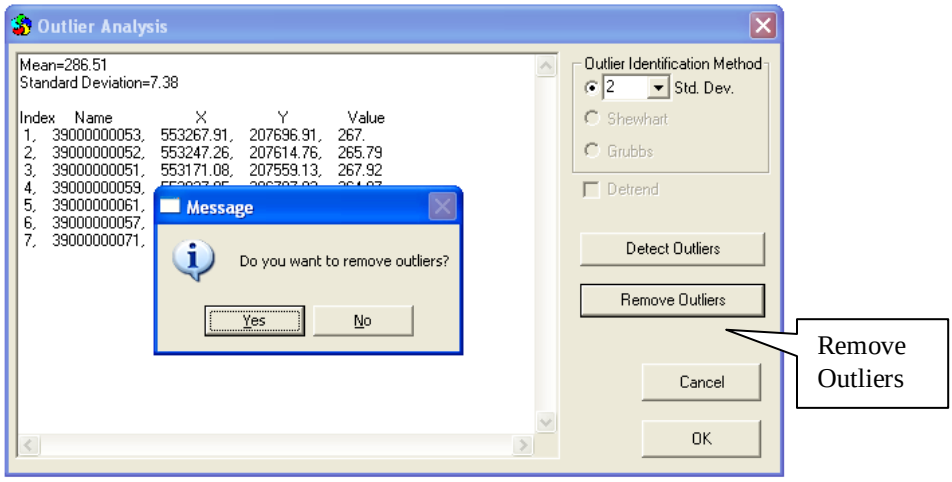


Figure 20-9 Removing Outliers

> **!** The user should experiment with the regression method and the number of standard deviations to use in determining data set outliers on a site-by-site basis. An outlier analysis should be performed on all scatter point attributes.

20.3. Data Interpolation

After data have been extracted and the outliers removed, it is necessary to interpolate the data. There are two main areas in the Right-Hand pane, **Regression** and **Interpolation / Simulation**. The user is therefore presented with three options for analyzing the scatter point data:

1) **Regression**,
2) **Interpolation / Simulation**, and
3) **Combination Analysis**.

To set up the particular analysis, the user simply may place a check mark in the appropriate box (**Interpolation / Simulation** is selected by default).

Checking both boxes sets up the **Combination Analysis** in which the data are first regressed and then the residuals analyzed through the desired interpolation / simulation scheme. If there are not enough data to perform a certain analysis, the software will prompt the user (in a separate window) with a message indicating the specific problem.

20.3.1. Regression

The user may choose from four preset regression types by selecting the desired one. The options are: **Linear**, **Bi-Linear**, **Quadratic**, and **Bi-Quadratic**. Formats for these regression types are explicitly given. The user may also choose a custom regression format by selecting **More** button. Clicking the **More** button opens the **Regression** window shown in **Figure 20-10**.

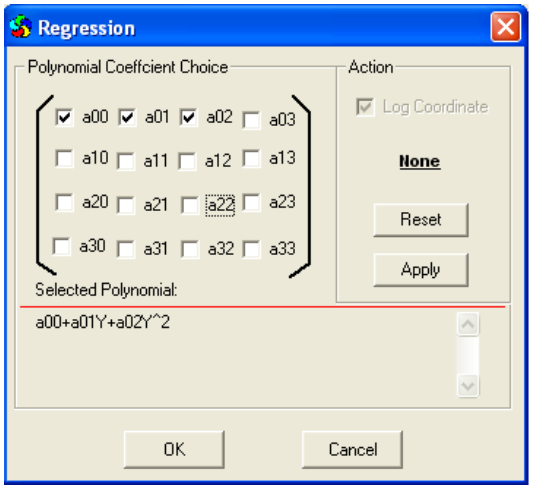


Figure 20-10 Manual Regression Analysis

In this window the user may choose the desired coefficients by placing a check mark in the associated boxes. The polynomial form is presented in the lower portion of the window. Clicking the **Apply** button shows the numerical form of the polynomial (the black **None** will change to red **Done**) and clicking the **Reset** button shows the algebraic form (changes the **Done** message to **None**). Clicking the **OK** button closes the window and sets the changes in the software. Clicking the **Cancel** button closes the window and discards the changes. Note that **Log Coordinate** is always selected and therefore the interface is disabled.

> **!** Using high order regressions can produce unrealistically high values, especially for large polygons. Use these with discretion.

20.3.2. Interpolation Methods and Simulations

The user may choose from three options in this area: **Interpolation Method** (default), **Unconditional Simulation**, and **Conditional Simulation**.

For the **Interpolation Method**, the user may choose (from the associated drop-down list) either **Inverse Distance** (default) or **Kriging Method**.

20.3.2.1. Inverse distance

One of the most commonly used techniques for interpolation of scatter points is the **Inverse Distance Weighted** (**IDW**) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced the most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases.

The simplest form of inverse distance weighted interpolation is sometimes called **Shepard's Method** (Shepard 1968). The equation used is as follows:

$$F(x, y) = \sum_{i=1}^{n} w_i f_i$$

where $n =$ *number of scatter points in the dat set,*

$f_i =$ *prescribed function values at the scatter points (e.g. the data set values), and*

$w_i =$ *weight functions assigned to each scatter point.*

The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=1}^{n} h_j^{-p}}$$

where $p =$ *arbitrary positive real number called the power parameter (typically, p = 2), and*

$h_i =$ *distance from the scatter point to the interpolation point.*

$$h_i = \sqrt{(x - x_i)^2 + (y - y_i)^2}$$

where (x, y): *coordinates of the interpolation point, and*

(x_i, y_i): *coordinates of each scatter point.*

The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to convergence.

The effect of the weight function is that the surface interpolates each scatter point, and is influenced the most between scatter points by those closest to the point being interpolated.

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's Method has been used extensively because of its simplicity.

There are two parameters in the **Deterministic Parameters** area that need to be set when using the **Inverse Distance** interpolation method. The exponent is set in the **Inverse Distance Exponent** field, and the number of nearest scatter points to use is set in the **No. of Nearest Points** field. The default values are 2 and the number of scatter points, respectively.

20.3.2.2. Kriging Method

The **Kriging Method** is based on the assumption that points that are near each other have a certain degree of spatial correlation, however points that are widely spread are statistically independent. Kriging is a set of linear regression routines that minimize estimation variance from a predefined covariance model.

There are two options that appear in the **Spatial Statistics Parameters** area when the **Kriging Method** is chosen. Choosing the **User Specified** option allows the user to explicitly define kriging parameters by accessing the **Input Parameters** window (by clicking the appropriate **Edit** button).

Choosing **Infer from Data** instructs the software to automatically determine the parameters and apply them. Clicking the associated **Edit** button opens the **Variogram** window. This window provides visualization of the statistical analysis and allows the user to tweak some of the automatic settings. Variogram modeling will be further explained later in this chapter.

!	It is important to note that the number of nearest scatter points to use is set in the **No. of Nearest Points** field. The greater the number of points used, the longer the computational time. Typically the number of nearest points is approximately that found within 1 correlation scale (from the variogram).

The basic equation used in ordinary kriging is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

$$\begin{aligned} &where\ n = number\ of\ \ scatter\ \ points\ in\ the\ data\ set, \\ &f_i = values\ of\ \ scatter\ \ points,\ and \\ &w_i = weights\ assigned\ to\ each\ scatter\ \ point. \end{aligned}$$

This equation is essentially the same as the equation used for inverse distance weighted interpolation, except that rather than using weights based on an arbitrary function of distance, the weights used in kriging are based on the model variogram. The weights are found through the solution of the simultaneous equations based on those below. Since there are now four equations and three unknowns, a slack variable, λ, is added to the equation set.

$$\sum_{i}^{n} w_j \gamma_{ij} + \lambda = \gamma_{ip}$$

$$\sum w_i = 1$$

By using the variogram in this fashion to compute the weights, the expected estimation error is minimized in a least squares sense. For this reason, kriging is sometimes said to produce the best linear unbiased estimate.

20.3.2.3. Unconditional Simulations

An unconditional simulation is one in which the simulated field is not constrained to pass through the (known) data points, honoring the covariance of the variogram only (instead of the explicit data values).

The **Unconditional Simulation** procedure generates a spatially-correlated random field based on sample statistical parameters. This procedure will generate new values for the locations corresponding to the measured values.

There are six options available in the **Simulation Methods** area when this procedure is selected:

1) **Spectral Algorithm** (the default),
2) **Sequential Gaussian Simulation**,
3) **Turning Bands**,
4) **Sequential Indicator**,
5) **LU Decomposition**, and
6) **Simulated Annealing.**

There are two options that appear in the **Spatial Statistics Parameters** area when the **Unconditional Simulation** is chosen. Choosing **User Specified** allows the user to explicitly define simulation parameters by accessing the **Random Field Options** window, clicking the appropriate **Option** button. Choosing **Infer from Data** instructs the software to automatically determine the parameters and apply them. Clicking the associated **Edit** button opens the **Variogram** window (Figure 20-11). This window provides visualization of the statistical analysis and allows the user to adjust some of the automatic settings. The user is referred to the variogram modeling (**Section 20.4**).

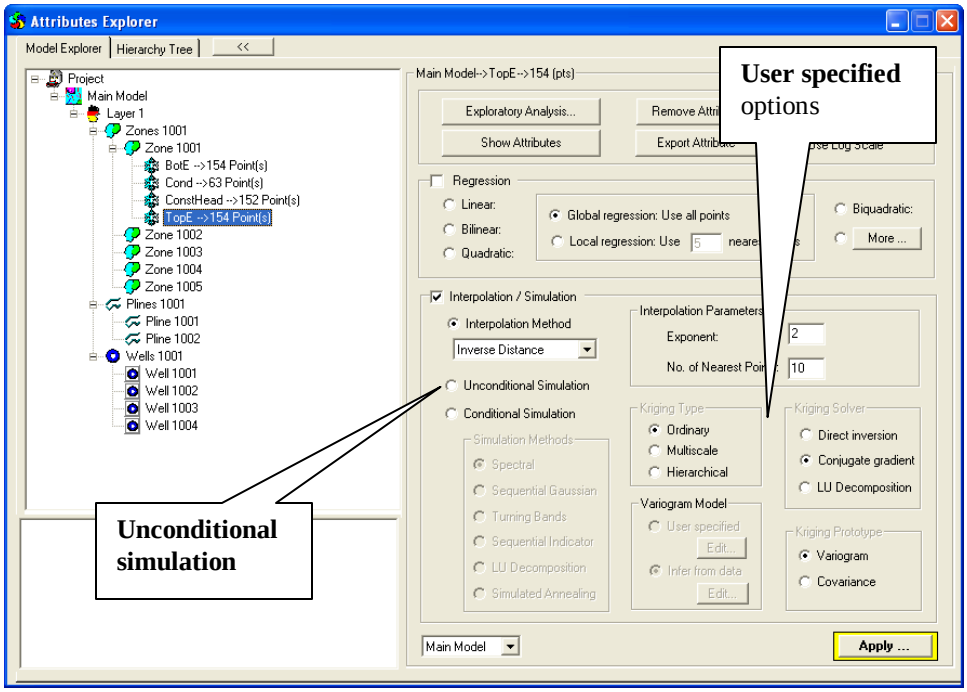


Figure 20-11 Interpolation Window with Additional Choices

20.3.2.4. Conditional Simulation

A conditional simulation is the one in which simulated field is forced to pass through known data points. This honors both the data and the statistics. The **Conditional Simulation** procedure generates a spatially-correlated random field based on sample statistical parameters. This procedure is similar to the **Unconditional Simulation** method, except the values for the locations corresponding to the measured values are held equal to those values when the field is generated.

There are two options available in the **Simulation Methods** area when this procedure is selected:

3) **Spectral Algorithm** (the default), and
4) **Sequential Gaussian Simulation**.

20.4. Variogram Models

The following example illustrates the interpolation of imported static water level elevation data. Highlight the model feature labeled **StartHead**, then uncheck the **Regression** option and select the **Interpolation/Simulation** button. Select the radio button to the left of the **Interpolation Method** option. The user has the option of selecting either **Inverse Distance** or **Kriging** as the interpolation method, and it is generally recommended that the user select the **Kriging Method**. This is accomplished by scrolling down to **Kriging Method** in the scroll-down window below **Interpolation Method** (**Figure 20-12**). The information contained within the boxes in the lower right-hand corner of the **AE** window change to fit whichever interpolation method is selected, **Inverse Distance** or **Kriging**. It is also necessary for the user to edit the **semi-variogram** model parameters.

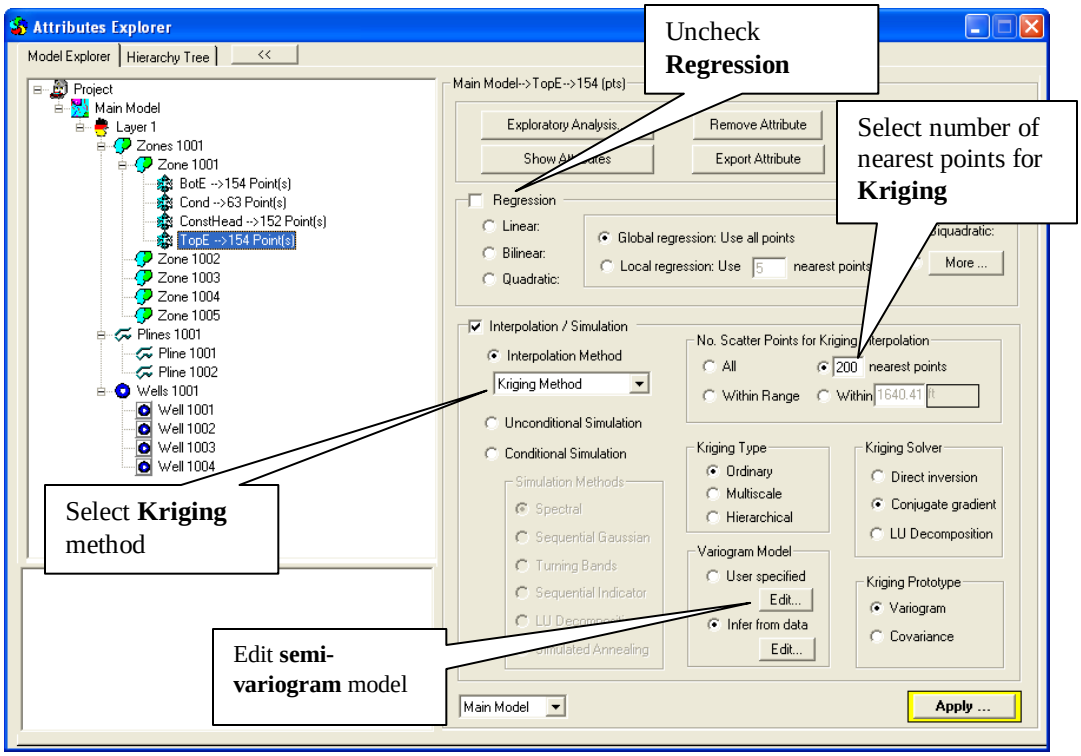


Figure 20-12 Kriging Analysis

The window shown in **Figure 20-13** opens after the user selects the **Infer from data Edit** button. By default, the program attempts to automatically fit an exponential **semi-variogram** model using **least squares** approximations. The data and the **least-squares** fit model are shown in the figure.

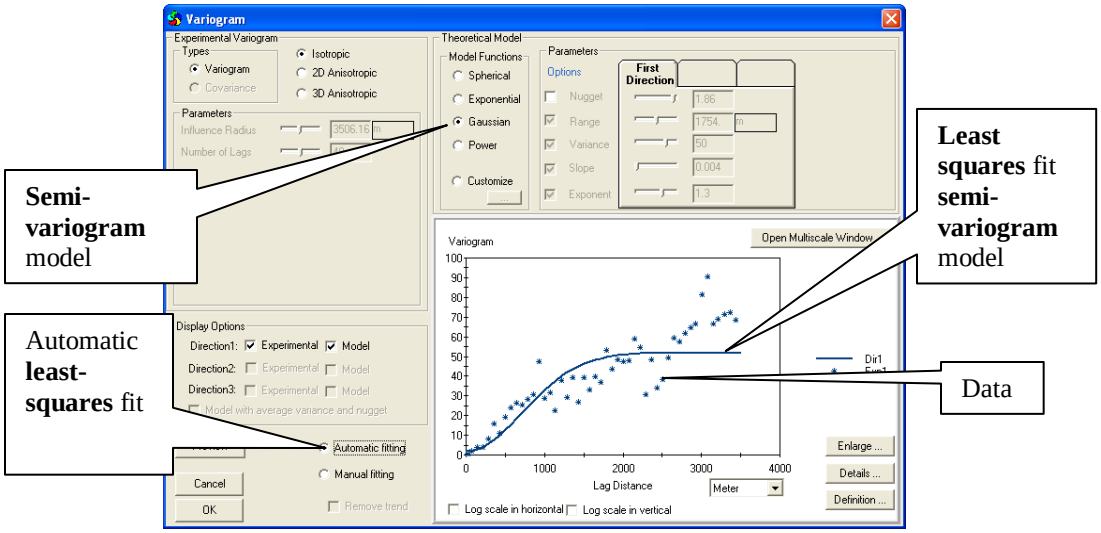
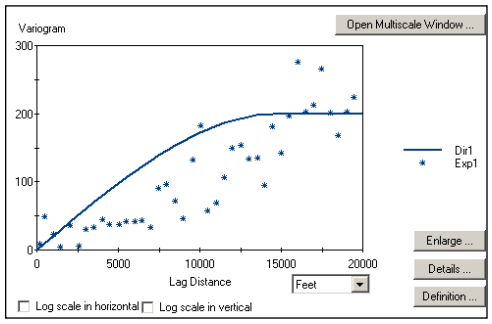
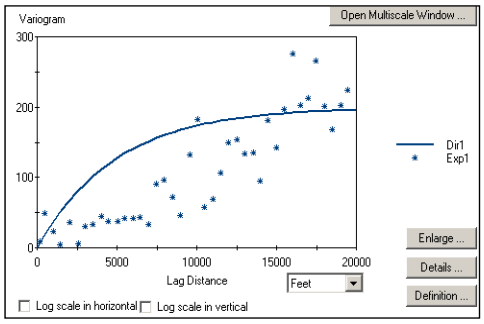


Figure 20-13 Automatic Fitting of the Semi-Variogram Model

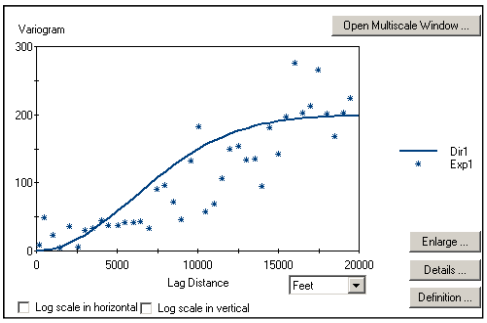
It is recommended for the user to manually fit a semi-variogram model to the data. This is accomplished by selecting the radio button next to **Manual fitting**, and then selecting the **Preview** button to initialize the manual fitting process. The first step in the process is to select the model that best fits the data distribution. **Figure 20-14** (**A** through **D**) show the curves for the **Spherical**, **Exponential**, **Gaussian** and **Power semi-variogram** models, which can be fitted to data.



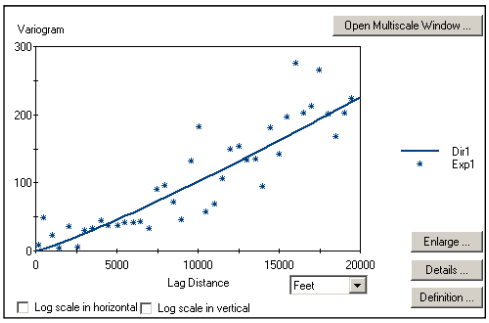
A) Spherical



B) Exponential



C) Gaussian



D) Power

Figure 20-14 Manual Fitting of Semi-Variogram Models

It is necessary to select the **Preview** button after changing model selection to re-initialize the manual fitting process. The parameters, **Nugget**, **Range**, and **Variance** are illustrated in **Figure 20-15**, which is a semi-variogram fit to log K data. These parameters apply to the first three semi-variogram models (**Spherical**, **Exponential**, and **Gaussian**).

An **experimental variogram** is a plot showing the **variance** of data attributes in relation to the separation distance between the locations of the measured attributes. The experimental variogram is calculated from a spatial data by averaging one half the differences squared of the *z*-values over all pairs of observations, using the specified separation distance (**h**,) that is represented mathematically as:

$$\gamma(h) = \frac{1}{2n(h)} \sum_{|s_i - s_j| = h} (z_i - z_j)^2$$

where s_i *and* s_j *: points in spase separated by h,*

z_i *and* z_j *: attribute values at* s_i *and* s_j*,*

n *: number of pairs in the data set separated by lag h, and*

$\gamma(h)$ *: experimental variogram.*

A **theoretical variogram** is a mathematical function optimally fit into the variogram points of an experimental variogram plot. Some commonly used mathematical models are **spherical**, **exponential**, and **Gaussian**. The parameters governing the shape of these mathematical curves are **sill**, **range**, and **nugget** of the variogram. **Figure 20-15** shows a theoretical variogram fitting into an experimental variogram.

Range can be defined as the minimum lag distance **h**, for which the value of the variogram becomes equal to variance. The **correlation scale** is approximately 40% of range. The data are still correlated within range, but after incorporating the scale the correlation is weaker. Within the **Range**, the **Variance** is assumed to have reached its maximum value for the data. The **Variance** is the difference between the **Nugget** and the **Sill**. Each of these parameters may be varied either by moving the sliding button until the desired value is achieved, or typing a value into the window to the right of the sliding button.

Nugget in a variogram represents measurement error; data noise and the variability that cannot be resolved also subscale variability (**Figure 20-15**).

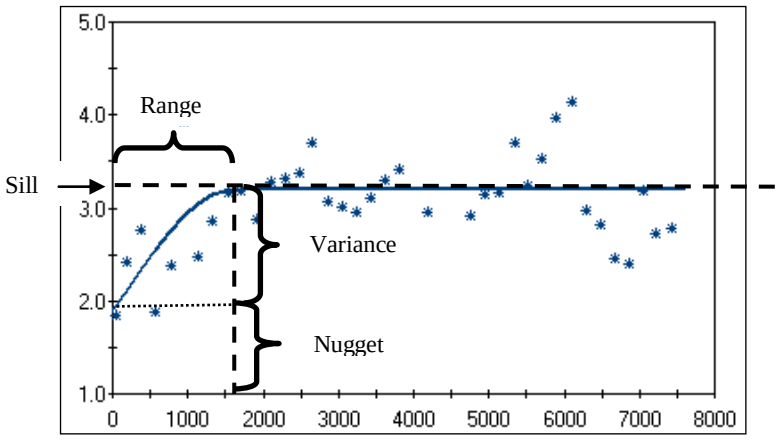


Figure 20-15 Illustration of Nugget, Range, and Variance

The **Power** model lets the user adjust the **Nugget**, **Slope**, and **Exponent** to achieve a statistical fit to the data. In **Figure 20-16**, a **power semi-variogram** model is fit to the static water level elevation data.

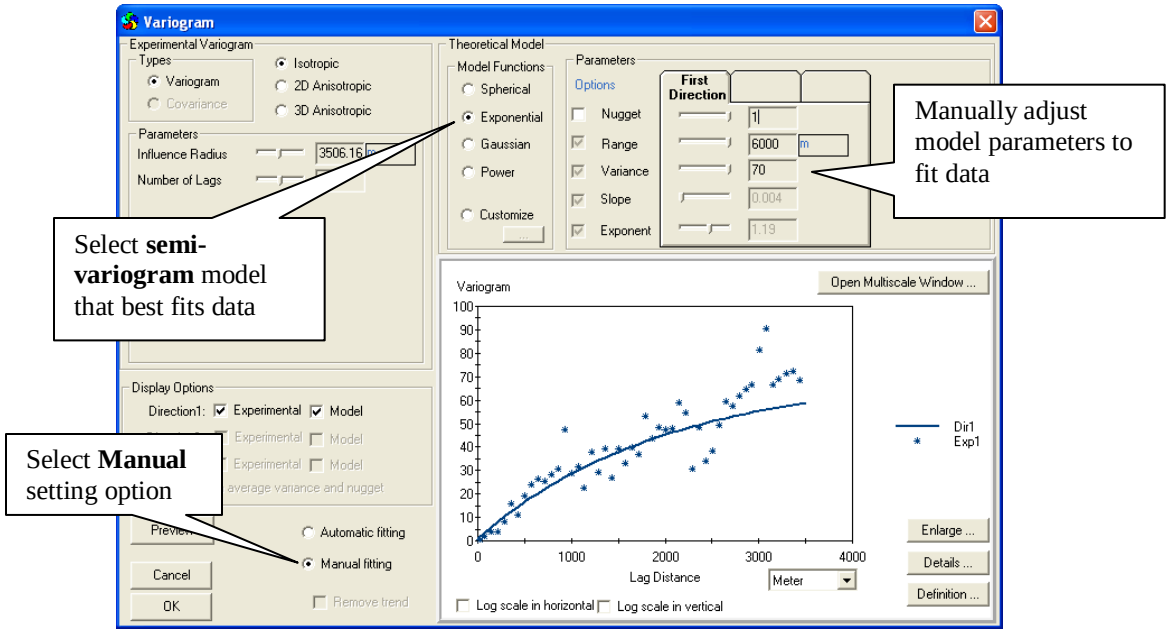


Figure 20-16 Manual Fitting of Semi-Variogram Model

Before performing data interpolation, it is necessary to specify the number of scatter point data that are to be included in the **Kriging semi-variogram** model. In general, with a relatively small number of data points, it is best to use all of the data. By comparison, if there are several hundred data points, it is best to either specify the number of points to use (**nearest points**), use all points within the **semi-variogram** range (**Within Range**), or all points within a certain radial distance (**Within __ ft**).

! It is recommended that the user experiment with variogram parameters and observe their impact on the interpolated attribute surface.

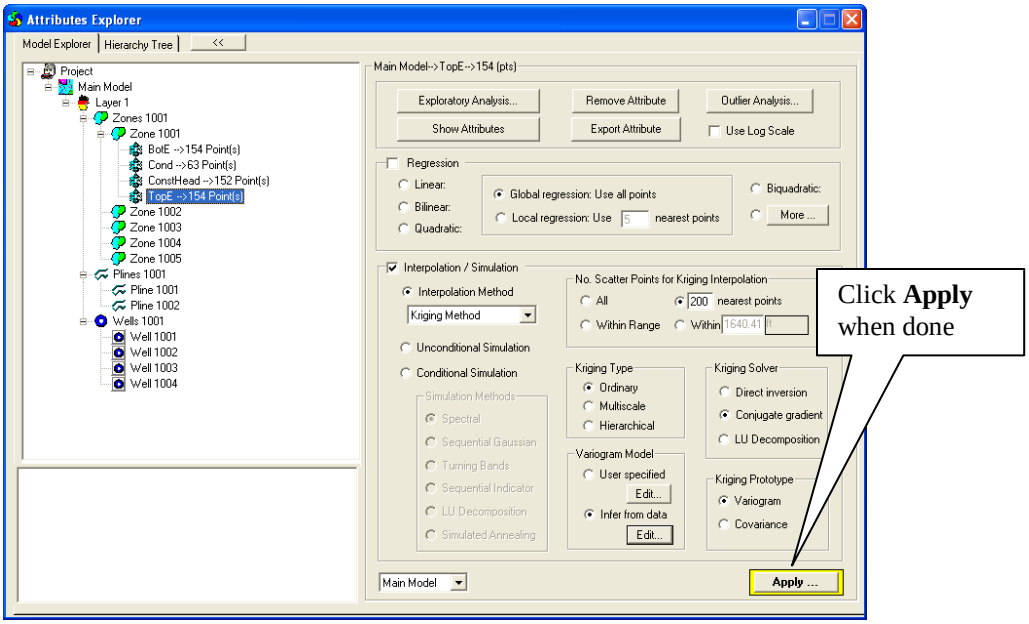


Figure 20-17 Changing Number of Nearest Points Used in Kriging Interpolation

A different semi-variogram model should be fit to each scatter point attribute. This is accomplished by selecting the attribute from the **AE** window and repeating the model fitting process. After a semi-variogram model has been fit to all scatter point attribute data, the user then selects the **Apply** button to apply all **semi-variogram** models. The **AE** window shown in **Figure 20-17** closes and the **IGW Modeling Environment** window appears.

Chapter 21 THREE-DIMENSIONAL VISUALIZATION

IGW offers the ability to display models in three dimensions, either as a surface or as a volume. Each method contains many features that will be beneficial to the user, from cross-sectional analysis to zooming in directly on a plume's location by planar coordinates and depth. A detailed explanation of these capabilities is below.

21.1. Demonstration of 3D Surfaces

IGW Version 4.7 allows the model to be viewed as a three-dimensional surface, with x, y, and z coordinates that correspond to the values stated for distance and elevation. An example of the interface is shown below in **Figure 21-1**.

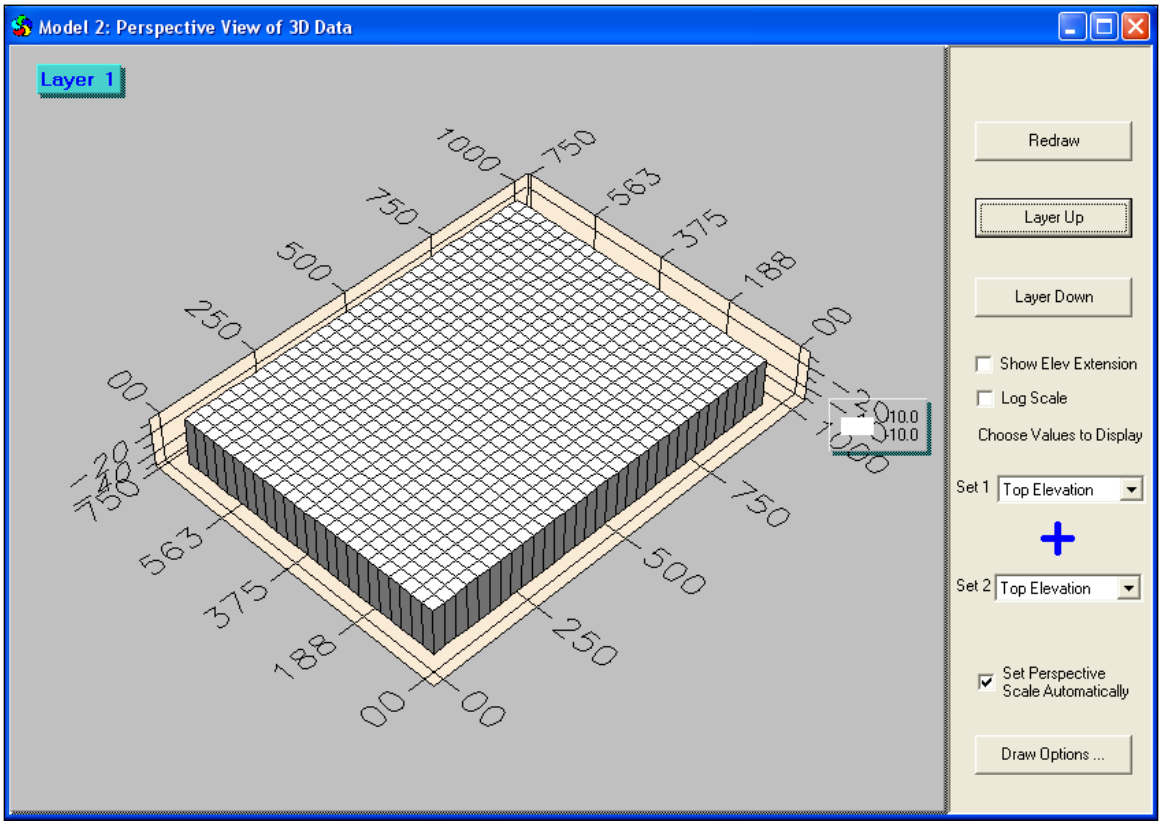


Figure 21-1 Main Window for Demonstrating 3D Surface

The interface allows for many different forms of visual manipulation, seen in the right-hand side of the screen. Options include 'Redraw', which resets the model to its original form, 'Layer Up' and 'Layer Down', which show different layers of the model, and options to show elevation extension and the model on a graphic log scale. The remainder of the options are discussed below.

21.1.1. Parameters to Display

This feature allows the user to select which attributes they would like to display on their three-dimensional surface model. Along with 'Top Elevation', there are several options available for display: 'Bottom Elevation', 'Aquifer Thickness', 'Head', 'Concentration', 'Velocity X, Y or Z', 'Conductivity', and 'Transmissivity'. **Figure 21-2** and **Figure 21-3** show the selected options and drop down menu for options respectively.

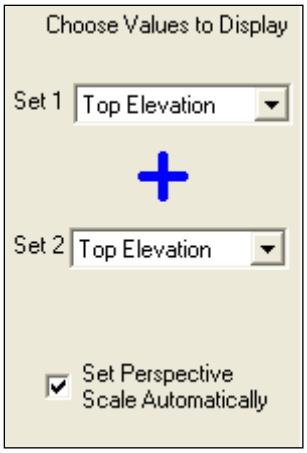


Figure 21-2 Display Menu

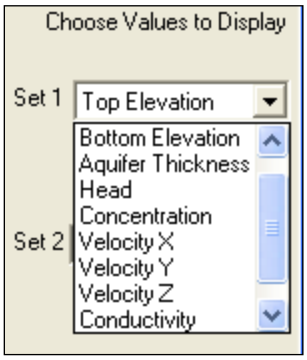


Figure 21-3 Drop-Down List for Display Variables

21.1.2. Draw Options

Selecting this button opens a vast array of drawing options to the user. There are thirteen different functions within this region, and each has several sub-options. A detailed discussion of each of these options follows.

21.1.3. 3D Chart Control Properties

3D chart control properties (**Figure 21-4**) are briefly given at **Table 21-1**.

R The user should remember that they can always find out the best settings to demonstrate the model in a descriptive fashion.

Table 21-1 3D Chart Control Properties

3D CHART CONTROL PROPERTIES		
TAB	**SUB BUTTONS**	**FUNCTION**
Control	General	Loading/saving 2D chart files in *.oc2 format
	Border	Changing the style of the border, distance of the title, and legend on the chart border.
	Interior	Changing the color of the border and the grids of the chart
	Image	Uploading image file to the background of the chart
	About	Credits for the manufacturer of the chart interface
Axes	General	Changing the label and view of the axes
	Scale	Changing the scale of the axes
	Title	Adding title and changing font style/font size of the title
	Grid Lines	Adding/removing gridlines for axes
	Value Labels	Changing number of intervals for the axes
	Stroke Font	Changing font style/font size of the axes
Chart Group	General	Changing the style (bar/surface) and the view of the chart
	Data	Loading/editing data, changing grid size
	Labels	Adding/removing/modifying labels
	Elevation	Changing the color of chart group components
	Internet	Giving data path for data from internet
Styles	General	Adding/removing styles for chart display
	Fill Style	Changing the color of contour area
	Line Style	Changing the line style of 3D graphic display
Titles	General	Switching between header/footer titles
	Label	Changing the label name of the title
	Location	Changing the location of the title in the model graphic
	Border	Changing the features of the title border
	Interior	Changing the background/foreground colors of the title box
	Font	Changing the font size/style of the title
	Image	Adding images to the background of the title box
Legend	General	Changing the location and the view of the legend
	Labels	Changing the legend label
	Location	Changing the legend location by entering coordinates
	Border	Changing the style of the legend border
	Interior	Changing the legend box colors
	Font	Changing the legend font style/font size of the legend text
	Image	Adding image to the background of the legend box
Chart Area	Location	Changing the location of the chart area
	Border	Changing the style and width of the chart border
	Interior	Changing the color of the chart area
	Image	Adding image to the background of chart area
Plot Cube	General	Changing the scale of the chart
	Ceiling	Changing the view of the top of the chart area
	Floor	Changing the view of the bottom of the chart area
	Viewport	Changing the scale and horizontal/vertical shifting of the chart
	Interior	Changing the color of the chart
Chart Labels	General	Adding/removing labels
	Attach	Attaching labels
	Label	Changing the name of the chart labels
	Border	Delineating chart labels
	Interior	Changing the background and foreground color of chart label boxes
	Font	Changing the font style and size of the chart label boxes
	Stroke Font	Changing the font style and size of chart labels
	Image	Importing image to the background of the chart label boxes
Bar	General	Changing the row/column format for bar style chart groups
	Colors	Changing bar colors
Surface	General	Changing the view for surface style chart groups
View 3D	General	Changing 3D view settings of chart
Levels	General	Changing contour levels for the chart

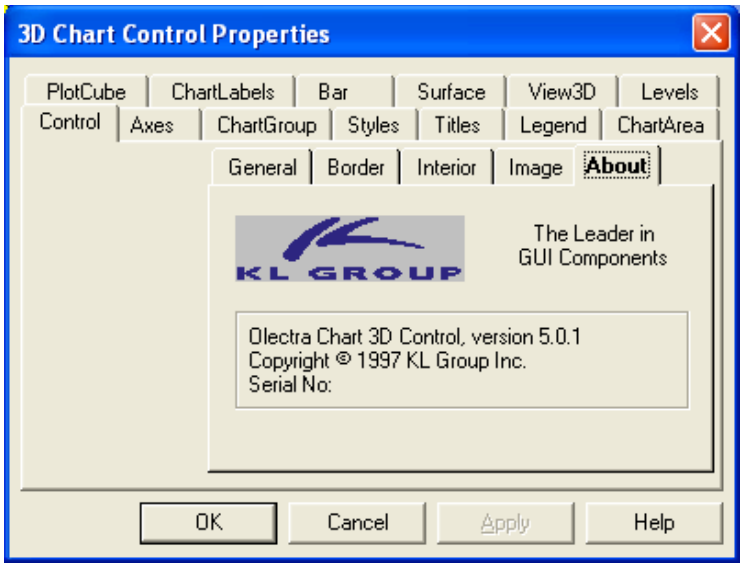


Figure 21-4 3D Chart Control Properties

21.2. Visualizing Model as 3D Volume

The three-dimensional surface feature is a powerful tool for viewing models created in **IGW**. However, the second main option, viewing as a three-dimensional volume, offers the user unprecedented ways of viewing and even dissecting their model.

The main display screen is shown below in **Figure 21-5**, and contains an example (highly simplified) model for consideration. This model will be the one used in future demonstrations of the capabilities of this feature.

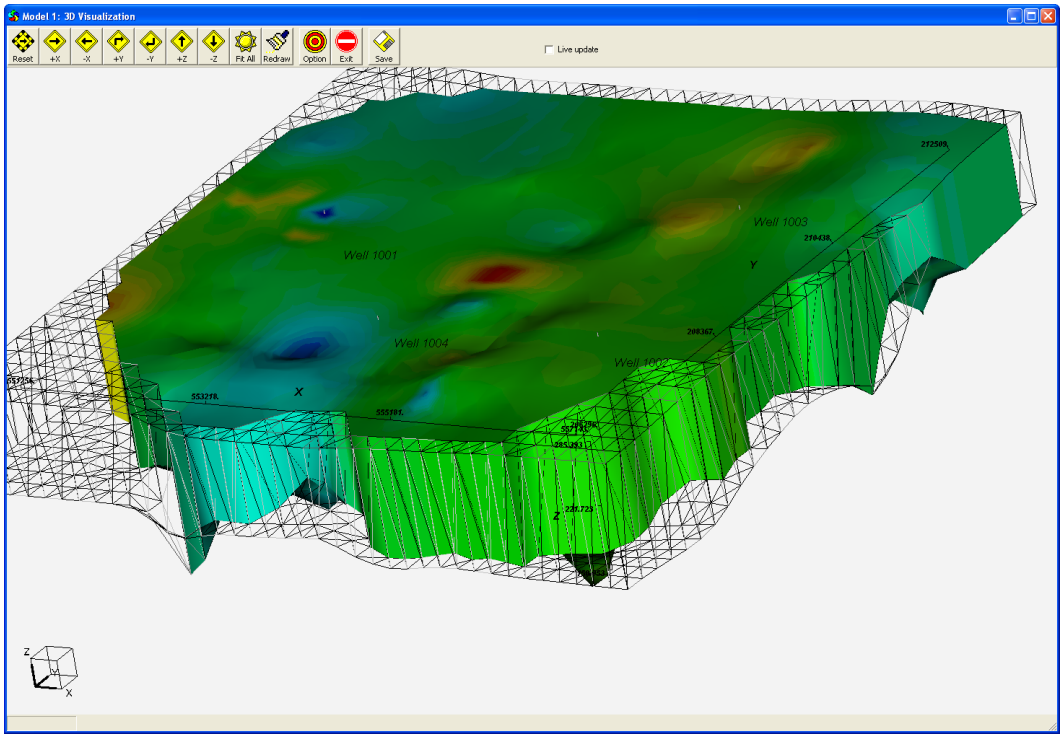


Figure 21-5 3D Volume Main Display Window

21.2.1. Graphic Display Options

The buttons along the top row allow for the user to increase or decrease the viewing area in any direction (x, y, or z). Put another way, this is a means of zooming in and out of the model, based on one or more axis the user adjusts.

The **Fit All** button allows the entire model to fit within the working screen, whereas the **Redraw** button resets the model inside the Working Area to its original conditions.

21.2.2. Options

Selecting the **Option** button, like the **Draw Options** button in 3D Surface, opens a wide array of editing features to the user. Additionally, the user may use this button to facilitate three-dimensional cross-sections of their model, in a number of ways.

21.2.2.1. General Aspects

This tab allows the user to enlarge (or shrink) the graph by a stated factor, on any axis. For instance, if the user wants to see the x-axis in a larger context, they could increase the enlarge factor from a value of 1 to a value of 2, 5, 10, etc. The same holds true for the Y and Z-axes. Additionally, options for displaying the model grid are given in this tab (**Figure 21-6**).

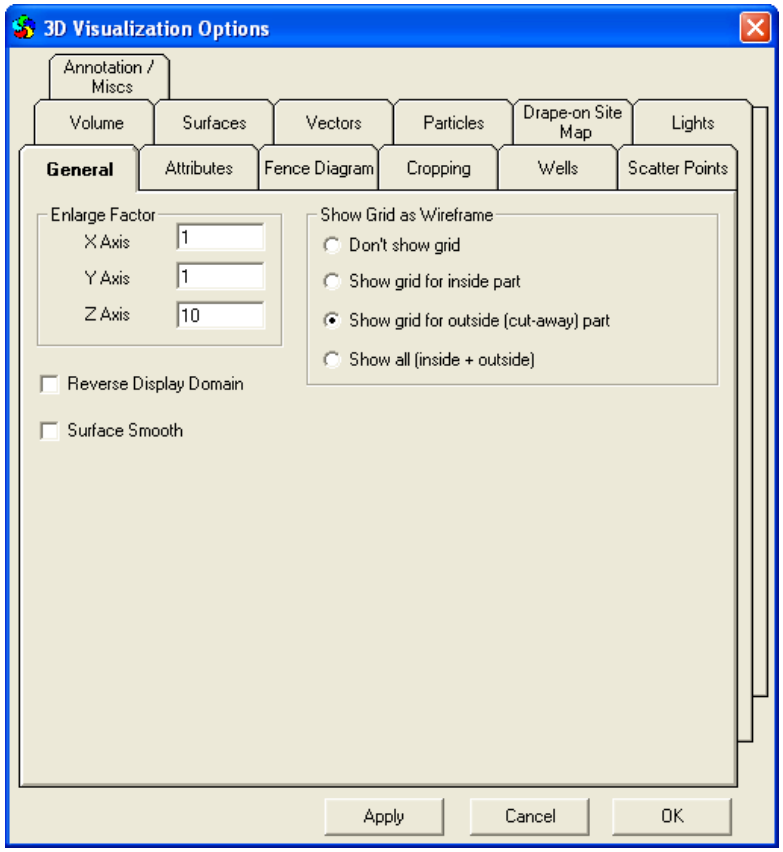


Figure 21-6 3D Visualization Options

21.2.2.2. Attributes

This tab offers several options for the three-dimensional volume model. In particular, the user can choose to show or hide the conductivity, head, and/or concentration attributes along with dry areas. This option is also given to map the model to an isosurface. Minimum and maximum value options are given for the model, if the user wishes to place boundaries on the limits of their simulation. Several style feature options are also given in this tab (**Figure 21-7**).

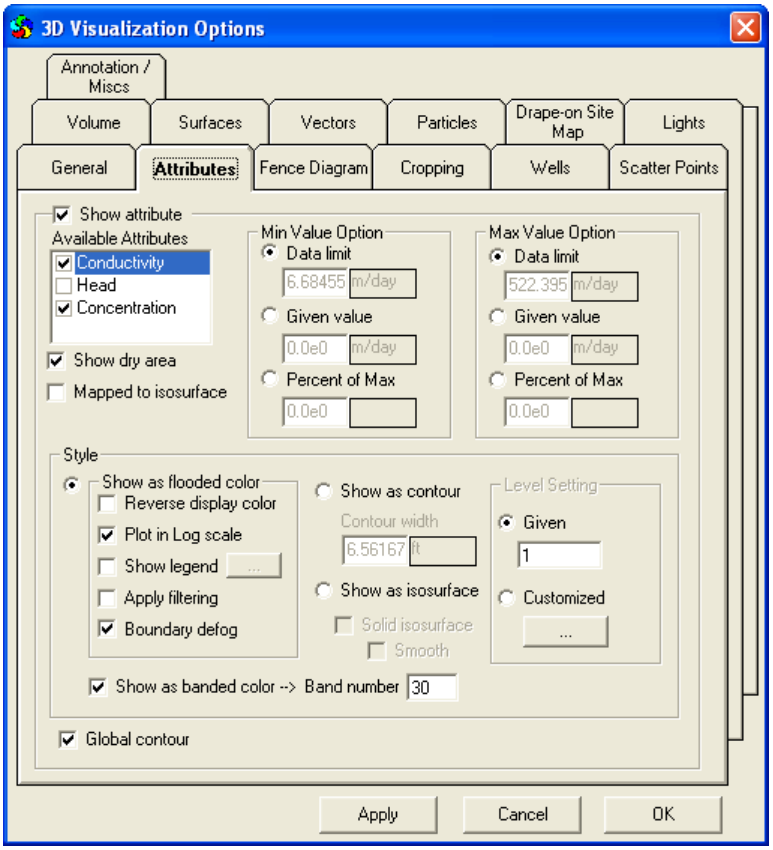


Figure 21-7 3D Visualization Attributes

21.2.2.3. Fence Diagram

This feature is one of the highlights of **IGW**, and offers an incredible new way in which to view three-dimensional models. The user has the option to dissect the model into a series of linear planes (or crosses through the model). Using the features described in **Section 21.2.3**, the graph may be rotated any number of directions, showing the full extent of the model's outcome by looking at one or more simultaneous cross-sections through the simulation.

The **Fence Diagram** tab specifically allows for either pre-set 'slices' through the model (one or two crosses), or the user can custom-design the slices they wish to view. Additionally, these can apply to the volume, surface, water table, sitemap, isosurface and vectors of the simulation.

When the desired slice is selected, the user should click the **Apply** button to make these features active within their model. **Figure 21-8** displays the **Fence Diagram** interface window.

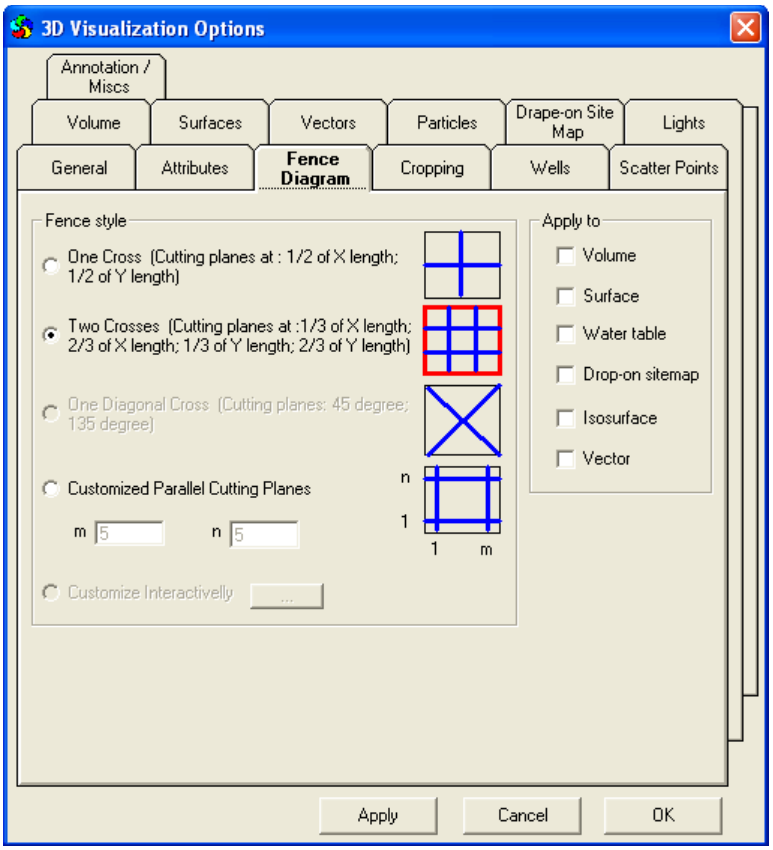


Figure 21-8 Fence Diagram Options

An example of fence diagram is shown in **Figure 21-9**.

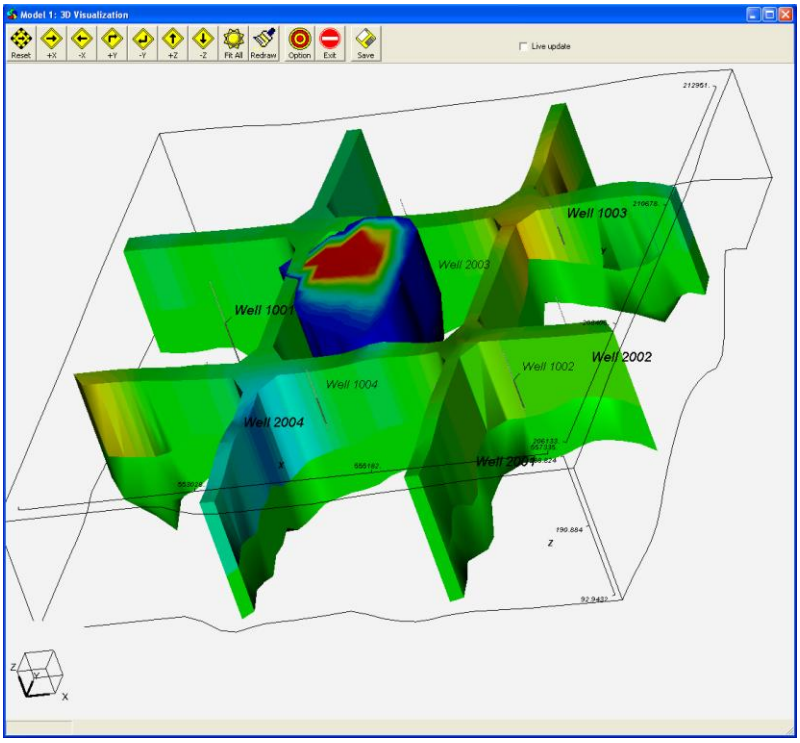


Figure 21-9 Fence diagram

21.2.2.4. Cropping

The second way of dissecting the model is **Cropping**, in which the user can cut 'blocks' off of the three-dimensional volume model to view the interior workings of the simulation. This can be useful in any number of ways including observation of contaminant plume migration, well drawdown effects, local variation within the water table, and so on. The options available include seven styles (**Figure 21-10**), from a 1/8th cut of the model, all the way up to a 7/8th cut of the simulation. The user may also manually specify the cut size and location within this interface. These cuts, like the fence diagram, can be applied not only to volume, but also the surface, water table, isosurface, the sitemap and the vectors of the model.

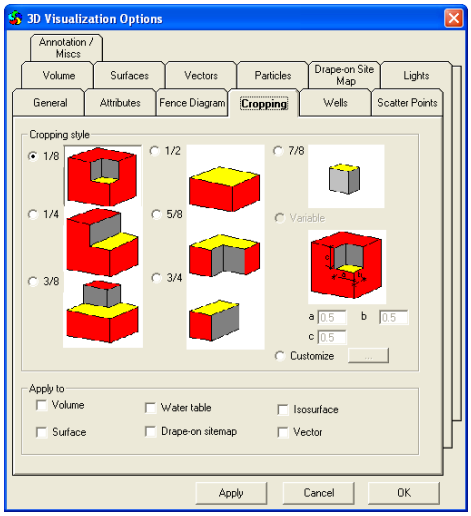


Figure 21-10 Cropping options

Figure 21-11 shows a 1/8th cut of a 2 layer model, with variable conductivity in the upper layer and constant in the lower one.

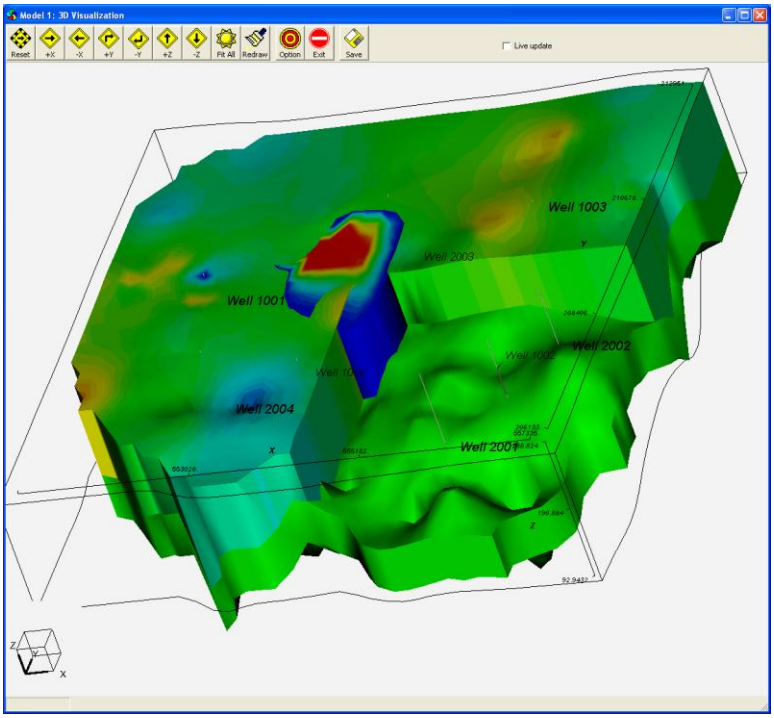


Figure 21-11 1/8th Cropped Model

Cropping can also be customized by selecting ‘Customize’. Clicking options button will after ‘Customize’ is selected opens the ‘Customize Cropping’ window as shown in **Figure 21-12**. This window lets the user draw different shapes from the menu and select the layer in which the shape is to be applied.

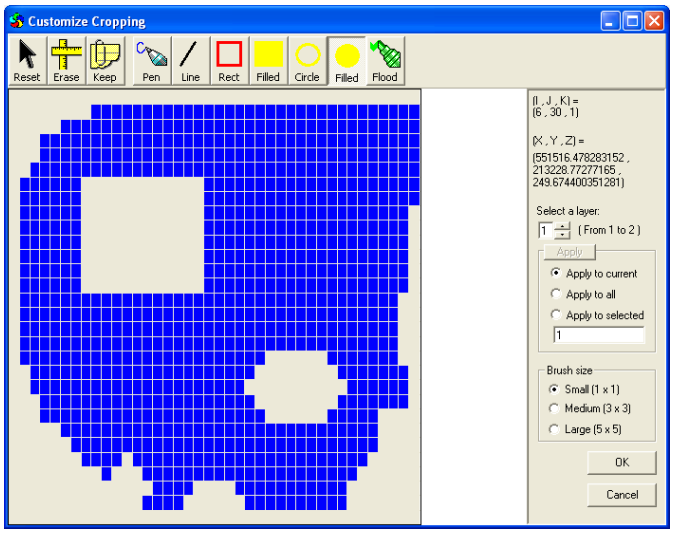


Figure 21-12 Customize Cropping window

The user can experiment with options in this window to learn the effects which can be created in 3D visualization. An example of customized cropping is shown in **Figure 21-13**.

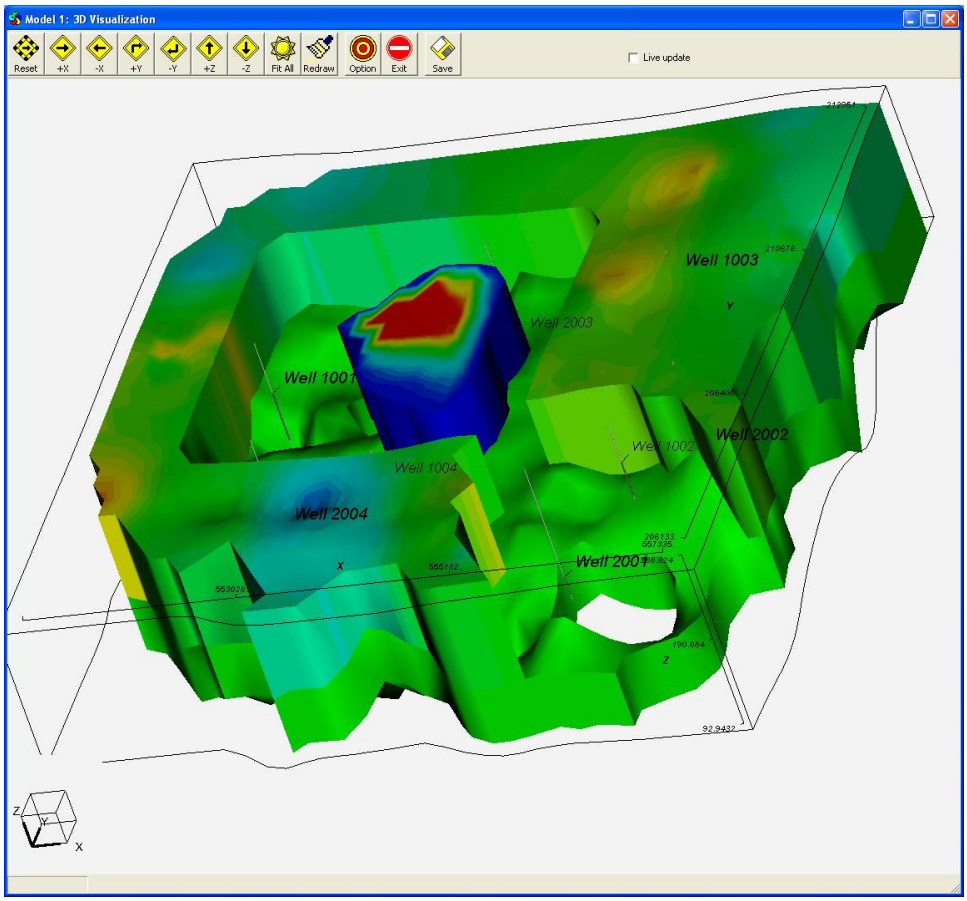


Figure 21-13 Custom cropped model

21.2.2.5. Wells

This tab (**Figure 21-14**) gives options for the well screen location, cap height, bottom elevation, well diameter and color options within the three-dimensional model.

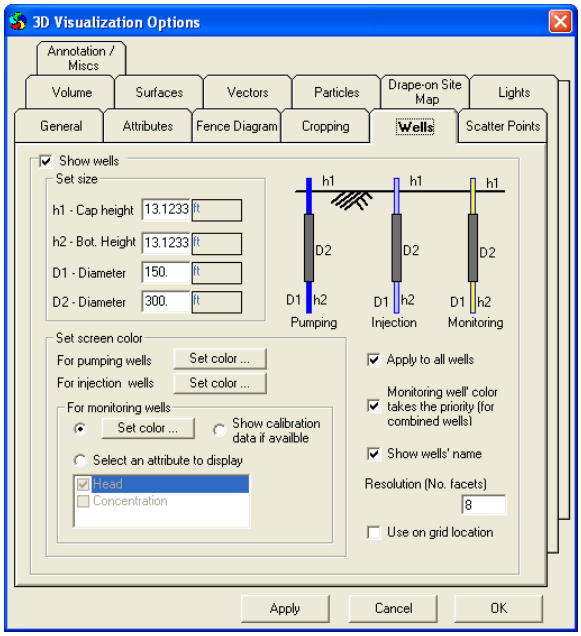


Figure 21-14 3D Visualization Options for Wells

Using these options, the user can enhance the appearance of wells. It can be seen in **Figure 21-14** that well are presented more prominently than how they look like in **Figure 21-15**.

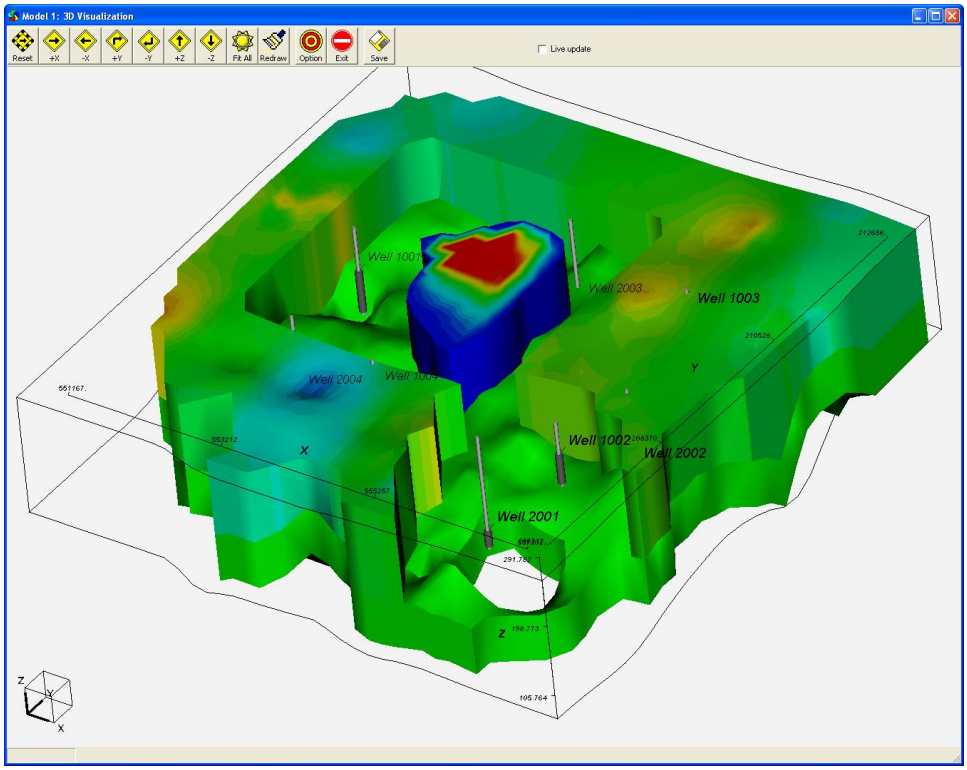


Figure 21-15 Wells seen in a customized cropped model

21.2.2.6. Scatter Points

This feature is similar to the **Wells** interface, in that different elevations and diameters of scatter point locations can be specified. The upper and lower-bound contaminant concentrations may also be denoted within this interface (**Figure 21-16**).

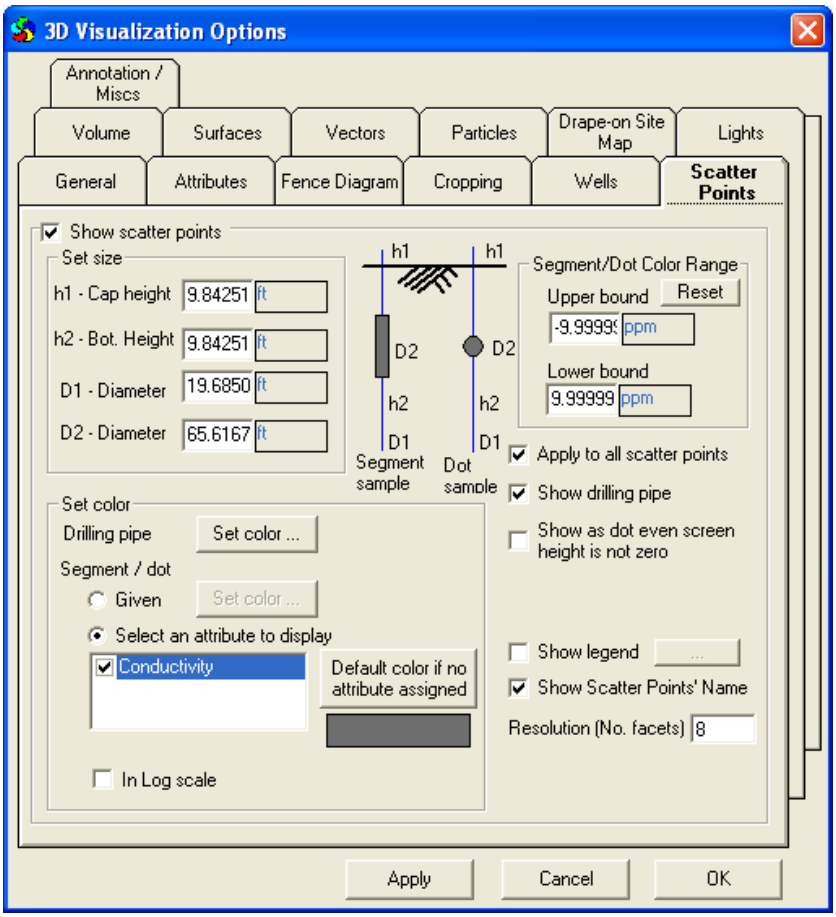


Figure 21-16 3D Visualization Options for Scatter Points

Legend options for scatter points are available through **Legend** button (**Figure 21-17**).

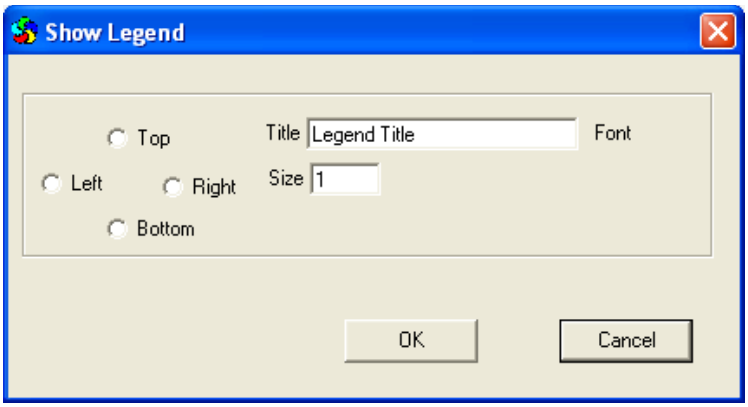


Figure 21-17 Legend Window

21.2.2.7. Volume

This feature allows the user to specify the layers they wish to view in their three-dimensional volume model (**Figure 21-18**).

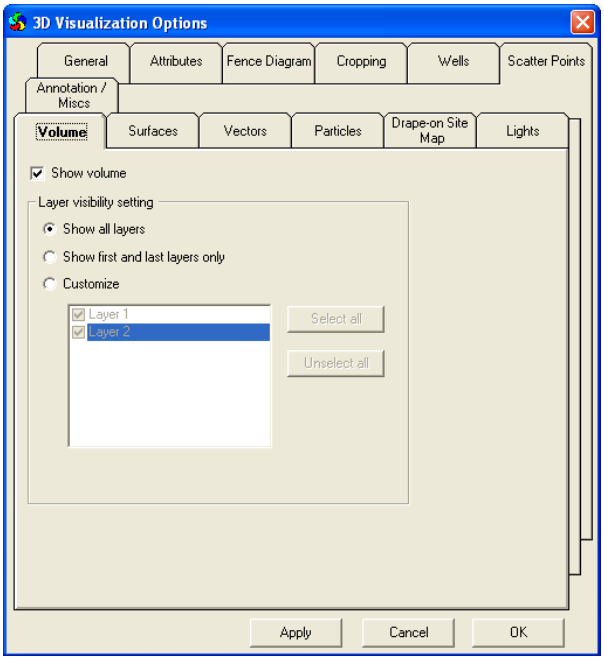


Figure 21-18 3D Visualization Options for Demonstrating Volume

21.2.2.8. Surfaces

This feature is identical to the **Volume** tab, in that the user may specify which surfaces to display within their three-dimensional model (**Figure 21-19**).

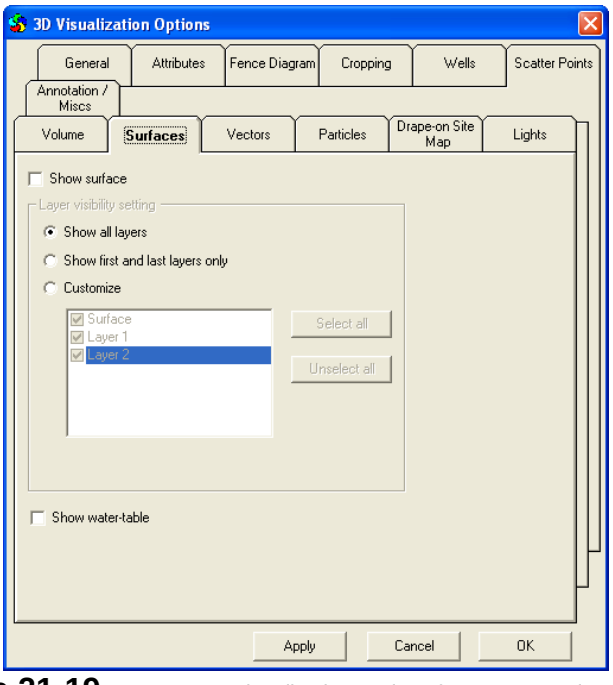


Figure 21-19 3D Visualization Options for Demonstrating Surfaces

21.2.2.9. Vectors

This tab gives options for displaying velocity vectors within a three-dimensional model, including vector length, color, and frequency of location (**Figure 21-20**).

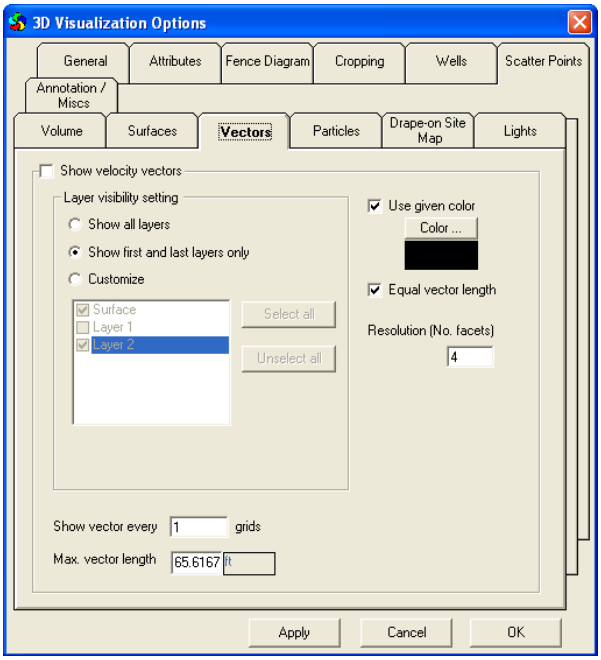


Figure 21-20 3D Visualization Options for Vectors

21.2.2.10. Particles

This feature gives options for displaying particles within a three-dimensional model, including color, shape, style, and diameter of each particle (**Figure 21-21**).

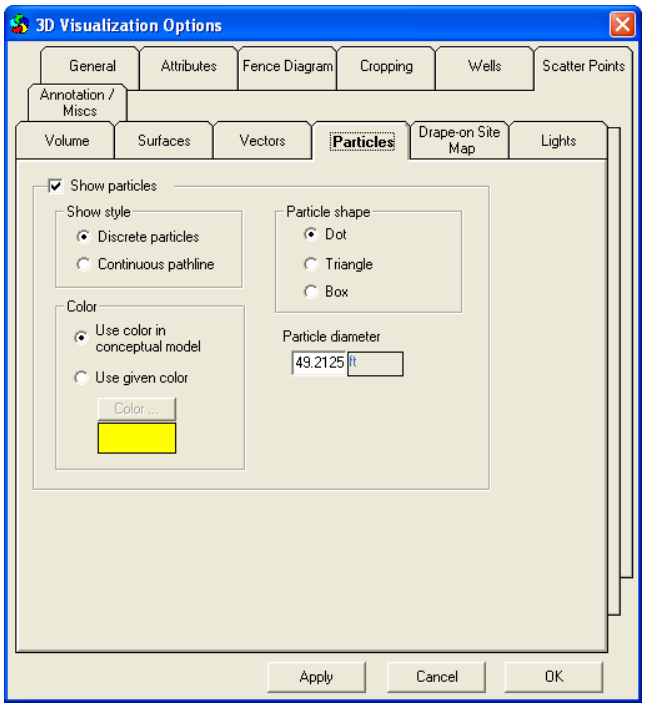


Figure 21-21 3D Visualization Options for Particles

21.2.2.11. Drape-On Site Maps

This tab gives options for adding and editing a map that will be overlain (or 'draped') onto the three-dimensional model (**Figure 21-22**). Features include **style**, **rotation**, **display**, and **image mode** options.

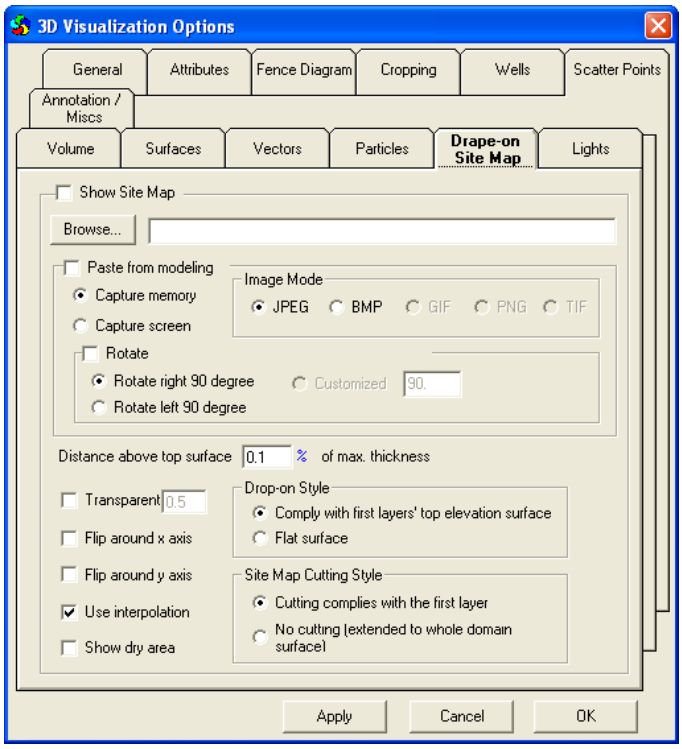


Figure 21-22 Drape-On Site Map Window

21.2.2.12. Lights

This feature allows for a 'top light' to be added to the model, changing its view to the user (**Figure 21-23**).

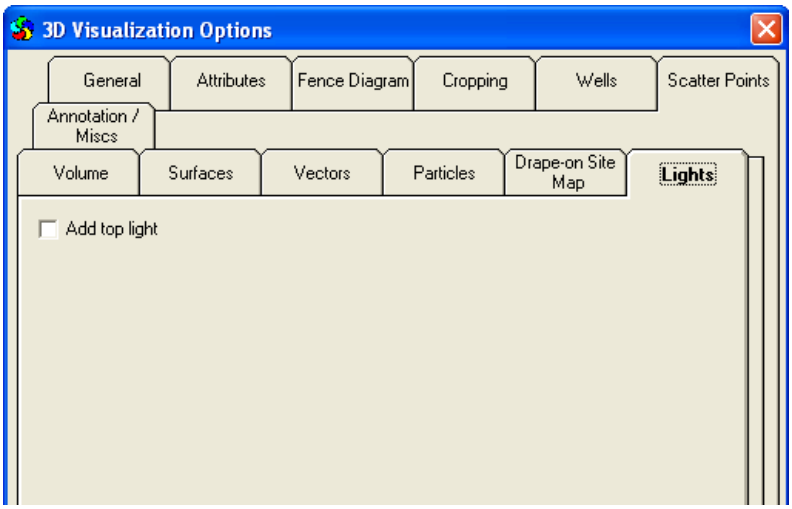


Figure 21-23 Lighting Options for 3D Visualization

21.2.2.13. Annotation / Miscs

This tab allows for manipulation of the three-dimensional model in terms of font style, whether to display the 3D axes and the respective tick numbers for each, and the background color (**Figure 21-24**).

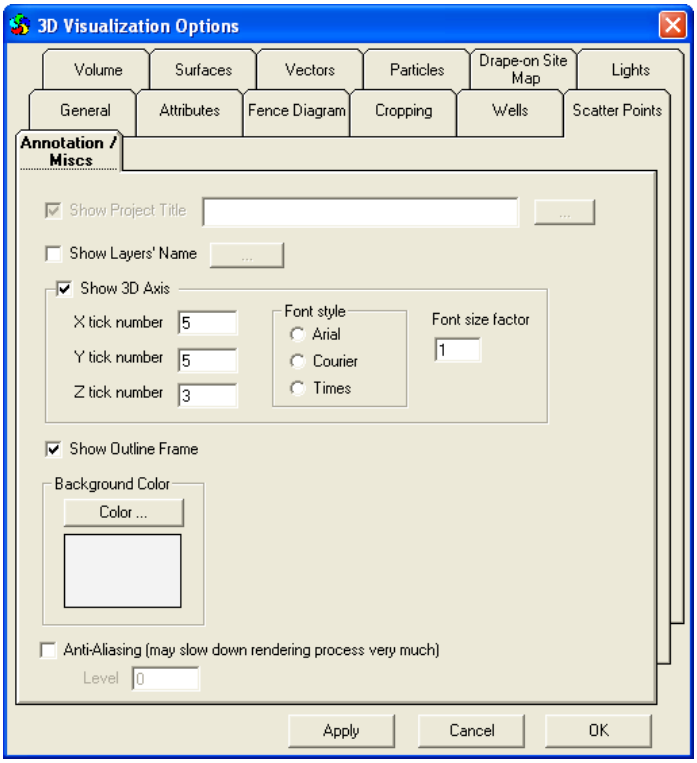
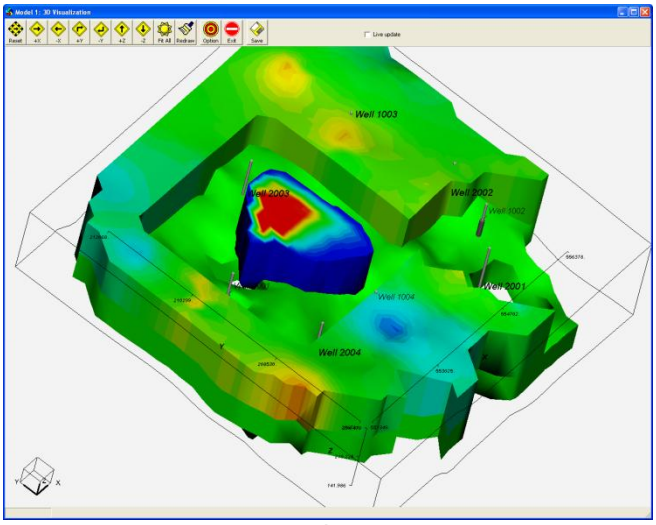


Figure 21-24 Miscellaneous Options for 3D Visualization

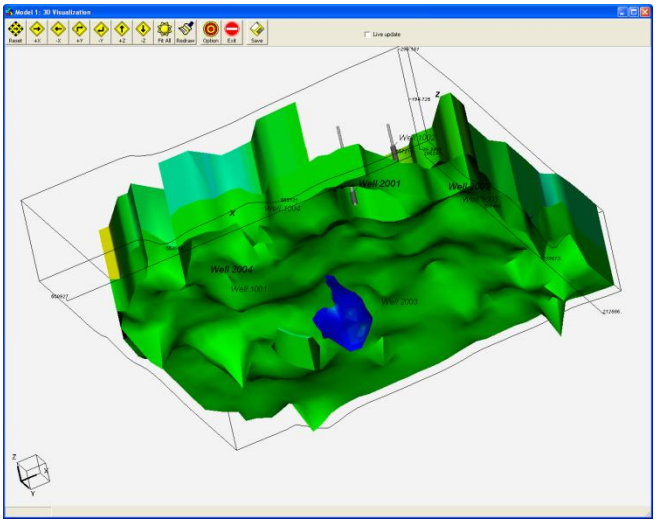
21.2.3. Manipulating Graphics in 3D

When the new window is opened that displays the three-dimensional volume model, the user has several ways in which to change the view. In particular, they can click either above the model or below the model, which will rotate the model in that respective direction. The same can be done for clicking on the left and right side of the model.

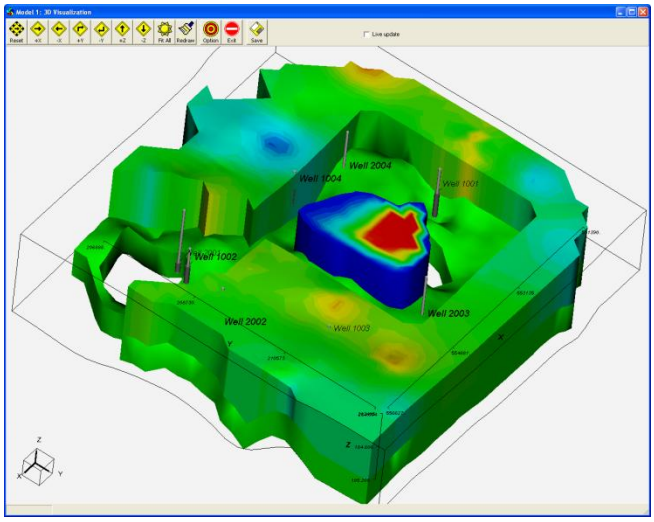
The user may also zoom in or out, by right-clicking the right side of the model to zoom in, or right-clicking the left side of the model to zoom out. Examples of simple model manipulation are shown in **Figure 21-25** (**A** through **C**).



A



B



C

Figure 21-25 3D Volume Grid View Manipulation Windows

21.2.4. Saving and Exiting

The user may save the model in their working screen by simply clicking the **Save** button in the button row along the top of the window, and then designating a pathway for the file.

The user may exit the window and return to the **Working Area** of **IGW** by either clicking the **Exit** button, or clicking the **X** in the top right corner of the window.

Chapter 22 GEOGRAPHICAL INFORMATION SYSTEM (GIS) INTERFACE

A GIS file (mainly shapefile,*.shp, here) is basically a digital vector file format which can store location and attribute information in it. IGW can use shapefiles for introducing groundwater modeling features into the modeling environment, just in a few minutes and interactively.

> **i** The IGW-GIS interface has mainly been developed for Michigan State flow and contamination data files by the Michigan Department of Environmental Quality (MDEQ), United States Geological Survey (USGS), and Michigan State University (MSU). IGW can ONLY recognize the specific data format for those files. In order to introduce GIS files for other regions, the user should match that specific format. Details will be given in later sections.

22.1. Opening the County-Based GIS Importer

Under the **GIS** tab in the **Menu Bar**, selecting **County-Based Assistant** brings up the GIS file interface (**Figure 22-1**) that opens the **County-based Assistant** window (shown in **Figure 22-2**). This feature has been added to aid in importing various shapefiles used specifically for Well Head Protection Area (WHPA) delineation on a county basis. The **County-Based Assistant** is a modification of the standard **GIS Model Importer** (see **Section 22.2**) in that it gathers (in a single step) all the pre-determined shapefiles within a single county, or collection of counties, that are typically used for delineating WHPA's. The user selects the County-Based Assistant from the **GIS** drop-down menu on the **IGW Modeling Menu Bar** as shown in **Figure 22-1**.

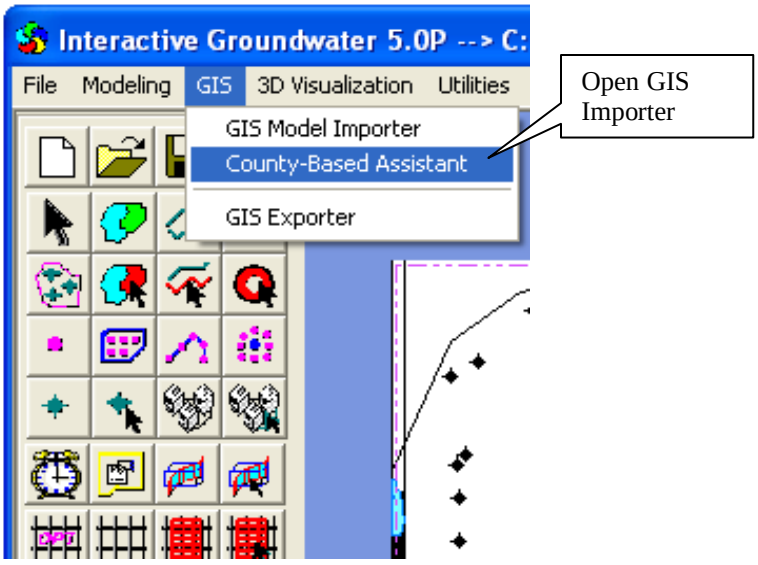


Figure 22-1 Opening the County-Based Assistant

Figure 22-2 contains a map showing the outlines of each county in the state. The user must first highlight the **county.shp** shapefile in the **GIS Layer Explorer** and select the **Selection Tool** icon**.** Using the left mouse button, the user may select either a single county or multiple counties. In order to select multiple counties, the user may depress and hold down the left mouse button and draw a selection rectangle as shown in **Figure 22-3**. It is also possible to select multiple counties by holding down the [**CTRL]** key and individually selecting multiple counties.

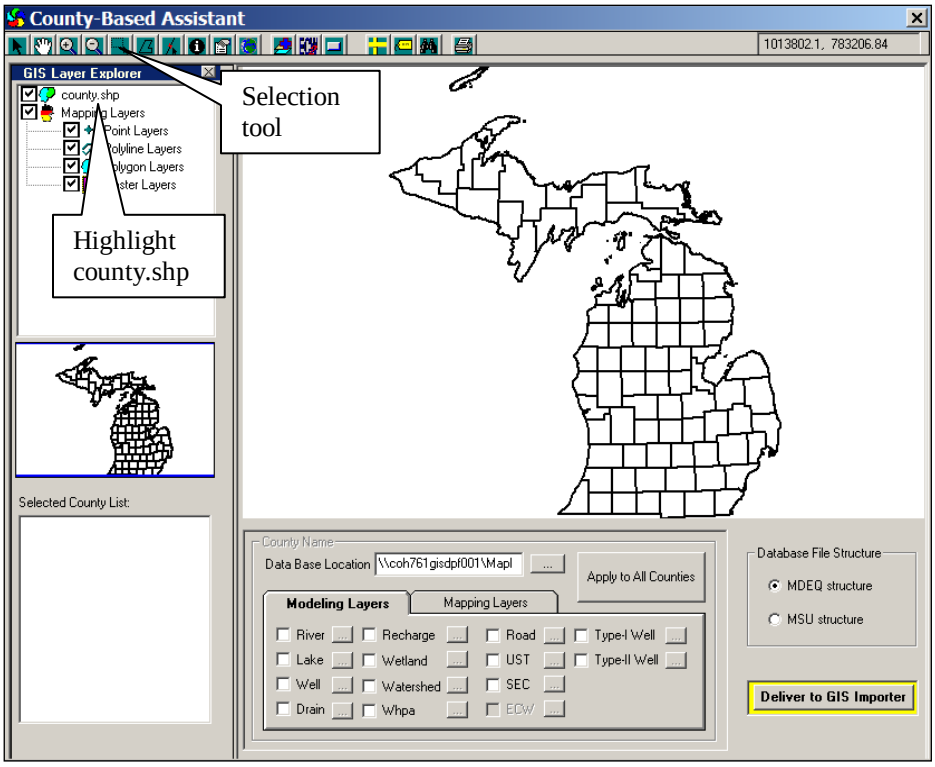


Figure 22-2 County-Based Assistant Window

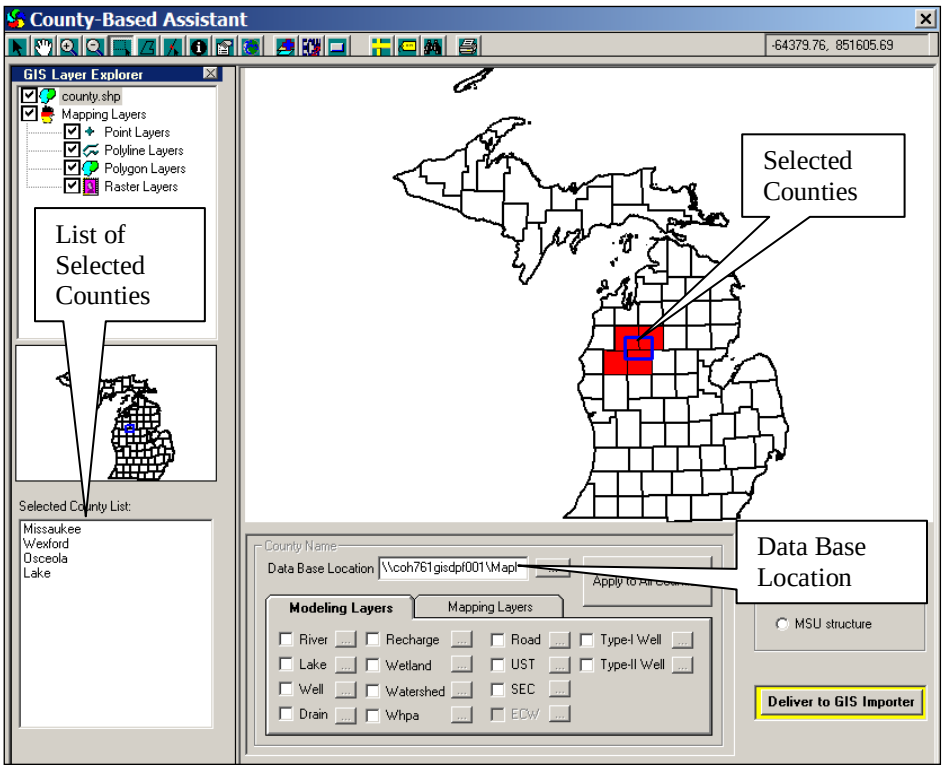


Figure 22-3 Selecting Counties

After selection, the name of each appears in the Selected County List in the lower left-hand side of the window. The user must define the location of the GIS database for each county. The location of the root database is defined by the user. It is necessary for the user to insert the specific folder

for each county as shown in **Figure 22-4**. The user must then define which shapefiles to import for that county, either as **Modeling Layers** or **Mapping Layers**.

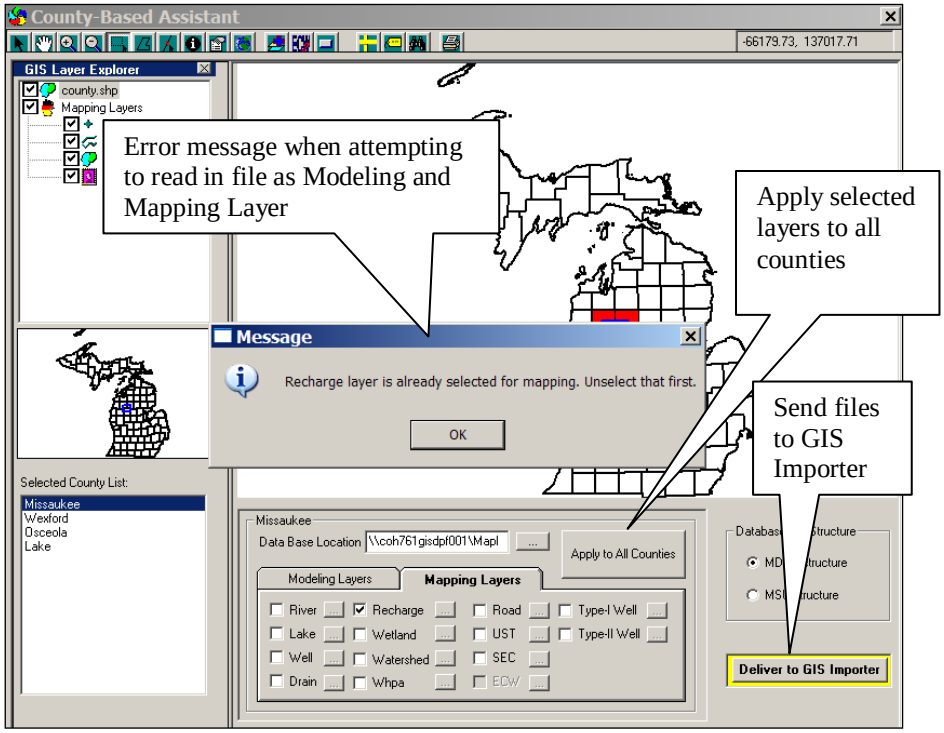


Figure 22-4 County-Based Assistant

A shapefile may be imported as either a modeling layer or a mapping layer, but not both. The message shown in **Figure 22-4** will be displayed should this occur.

It is possible to select the same shapefiles as **Modeling Layers** or **Mapping Layers** for each county by simply clicking on them and depressing **Apply to All Counties** button. It is also possible to select different shapefiles as **Modeling Layers** or **Mapping Layers** for each county, should that be necessary. This is done by simply selecting each county individually from the **Selected County List** and selecting a shapefile as a modeling layer or mapping layer.

All selected shapefiles may be sent to the GIS Model Importer by selecting the **Deliver to GIS Importer** button in the lower right-hand corner of the window shown in **Figure 22-4**.

22.2. Opening the GIS Importer

Return to the **IGW Modeling Environment** window by either closing the **County-Based Assistant** window, or moving the window to the side of the monitor screen under the **GIS Module.** Selecting **Import Model from GIS (Importer)** (**Figure 22-5**) opens the **GIS Model Importer** window (**Figure 22-6**).

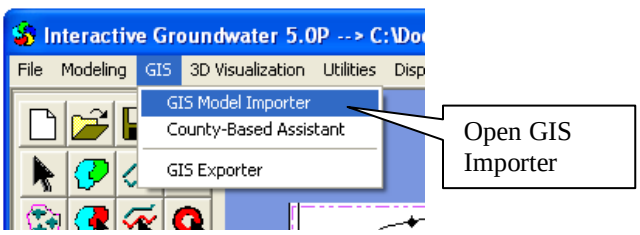


Figure 22-5 Opening GIS Model Importer

22.3. GIS Model Importer Environment

The **GIS Model Importer** environment is shown in **Figure 22-5**. There are three main areas in this window. The toolbar is at the top of the window, and the various buttons are used to import, visualize, and export attributes from the **shapefiles** to the **IGW Modeling Environment.** Most of the buttons on this toolbar are inactive until shapefile layers have been imported. The listing of the different shapefile layers on the left side of the **GIS Model Importer** window is referred to as the **GIS Layer Explorer**. The window on the right side of the **GIS Model Importer** window is referred to as the **GIS Viewing Window.**

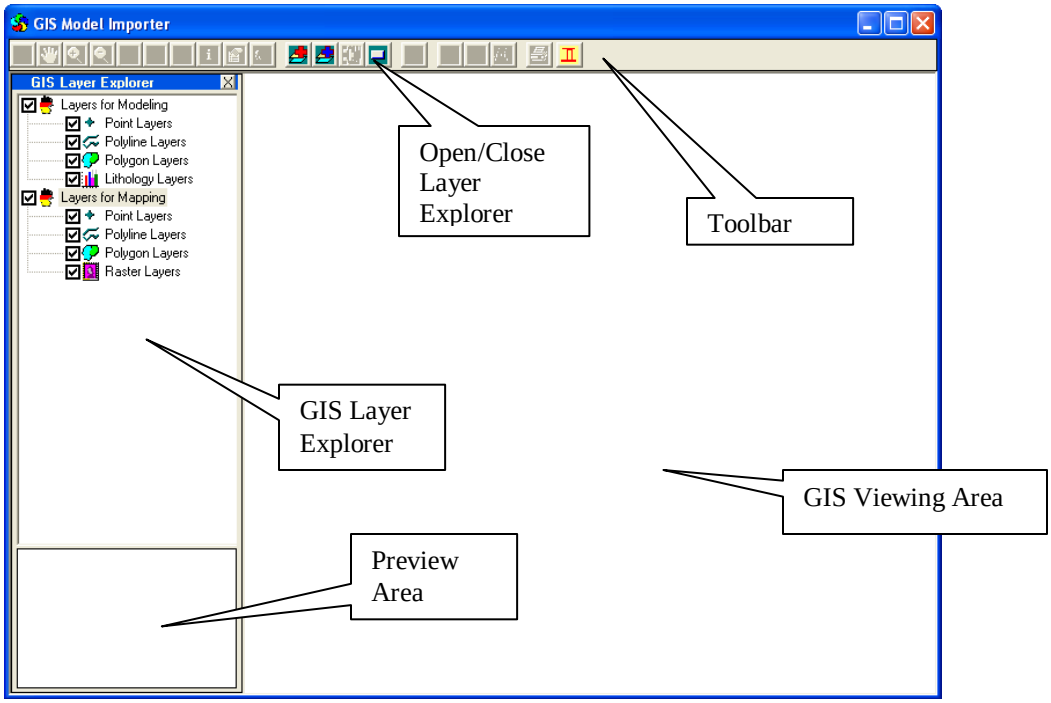


Figure 22-6 GIS Model Importer Environment

Within the **GIS Model Importer,** the user has the ability to import shapefiles that contain line-work that are used for display purposes only, or shapefiles that contain hydrogeological attributes to be used for groundwater modeling or WHPA delineation studies. The former are referred to as **GIS mapping layers**, the latter as **GIS model data layers**. In the **GIS Layer Explorer** these layers or shapefiles are grouped as **Layers for Modeling** or **Layers for Mapping**.

The Preview window at the bottom left of GIS Model Importer window displays the full extent of the imported GIS files. Within the viewing area, the user can use zoom in  , zoom out  and pan  buttons to navigate through the GIS data files. Please also see **Sections 22.12** and **22.13**.

The **GIS Layer Explorer** may be turned on or off by selecting the **Open/Close Layer Explorer** button (**Figure 22-6**).

22.4. Importing GIS Mapping Shapefiles

GIS mapping layers are imported by selecting the **Import Mapping Layer** button on the **GIS Model Importer** toolbar (**Figure 22-7**).

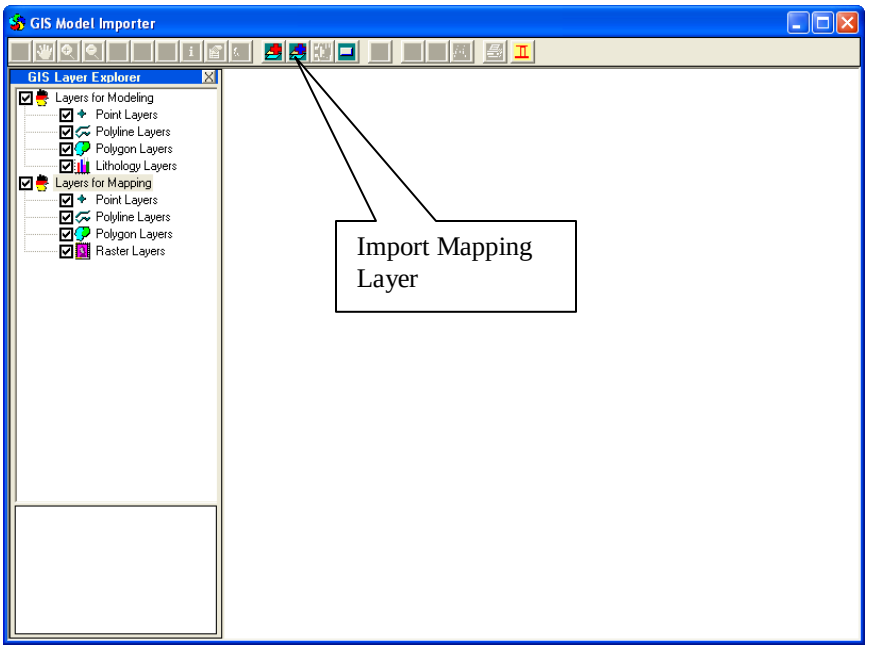


Figure 22-7 Importing the Mapping Layer

The GIS mapping layer shapefile to be imported is selected and opened in a second window. The imported shapefile appears as a graphic within the **GIS Importer Window**, and also in the appropriate position within the **GIS Layer Explorer**, shown on the left side of the **GIS Importer Window** as a **Point Layer**, **Polyline Layer**, or **Polygon Layer** under **Layers for Mapping** (**Figure 22-8**). It is not currently possible to import **Lithology** or **Raster Layers**. The example shown in **Figure 22-8** depicts the steps followed in importing road polylines as a mapping layer.

Any shapefile may be imported as a mapping layer. All data linked to these shapefiles are imported and may be viewed in the **GIS Model Importer** environment; however, only the graphic file (versus the data) from GIS mapping layers are exported to the **IGW Modeling Environment**.

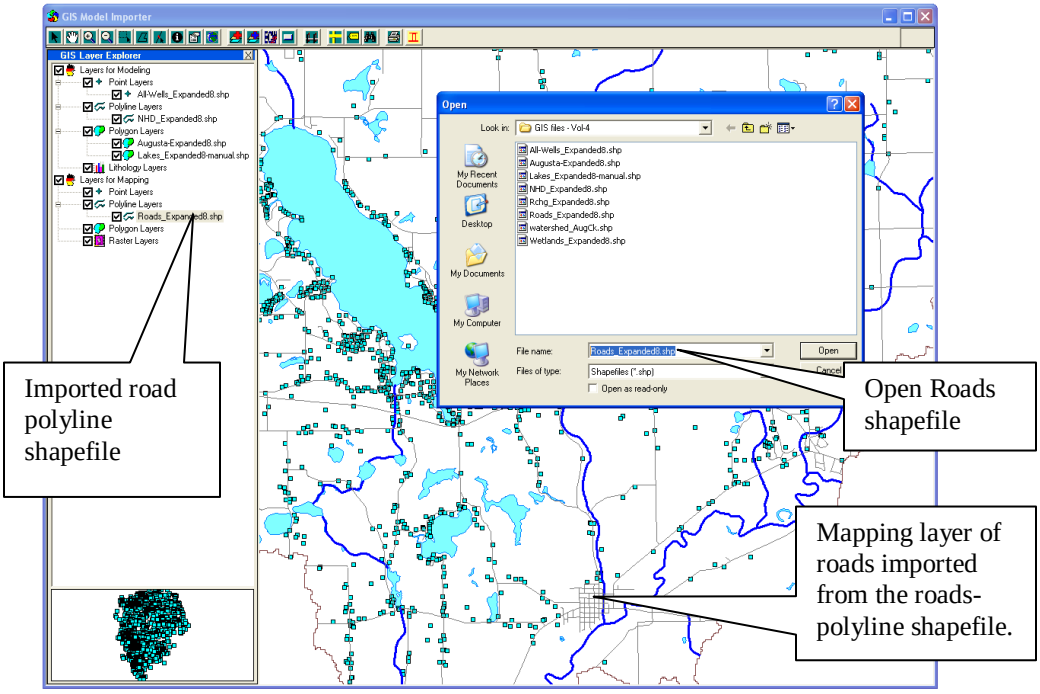


Figure 22-8 Opening GIS Shapefiles as Mapping Layers (Polylines)

22.5. Importing Model Data Shapefiles

In order to import the GIS model data layers, the **Import Model Data Layer** button on the **GIS Model Importer** toolbar (**Figure 22-9**) is selected.

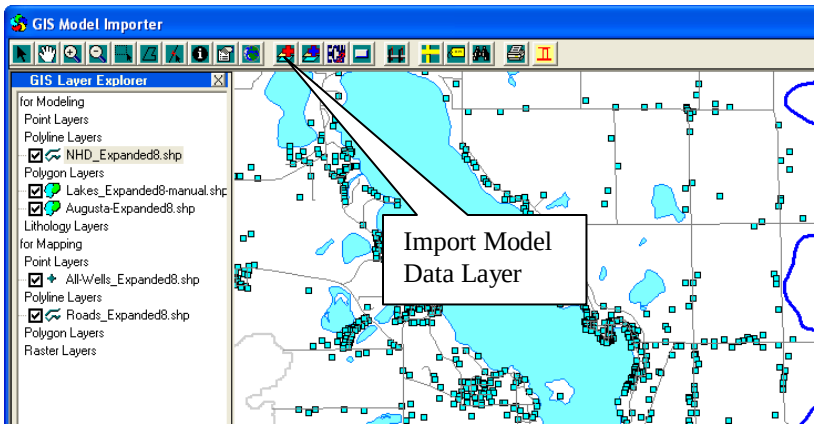


Figure 22-9 Importing Model Data Layer

The GIS model data layer shapefile to be imported is selected and opened in a second window using the left mouse (**Figure 22-10**). The imported shapefile appears within the **GIS Importer Window** and also appears automatically within the appropriate position in the **GIS Layer Explorer** shown on the left side of the **GIS Importer Window** as a **Point Layer**, **Polyline Layer**, **Polygon Layer**, or **Lithologic Layer** under **Layers for Modeling** (**Figure 22-10**). The example shown in **Figure 22-10** depicts the process of importing a point file containing wells as a model data layer.

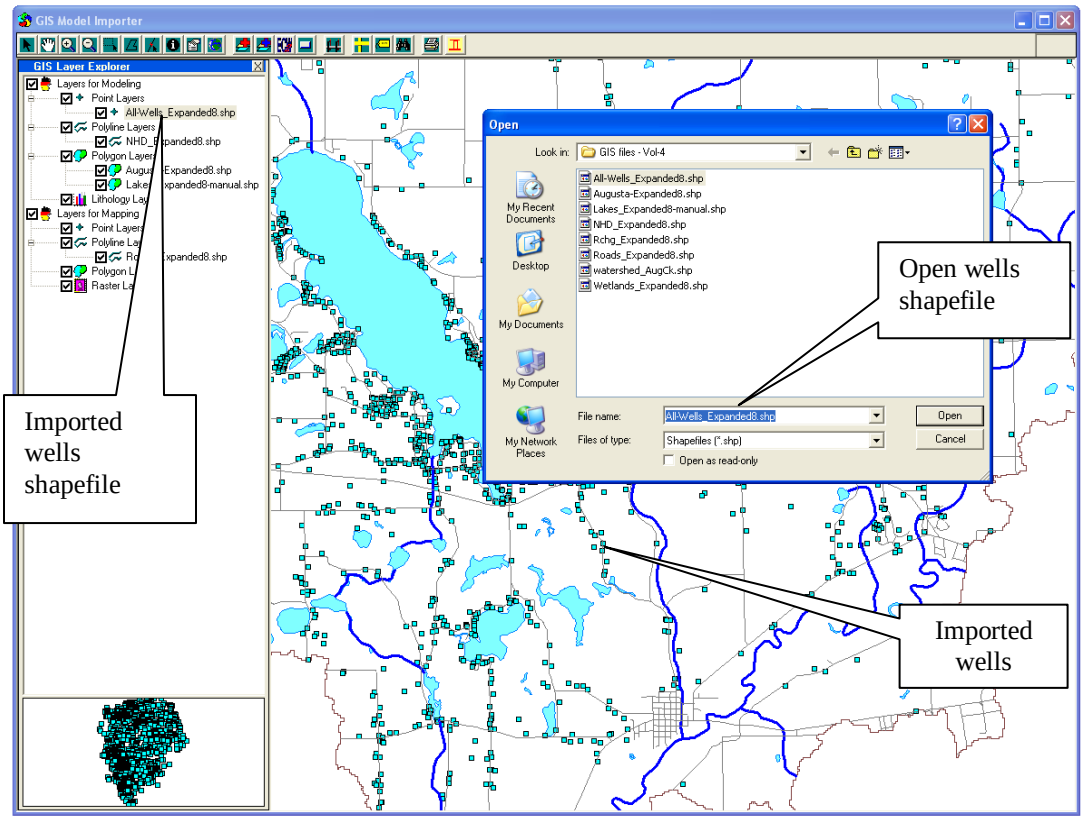


Figure 22-10 Opening GIS Shapefiles as Mapping Layers (Wells)

22.6. Viewing the Shapefile Data Table

The data attributes associated with any of the imported shapefiles may be examined by displaying the "data table" for that shapefile. This is done one of two ways. The first method is to select a shapefile in the **GIS Layer Explorer** and selecting the **Show Data Table** button on the **GIS Model Importer** toolbar (**Figure 22-11**). A data table for the entire selected shapefile is displayed (**Figure 22-12**).

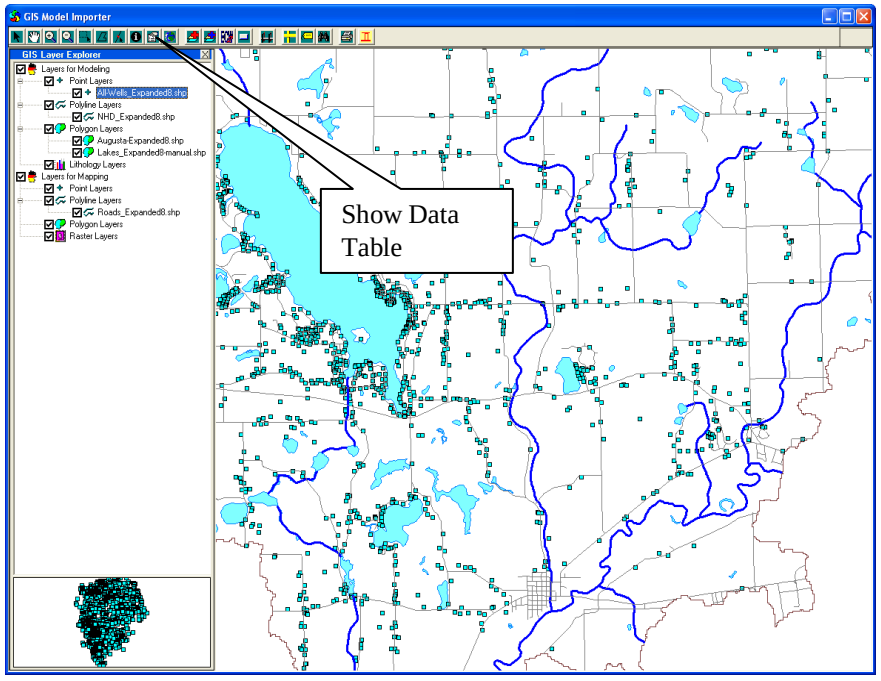


Figure 22-11 Opening the Data Table

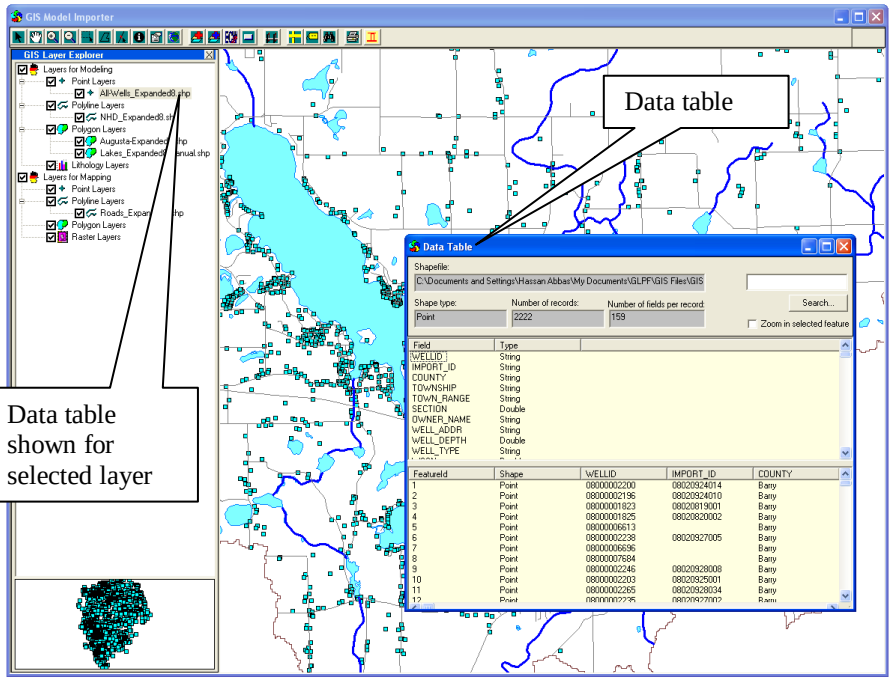


Figure 22-12 Opening the Data Table

The second method is accomplished by moving the cursor over the desired shapefile in the **GIS Layer Explorer** and right-clicking the mouse. The window shown in **Figure 22-13** will open. It is possible to examine the data table for the entire shapefile or just the portion of the shapefile that has been selected. Click the left mouse button on the desired selection option.

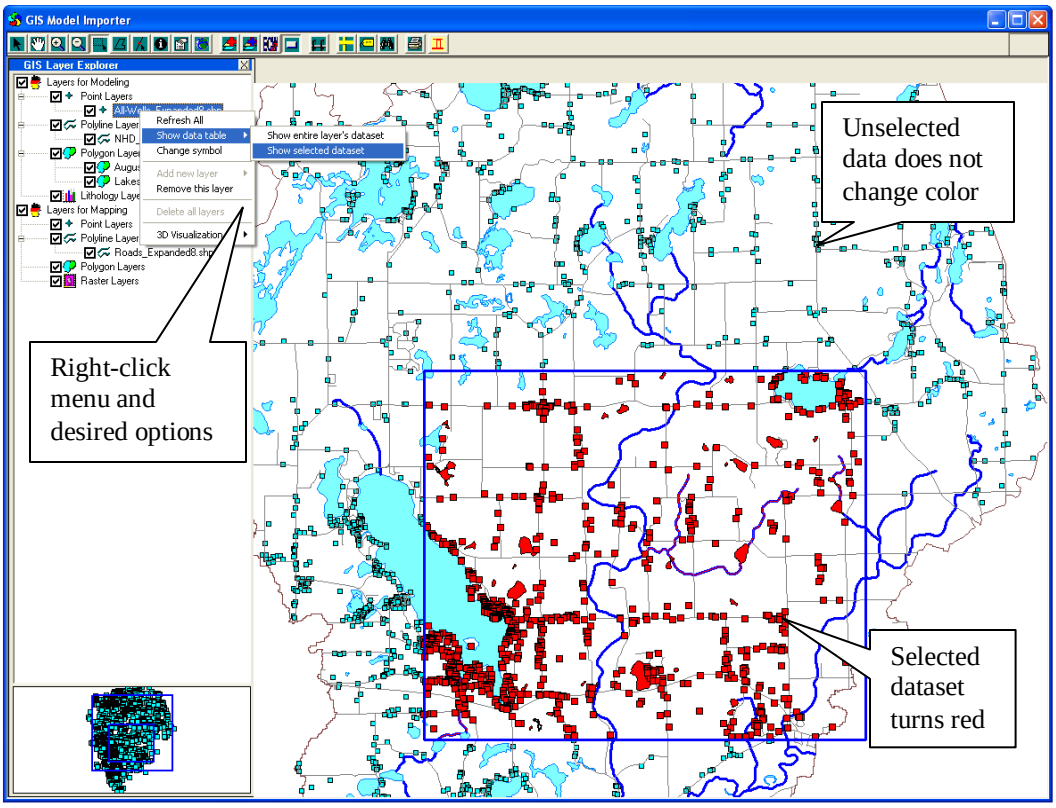


Figure 22-13 Showing the Data Table for the Entire or Selected Shapefile

> **!** It is not recommended that the data table for the entire shapefile be examined, as this table is typically quite large. A warning is displayed, informing the user as to the total number of lines in the data table.

Figure 22-14 shows an example of a partially displayed data table for a point shapefile containing wells. Please note that number of records in the data table in are only 829 for the selected wells. However, the total number of wells in this example file are 2222 as can be seen in the data table shown in **Figure 22-12**.

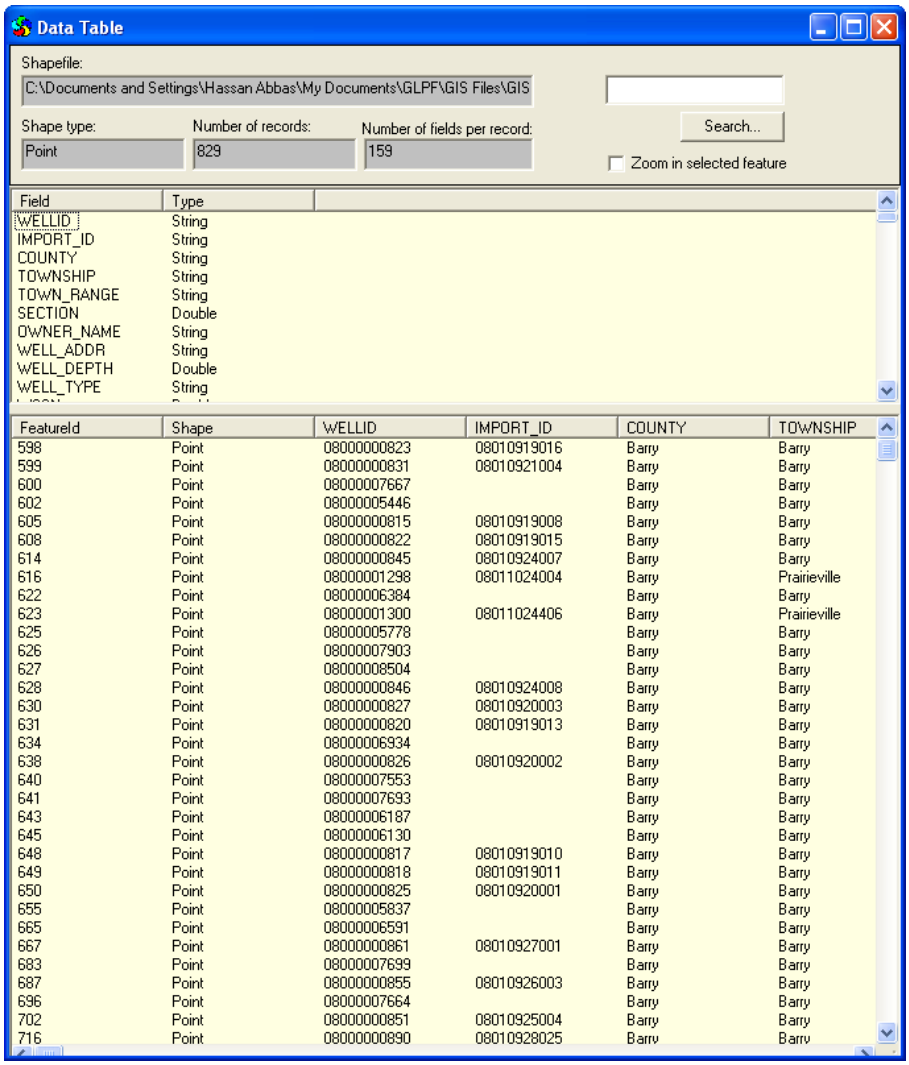


Figure 22-14 Shapefile Data Table

The pathname showing the location of the shapefile, the shapefile type (e.g. point, polyline, etc.), number of records and number of fields are displayed near the top of the **Data Table** window. The different field identifiers and field types are shown in a scroll-down window in the middle of the **Data Table** window, and all of the fields in the selected data table are shown as a horizontal-scroll window at the bottom of the data table window. It is also possible to search for a data table element entry using the "Search" window at the top of the **Data Table** window.

22.7. Viewing Attributes of a Single Point, Polyline, or Polygon

It is possible to select and display the attributes for a single point, polyline, or polygon, rather than for the entire shapefile, regardless of whether the shapefile is a model data layer or mapping layer. This is done by:

1) Selecting the desired shapefile in the **GIS Layer Explorer**,
2) Selecting the **Identify** button on the **GIS Model Importer** toolbar, and
3) Selecting the desired single point, polyline, or polygon in **GIS Viewing Window**.

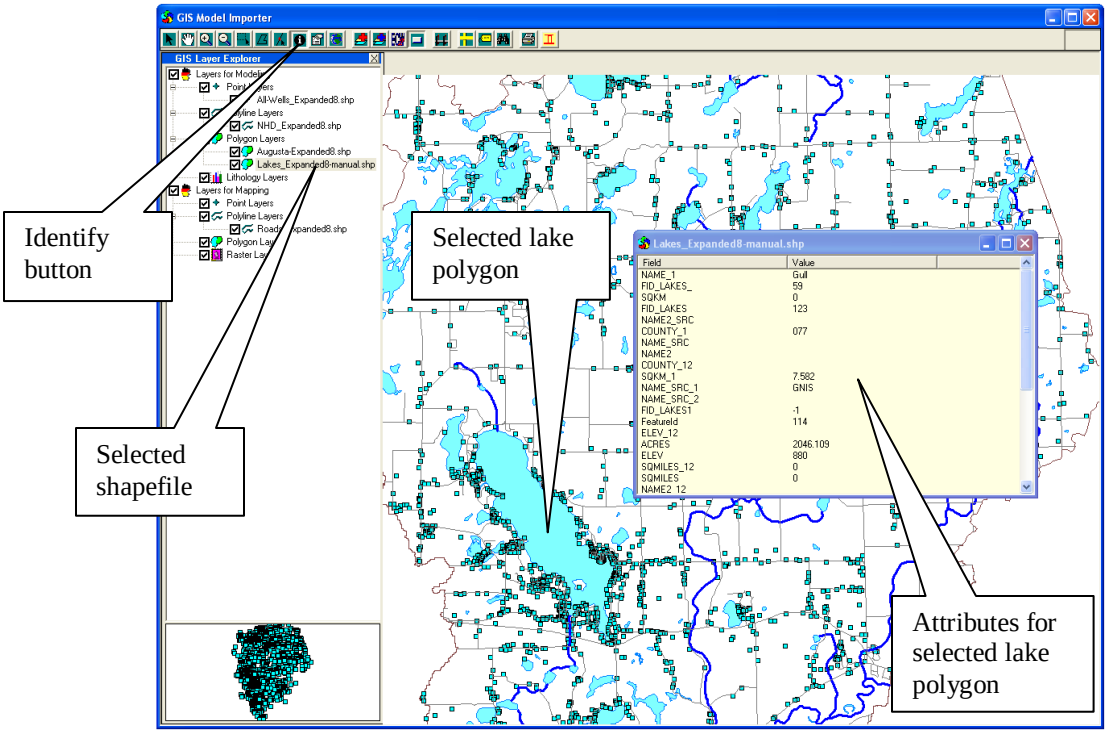


Figure 22-15 Selecting and Viewing Single Polygon Attributes

The example shown in **Figure 22-15** depicts the selection and viewing of the attributes for a single lake polygon. The same can be accomplished for point and polyline shapefiles.

22.8. Changing the Appearance of Shapefiles

When shapefiles are imported using the **GIS Model Importer**, their appearance is determined by default settings within the program. All point shapefiles are imported as solid squares, polylines as solid lines, and polygon shapefiles as solid polygons. Both points and polygons have a default black outline. The default colors for each shapefile have been pre-determined.

It is possible for the user to change the color and appearance for all shapefiles. This is done by positioning the cursor over the shapefile name in the **GIS Layer Explorer** on the left side of the **GIS Model Importer** window, and right-clicking the mouse. A window opens as shown (**Figure 22-16**). Scroll down and select the **Change symbol** option.

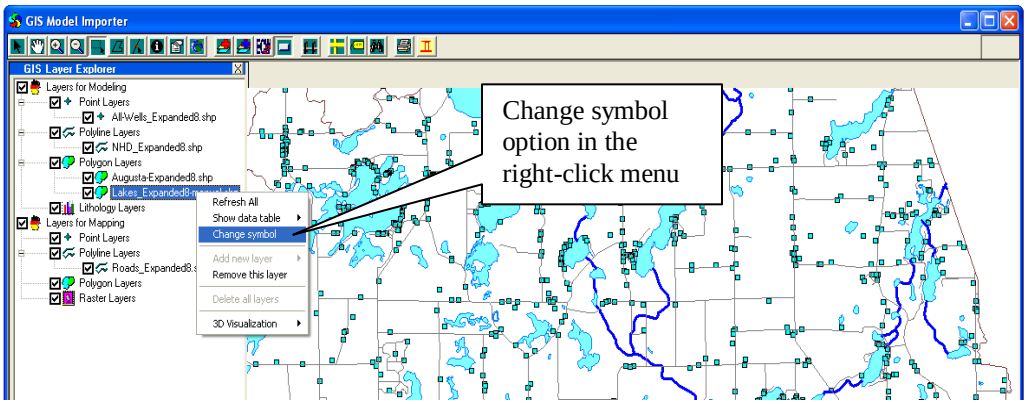


Figure 22-16 Changing the Shapefile Display

After selecting the **Change symbol** option, the following window opens with all shapefile graphic options (**Figure 22-17**).

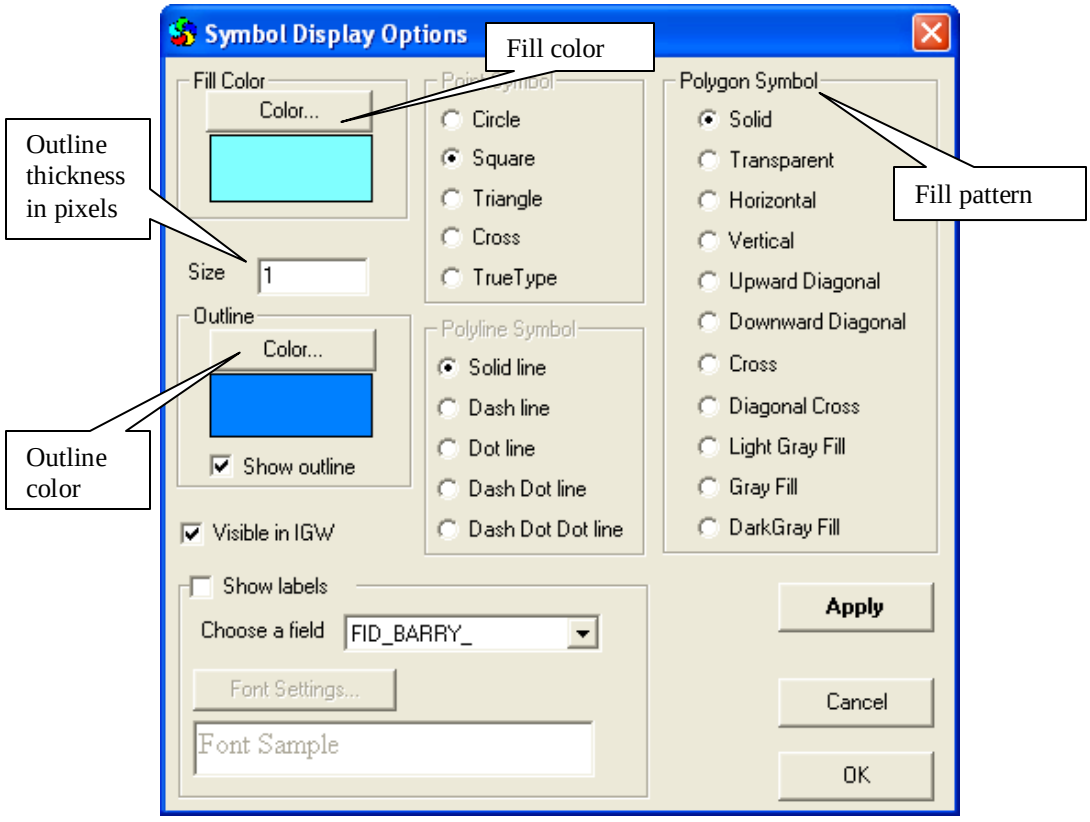


Figure 22-17 Changing the Symbol Display (Polygons)

The selected shapefile in this example are lake polygons. In this case, only the choices under **Fill Color**, **Size**, **Outline**, and **Polygon Symbol** in the **Change symbol** window are active. Those associated with point and polyline shapefiles (**Point Symbol** and **Polyline Symbol**) are inactive, and cannot be selected or changed.

With polygons, it is possible to change the outline color and weight, fill color, and fill pattern. It should be noted that all fill patterns (other than **Solid**) are somewhat transparent. The **Transparent** option makes the view completely transparent and does not use the fill color shown in the **Fill Color** box. Options **Horizontal** through **Diagonal Cross** indicates the orientation of lines comprising the fill pattern. The color of the lines is defined by the color shown in the **Fill Color** box. The space between lines is completely transparent, allowing any shapefiles beneath these polygons to be visible. **Light Gray Fill**, **Gray Fill,** and **Dark Gray Fill** approximately represent 75%, 50%, and 25% transparency, respectively. The colors for these variable transparency fill patterns are also defined by the color shown in the **Fill Color** box.

The following two figures (**Figure 22-18** and **Figure 22-19**) show the **Change Symbols** windows for polyline and point shapefile examples.

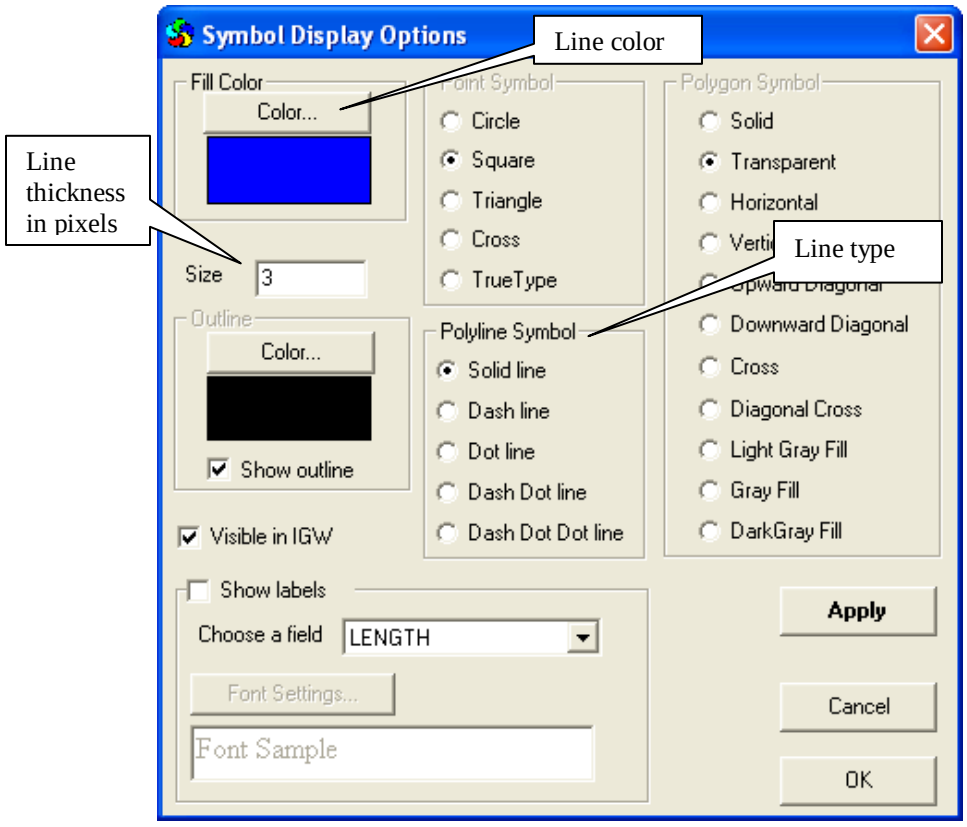


Figure 22-18 Changing Symbol Display (Polylines)

For the polyline example (**Figure 22-18**), only the line type under **Polyline Symbol,** the line weight (**Size**), and the **Fill Color** (to define line color) are active. It is not possible to use the color box in the **Outline** feature, as this option is inactive. The line weight is defined in the **Size** box under the **Fill Color** selection box.

For the point symbol example (**Figure 22-19**), only the symbol type, the symbol size (**Size**), and the **Fill Color** (to define fill color) are active. It is not possible to use the color box in the **Outline** feature as with the Polygon example (**Figure 22-17**). The default outline color is black for all symbol types. The size is defined in the **Size** box under the **Fill Color** selection box.

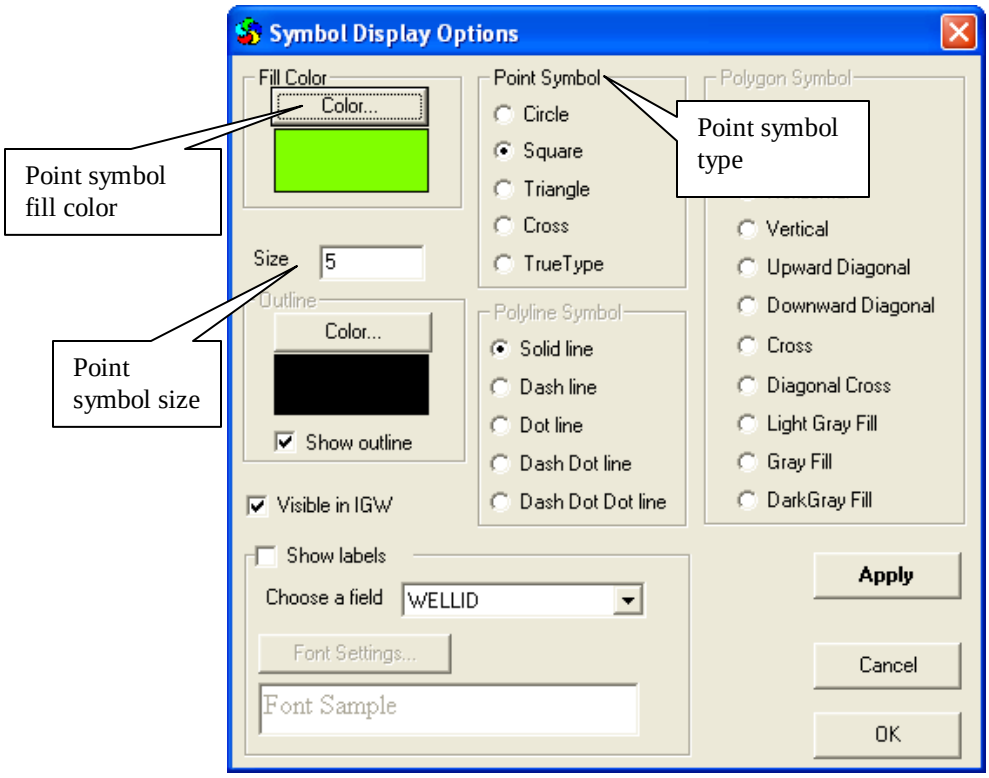


Figure 22-19 Changing Symbol Display (Wells)

22.9. Adding a Single GIS Layer

Sections 22.4 and **22.5** described the process for importing GIS mapping or model data layers using the appropriate buttons on the **GIS Model Importer** toolbar. It is also possible to add layers by positioning the cursor on either **Layers for Modeling**, or **Layers for Mapping** in the **GIS Layer Explorer** window and right-clicking the mouse (**Figure 22-20** and **Figure 22-21**). This feature functions exactly as the two "add layer" buttons on the **GIS Model Importer** toolbar.

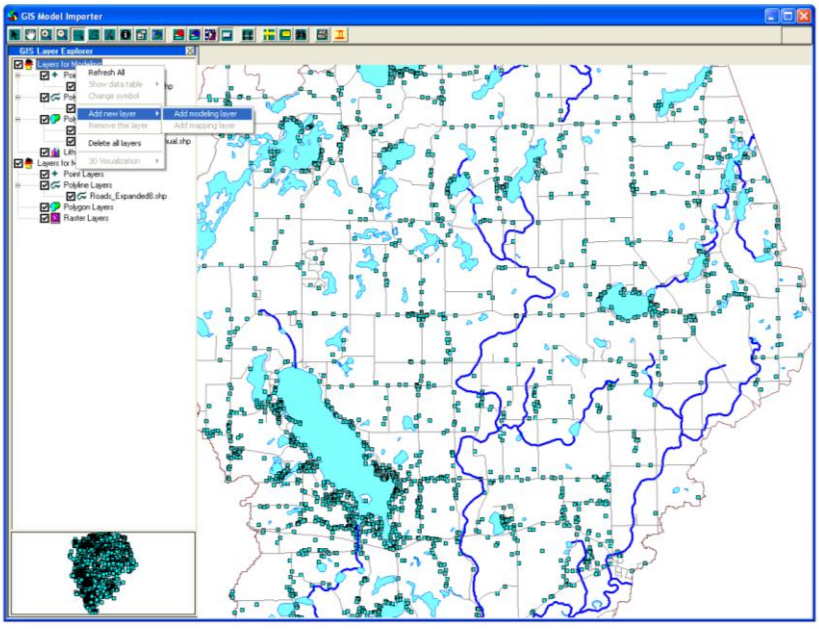


Figure 22-20 Adding New Modeling Layers

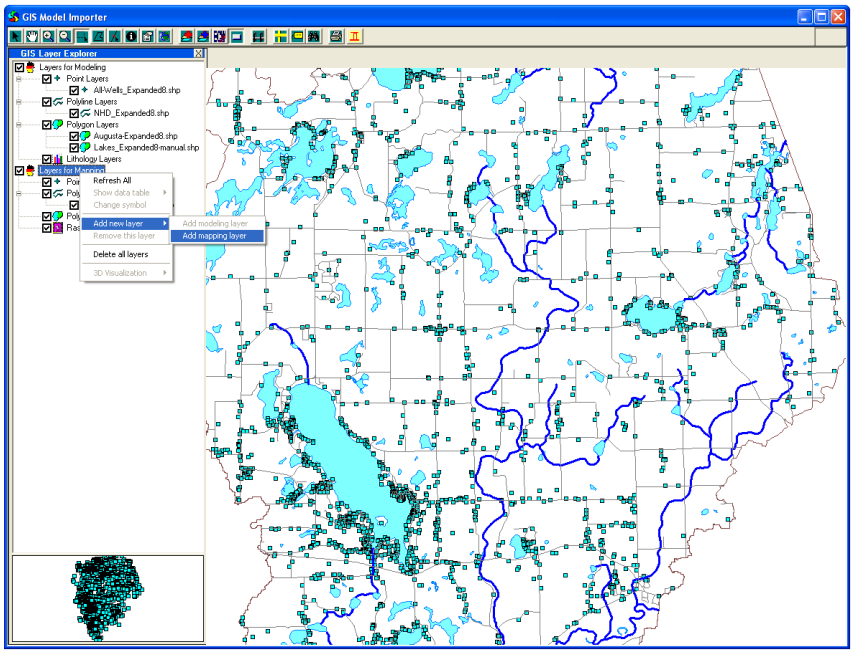


Figure 22-21 Adding New Mapping Layers

22.10. Removing a Single GIS Layer

It is possible to remove a single model data or mapping shapefile layer, if necessary. This is done by positioning the cursor over the shapefile in the **GIS Layer Explorer** and right-clicking the mouse. Scroll down to highlight the **Remove this layer** option (**Figure 22-22**), and left-clicking the mouse to complete the layer removal.

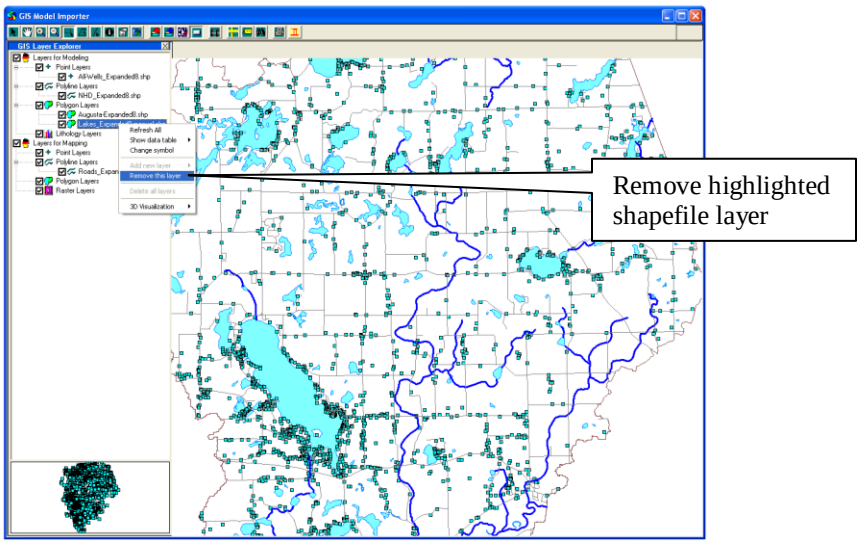


Figure 22-22 Removing a Single Layer in the GIS Model Importer

22.11. Deleting All GIS Layers

It may also be desirable to delete all shapefiles that have been imported. This is done by positioning the cursor over any shapefile category (e.g. Layers for Modeling, Point Layers, etc.) in the **GIS Layer Explorer** and right-clicking the mouse. Scroll down to highlight the **Delete all layers** option (**Figure 22-23**). Left-clicking the mouse completes the layer removal. All

shapefiles listed in the **GIS Layer Explorer** are removed, whether or not the selection box to the left of each shapefile is checked.

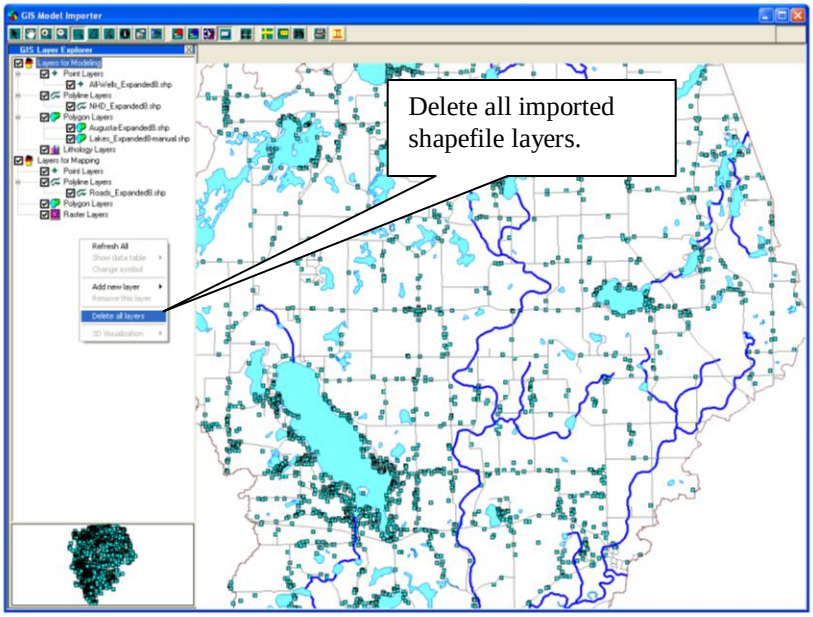


Figure 22-23 Removing all GIS Layers in the GIS Model Importer

22.12. Moving the Map Within the GIS Window

The entire map that is displayed in the **GIS Viewing Window** may be moved around the window. This is done by selecting the **Pan Map** button  on the **GIS Model Importer** toolbar (**Figure 22-24**). The "hand" cursor is placed on the map, and the left mouse button is depressed and held down as the cursor and map are moved about the window.

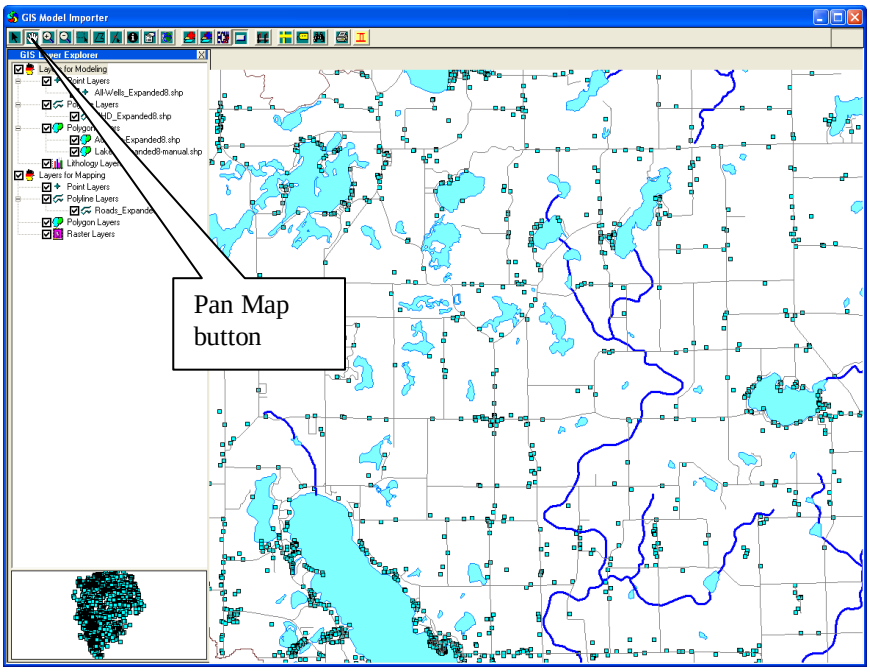


Figure 22-24 Moving the Map by Using the Pan Map Button

22.13. Changing the Viewing Area Scale in the GIS Model Importer

When shapefiles are first imported into the viewing area of the **GIS Model Importer** window, the default setting is to expand the viewing area to include the entire extent of the shapefile. As shapefiles are added, the scale of the viewing area remains unchanged and must be adjusted so that the entire extent of all shapefiles is visible in the **GIS Model Importer** window.

There are three options for adjusting the viewing area scale: **Zoom In**  , **Zoom Out**  **(incremental),** and **Zoom Out to Full Extent**. Care should be used when selecting the **Zoom Out to Full Extent** button, as the map will expand to the full extent of the largest shapefile coverage. If any of the shapefiles have "state-wide" extent, the map will expand showing the entire extent of the state. **Figure 22-25** shows the location of each option button on the **GIS Model Importer** tool bar.

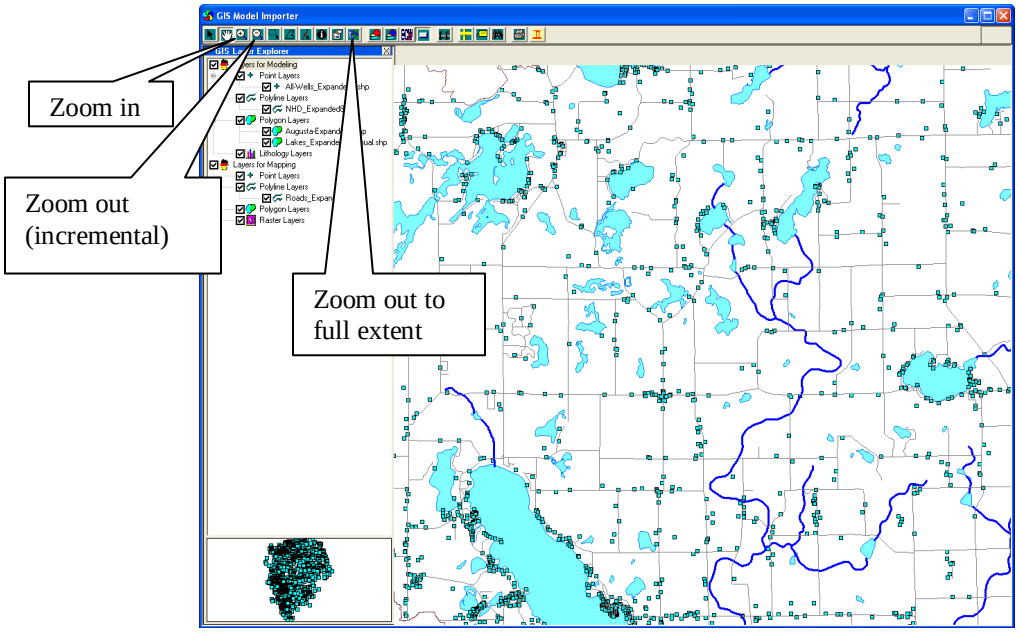


Figure 22-25 Options to Change the Viewing Area Scale

To zoom in, select the **Zoom In** button. Holding down the left mouse button, draw a rectangle around the area to be enlarged. Once a user has "zoomed-in" on a sub-area of the full extent of the shapefiles, there are two methods for "zooming-out". The first is to select the **Zoom Out** button and depressing the left mouse in the center of the viewing area. The viewing area scale is increased by a factor of 1.5 every time the left mouse button is depressed, until the original full scale has been recovered. The second method of "zooming-out" is to select the **Zoom Out to Full Extent** button. The original viewing scale is recovered in a single step.

22.14. Using the Measuring Tool

It is possible to measure the linear distance between two points on the map displayed in the **GIS Model Importer** window, by using the **Distance Measuring Tool** that is found on the **GIS Model Importer Toolbar** (**Figure 22-26**). The user should select the starting point by left-clicking the mouse, release it and move the cursor to the end point. A window opens displaying the distance (**Figure 22-26**).

i	Presently, the distance is displayed in meters only.

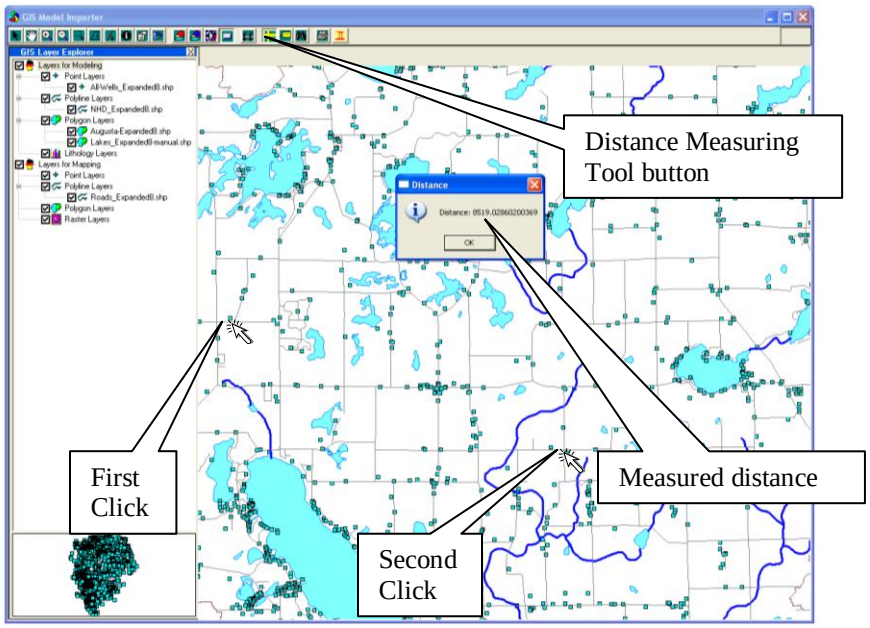


Figure 22-26 Measuring Distance

22.15. Selecting Shapefile Data for Exporting Into IGW

After all model data layers and mapping layers have been imported into the GIS model environment, any of these data may be exported to the **IGW Modeling** Environment. Prior to selecting data to be exported, the data box to the left of each shapefile in the GIS Layer Explorer must be "checked" to identify the appropriate model shapefile data to export to IGW. Only shapefiles that are "checked" will be exported to the **IGW Modeling Environment**.

Select the **Data Selector** button on the **GIS Model Importer** toolbar (**Figure 22-27**). Holding down the left mouse button, draw a rectangle around the area to be exported to the **IGW Modeling Environment**.

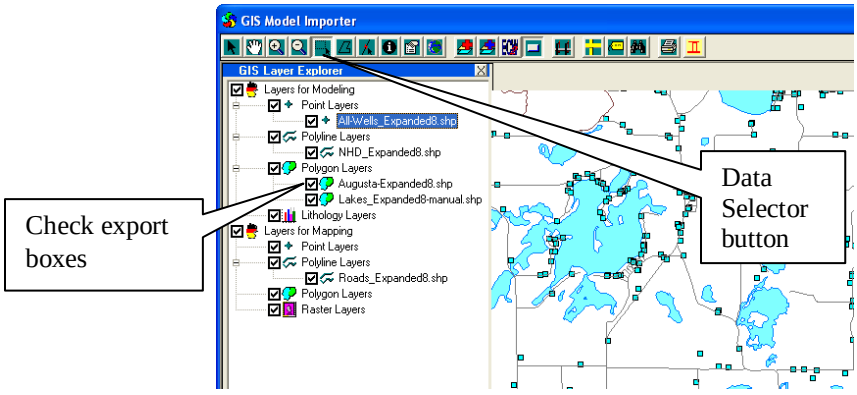


Figure 22-27 Rectangular Data Selector Function

All checked model layer shapefile data that fall within the selection area will be highlighted (**Figure 22-27**). Model layer data do not have to fall entirely within the selection box to be exported. In **Figure 22-28** there are several point shapefiles that are bisected by the selection box. Any model data layer shapefile (point, polyline, or polygon) that is intersected by the selection box will be exported. All mapping layer shapefiles that fall within the selection box are not highlighted; however, the shapefile line work is exported to the **IGW Modeling Environment**.

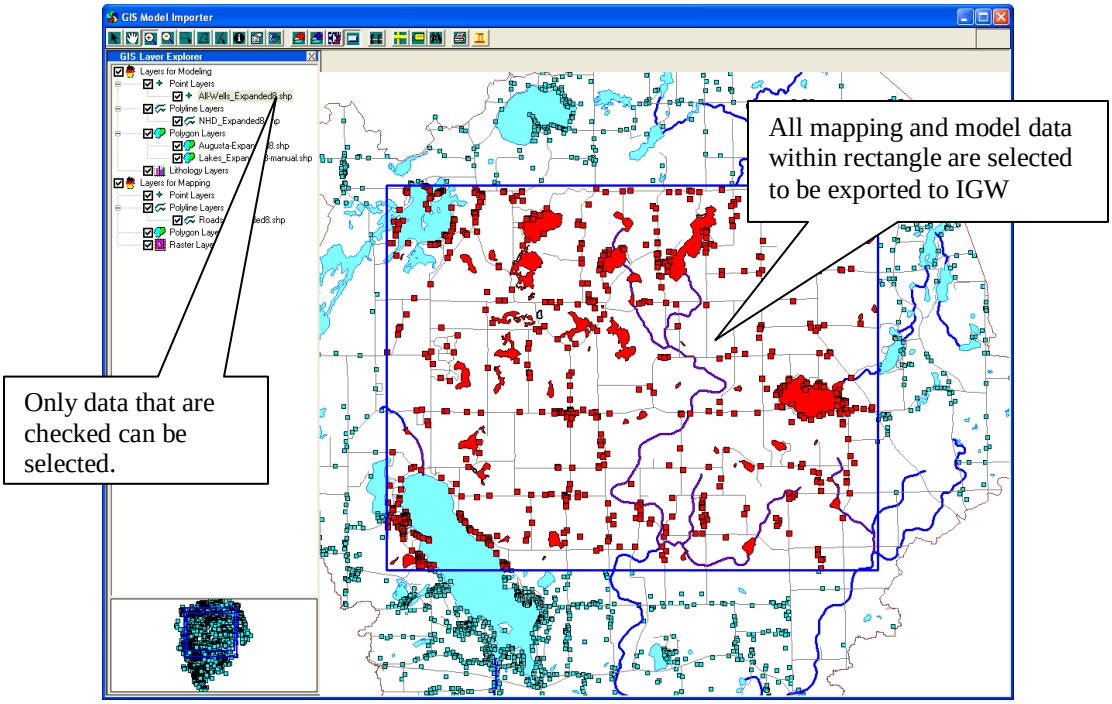


Figure 22-28 Selecting Data to be Exported to IGW

It is also possible to use the watershed boundary (instead of a rectangle) to select data for export into the **IGW Modeling Environment**. To do this, the watershed shapefile must have been imported into the **GIS Model Importer** environment as a **modeling layer**. Select the watershed shapefile, then select the **Data Selector** button. Move the cursor over the desired watershed and select the watershed. All model and mapping layers within the watershed boundary will be selected (**Figure 22-29**).

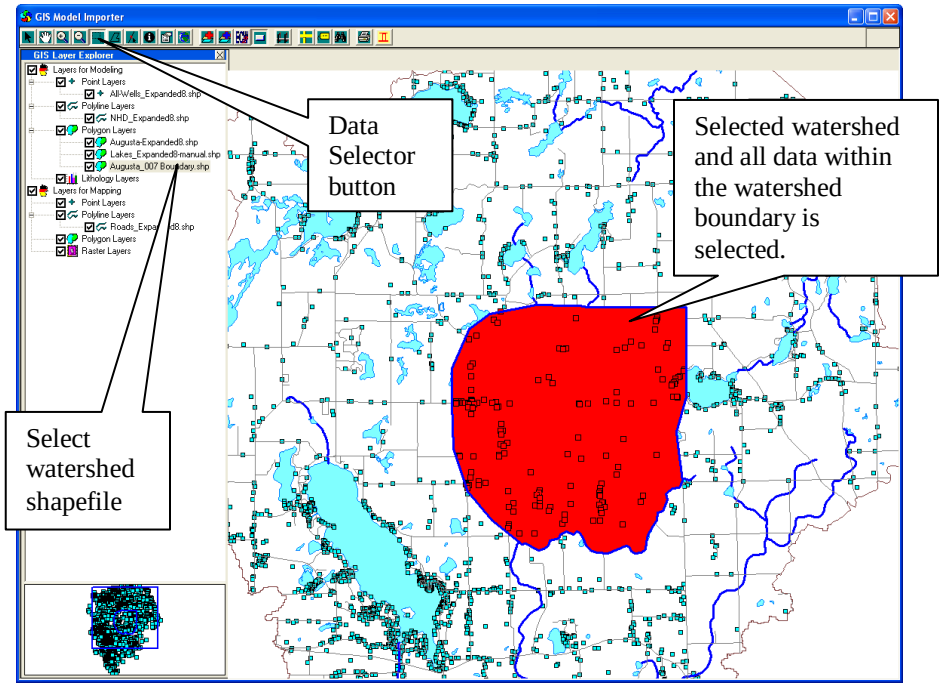


Figure 22-29 Selecting Data to be Exported to IGW Using Watershed Boundary

22.16. Extracting to IGW

Using the left mouse button, the user should select the **Data Exporter** button on the **GIS Model Importer** toolbar (**Figure 22-30**).

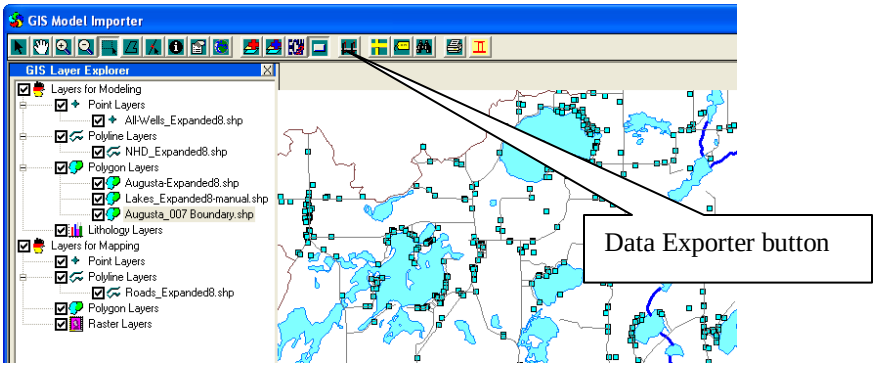


Figure 22-30 Extracting Data to IGW

22.17. Extraction Criteria

After the **Data Exporter** button has been selected, a window showing the extraction criteria for all selected model data shapefiles will open. The **Extraction Criteria** window appears in **Figure 22-31**.

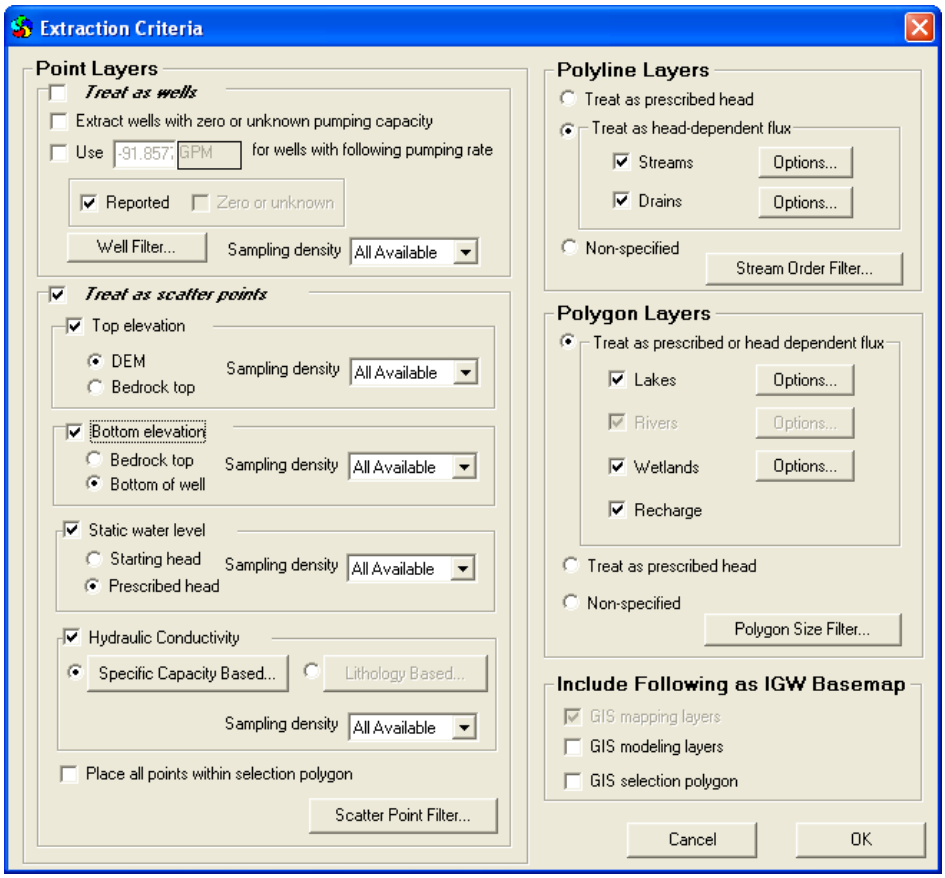


Figure 22-31 Extraction Criteria Window

The purpose of the options in the **Extraction Criteria** window is to provide a final processing or filtering of the raw data found in each model data shapefile.

22.17.1. Pumping Wells

The example shown in **Figure 22-32** is for exporting a point shapefile as a pumping well. The option to **Treat as wells** must be selected.

If the user chooses not to assign a uniform pumping rate, the pumping rate that is found in the well point shapefile will be used.

> **i** By default, any wells without an assigned pumping rate are not exported to IGW.

To over-ride the default settings, the user must select the "**Extract wells with zero or unknown pumping capacity**" option. By selecting this option, all wells found in the selected shapefile will be imported. Those with zero or unknown pumping rates will be assigned a pumping rate equal to zero.

The user also has the option of assigning a uniform pumping rate to all wells, or simply using the estimated pumping rate that is found in the model data shapefile. Assigning a uniform pumping rate will override any pumping rate assigned to that well in the point shapefile. The option for assigning a uniform pumping rate is selected using the left mouse button on the **Use ___ for wells with following pumping rate** selection box. The uniform pumping rate is entered, along with the appropriate units. One of the two boxes, **Reported** or **Zero or unknown**, must be selected if the uniform pumping rate is to override the reported pumping rate, or to be entered where the reported rate is zero, unknown, or both conditions. Note that the **Zero or unknown** option is active only after the **Extract wells with zero or unknown pumping capacity** option shown below is selected.

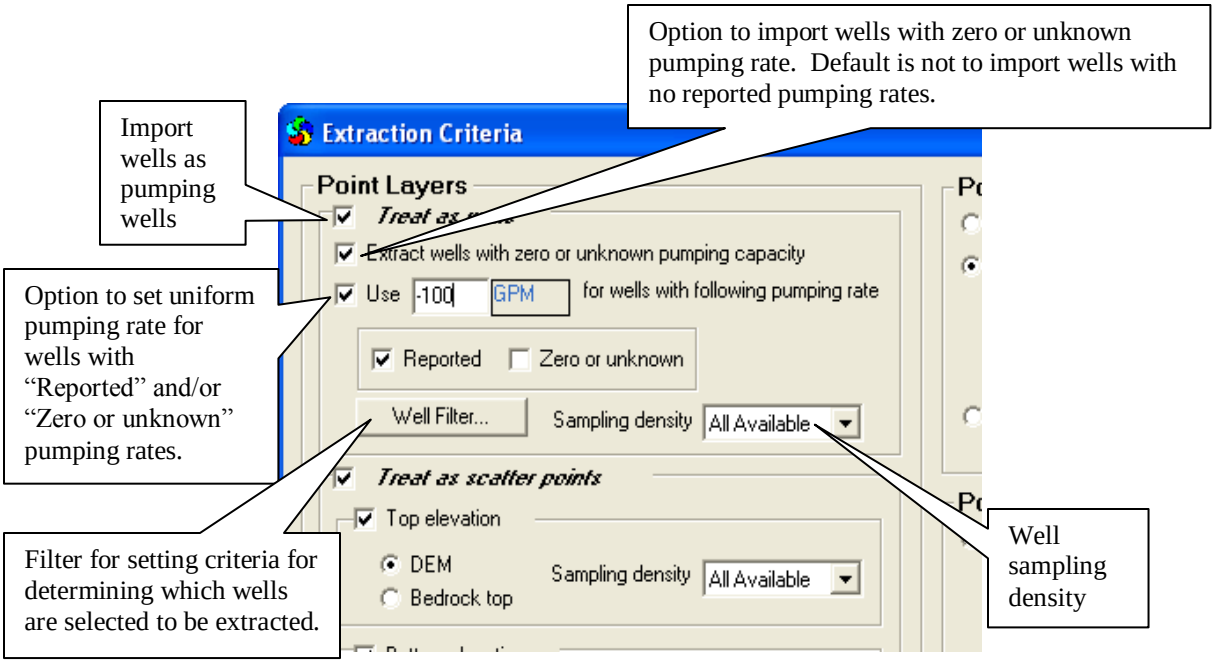


Figure 22-32 Extracting Point Data as Pumping Wells

In certain areas, there may be wells that extract groundwater from either the glacial drift aquifers (**Drift**) or the underlying bedrock aquifer (**Rock**). The **Well Filter** option button is shown in **Figure 22-32** and is selected using the left mouse button. The **Well Filter** window is shown in **Figure 22-33**. The **Aquifer Code** selection options, shown in the upper left part of the window,

allows the user to select wells that are open to either the **Drift** or **Rock** (if present) aquifer, or both. In some cases, the aquifer was designated **Unknown** in the shapefile.

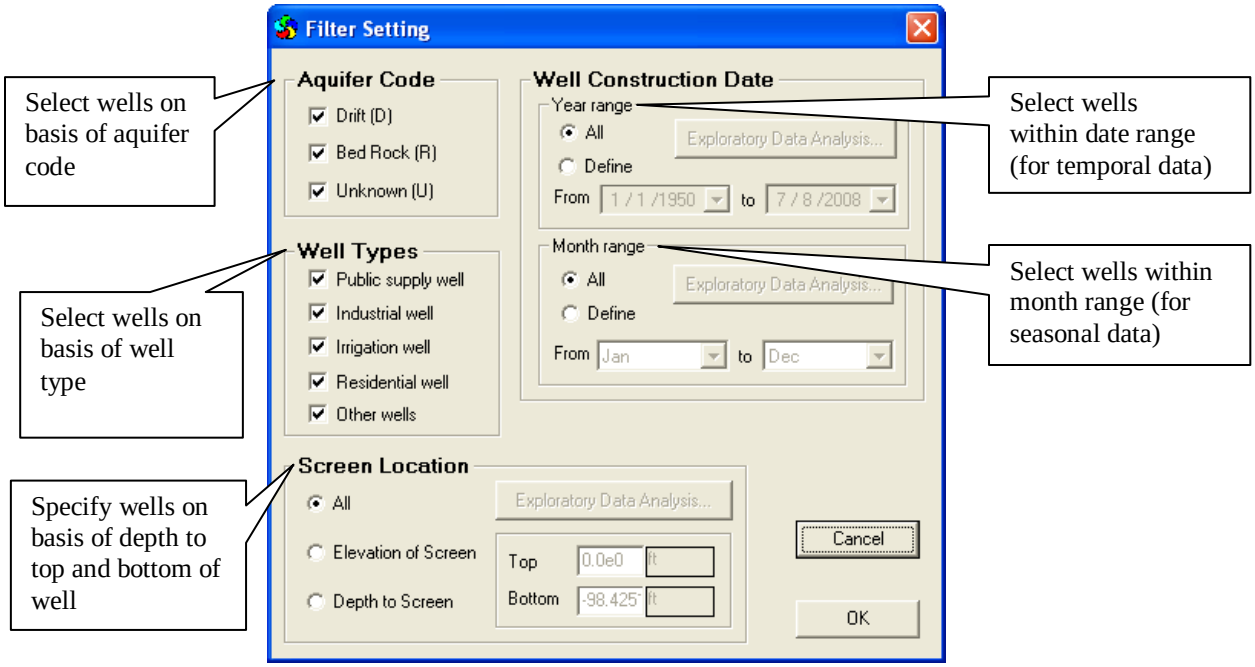


Figure 22-33 Well Filter

Each well has a designated use that falls into one of five categories: **Public supply**, **Industrial**, **Irrigation**, **Residential**, and **Other**. The user may choose wells on the basis of their intended use by selecting any or all of the desired well categories.

It is also possible to refine the well selection on the basis of well depth. This is shown as the **Screen Location** filter. The user has the option of selecting all wells, or entering the **Elevation of Screen Top** and **Depth to Screen Bottom** of the open interval of the well. This could be the depth to the top and bottom of the well screen for a **Drift** well, or the depth to the bottom of well casing (**Top**) and the bottom of the borehole (**Bottom**) for a **Rock** well. The user may select any of the available units by selecting the second box to the right of **Top** or **Bottom**.

The user may further refine the well selection by specifying a time interval from which to select wells on the basis of well construction date. This may be done by date (**Define year range**), by season (**Define month range**), or a combination of the two. The user may also choose not to select either time filter. In this case, all wells meeting the other selection criteria will be selected.

22.17.2. Scatter Point Data

Figure 22-34 is for exporting a 'point' shapefile as scatter points (hydrogeologic point data). The option to **Treat as scatter points** must be selected using the left mouse button.

The user is able to select the aquifer **Top elevation** based on land surface elevation (**DEM**) or bedrock top elevation (**Bedrock top**), aquifer **Bottom elevation** based on bedrock top elevation (**Bedrock top**), or **Bottom of well**, static water level elevation (**Static water level**), **Hydraulic Conductivity** estimated from the **WELLOGIC** data (**Specific Capacity Based**), or from the **GWIM** project (**Lithology Based**) as scatter points to be exported into IGW. The option box must be selected for each data type using the left mouse button in order to export these data. Data type not selected will not be exported.

With all scatter point data, it is possible to specify the sampling density of data that will be extracted. For many areas, it is advisable to select **All Available** data points. However, in areas where there are large numbers of data points, it may not be feasible to extract all data points. In these areas, it would be advisable to sample fewer data. Other than exporting all data, it is possible to select one out of every two, three, four, five, six, ten, fifteen, twenty, fifty, and/or a hundred data points.

The **Static water level** data has two export options. These relate to the manner in which these data will be used within the **IGW Modeling Environment**. If **Starting head** is selected, the data represent the "initial guess" for hydraulic head for the model. These values are not stored and will be changed during subsequent model simulations. If **Prescribed head** is selected, the data are imported as a "prescribed head" surface for the model. These values are not changed during subsequent model simulations.

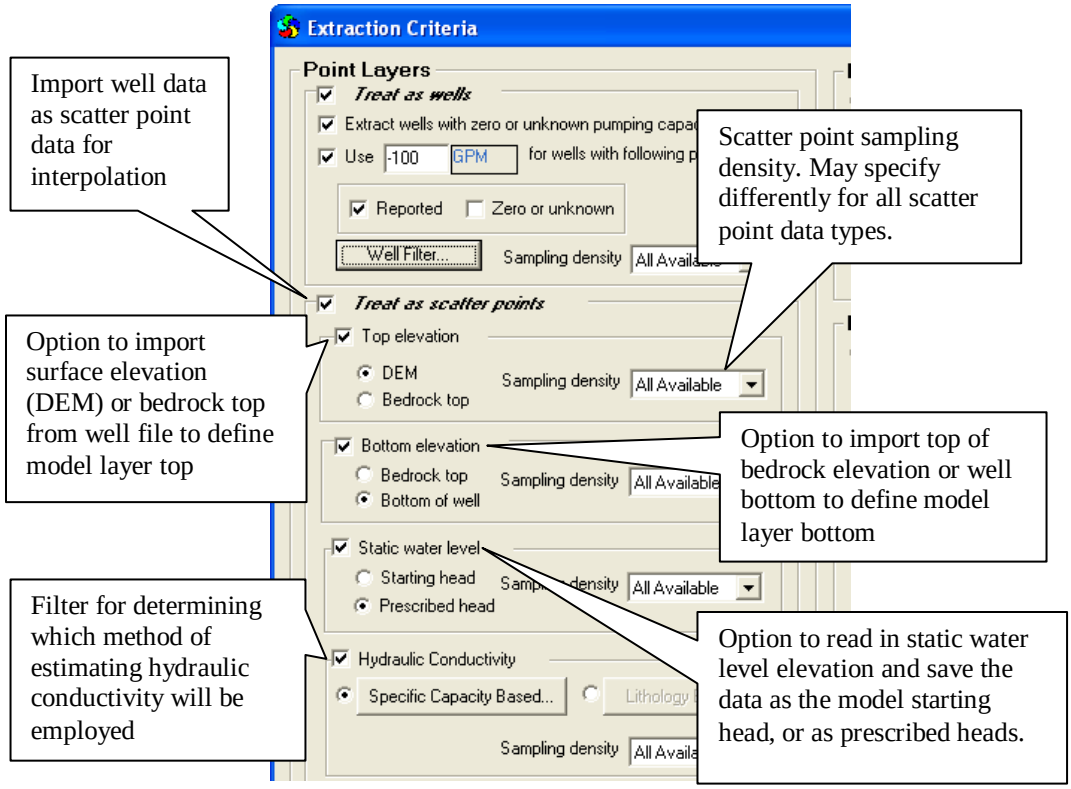


Figure 22-34 Scatter Point Extracting Criteria

22.17.3. Estimation of Hydraulic Conductivity

It is possible to use the information extracted from the **WELLOGIC** or **GWIM** databases for obtaining initial estimates of hydraulic conductivity to be exported to the **IGW Modeling Environment**.

> **i** Care should be taken when using either dataset as they are based on information reported by water well drillers and not carefully controlled in-situ testing.

There are two methods of obtaining these estimates:

1) Using specific capacity calculated from data obtained from well logs (**Specific Capacity Based…**), or
2) Using the hydraulic conductivity estimated on the basis of aquifer lithology (**Lithology Based…**). These methods are selected by checking the **Hydraulic Conductivity** box and then selecting either radio button shown in **Figure 22-35**.

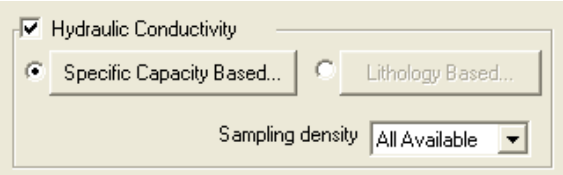


Figure 22-35 Selecting the Method for Calculating Hydraulic Conductivity

After selecting **Specific Capacity Based** option, the following window opens (**Figure 22-36**). There are four different methods for estimating transmissivity from specific capacity information. The first three methods (**Theis, Bradbury and Rothschild, and Modified Theis**) are based on variations of the Theis non-equilibrium well hydraulics model.

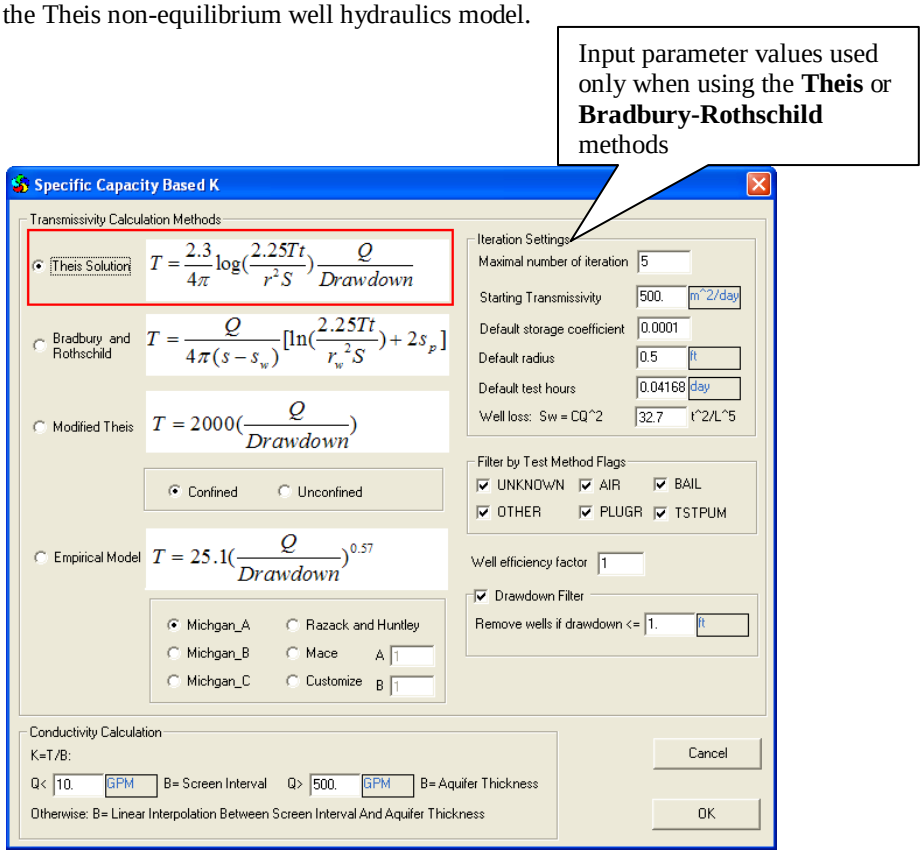


Figure 22-36 Iteration Settings for Theis and Bradbury Methods

In the upper right portion of this window (**Figure 22-36**), there is a section labeled **Iteration Settings**. These setting apply only to the **Theis** and **Bradbury-Rothschild** methods. The user can specify values for the different parameters in either equation or use the default values for these parameters. The program calculates the transmissivity value using these initial parameter values and the specific capacity data.

The **Modified Theis** method (**Figure 22-37**) is a commonly-used simplified variation of the **Theis** method. By assuming default values for all of the parameters in the Theis equation, it can be shown that the value for the specific capacity multiplier of a confined aquifer is approximately equal to 2000, and 1500 for an unconfined aquifer. Selecting the **Confined** or **Unconfined** radio buttons changes the multiplying factor from 2000 to 1500.

The last method, **Empirical Model**, (**Figure 22-38**) is a collection of methods that are based on a field data-derived empirical relationship between specific capacity and transmissivity. The user may presently select one of four empirical models: **MSU/MDEQ** estimated from aquifer tests for wells open to the glacial drift in Michigan, **Razack and Huntley** for unconsolidated alluvial deposits, **Mace** for limestone with secondary permeability, and **Customize**. By selecting **Customize**, the user may specify site-specific factors for this empirical relationship.

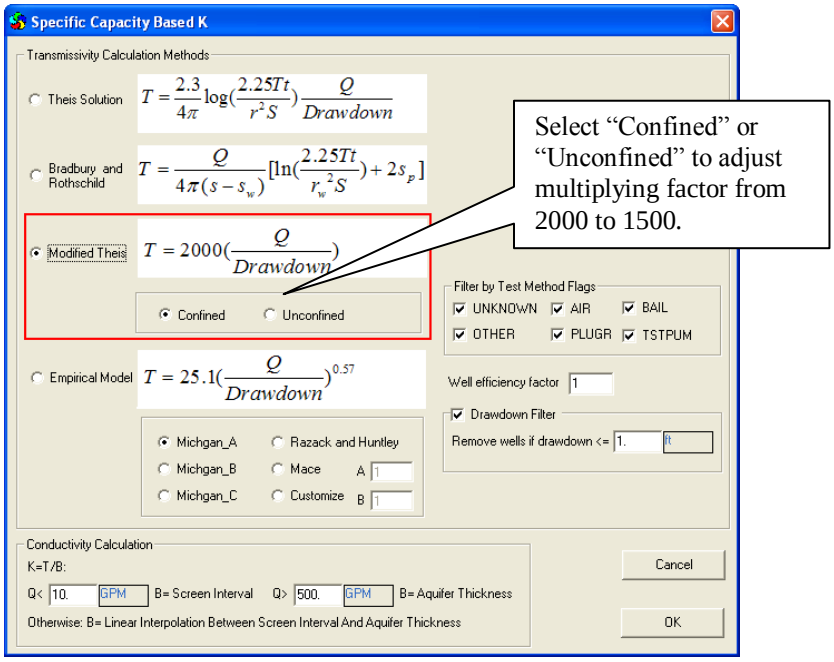


Figure 22-37 Modified Theis Method

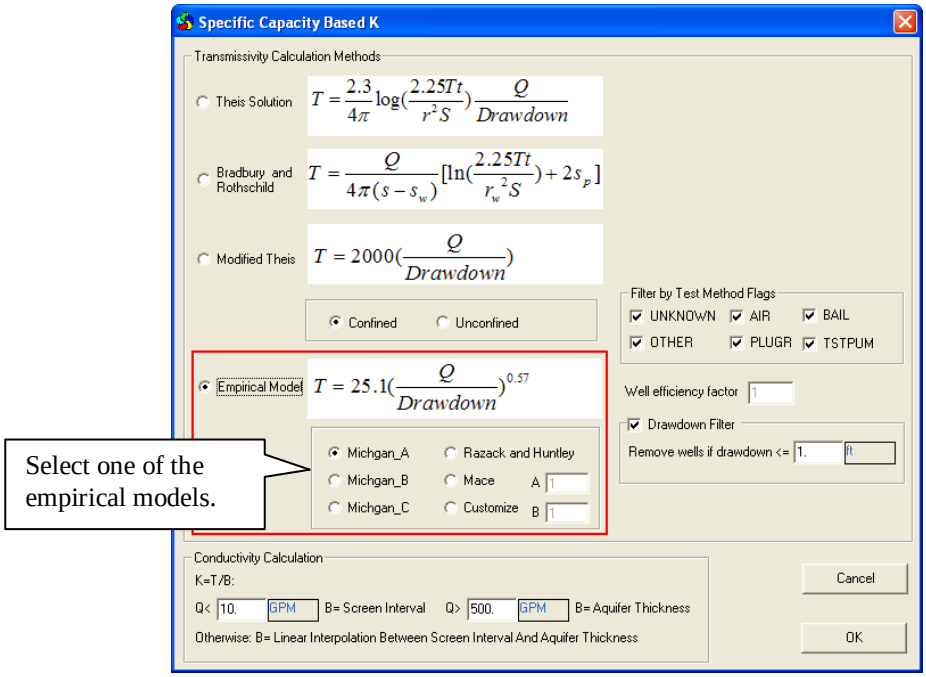


Figure 22-38 Empirical Model

At the bottom left corner of **Figure 22-39**, there is an option to define the efficiency of the well. This **Well efficiency factor** only applies to the specific capacity data and the methods, based on the Theis non-equilibrium model (**Theis**, **Bradbury and Rothschild,** and **Modified Theis**). The

reported drawdown is multiplied by the **Well efficiency factor.** A **Well efficiency factor** of 1 represents a 100% efficient well. Likewise a value of 0.6 represents a well that has 60% efficiency. Multiplying the drawdown by the **Well efficiency factor** results in a proportionally higher specific capacity and estimated transmissivity.

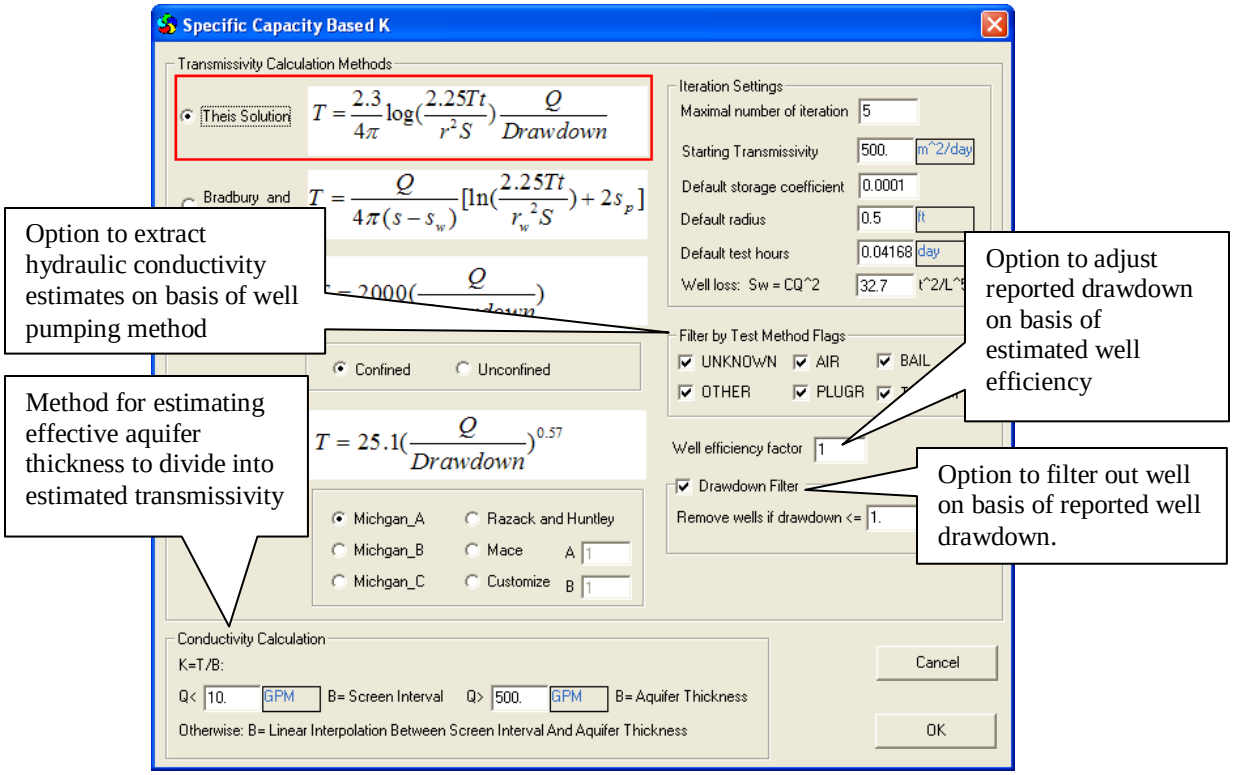


Figure 22-39 Specific Capacity Based K - Other Options

The transmissivity estimates must be divided by the aquifer thickness in order to obtain hydraulic conductivity. However, wells from which specific capacity data are derived are typically residential wells whose reported test pumping rate is very low. Because of this, it's not likely that a significant portion of the aquifer was stressed during initial testing. The user has the option of using only the thickness of the screen interval, the entire aquifer thickness, or some value in-between. To use the screen interval thickness, the user must specify a value for pumping rate that is much greater than any value in the extracted data. This value should be placed in the first box in the lower left portion of the window shown in **Figure 22-39**. The value in the second box should be equal to or greater than this value. To use the aquifer thickness, the user must specify a value much smaller than the smallest value in the extracted data. This value must be placed in the second box. In this example, the value in the first box should be less than or equal to the value in the second box. To use a value between the screen interval thickness and aquifer thickness, the user should specify a first box value that is typical of the lower range of pumping, and a value that is typical of the upper range of pumping should be placed in the second box.

The user also has the option of including only the specific capacity data derived using a particular well testing method, by checking any or all of the boxes to the left of the method (**UNKNOWN**, **AIR** – air lift, **BAIL** – bailer, **OTHER**, **PLUGR** – plunger, **TSTPUM** – test pump). It may be that the method of estimating pumping rate or measuring drawdown with a particular test method may not be as accurate as another method. The methods for which the box has been checked (**Figure 22-39**) will be extracted to IGW.

As with wells, the user has the ability to filter scatter point data on the basis of aquifer type. This is done by selecting the **Scatter Points Filter** button at the bottom of the **Extracting Criteria** window (**Figure 22-40**).

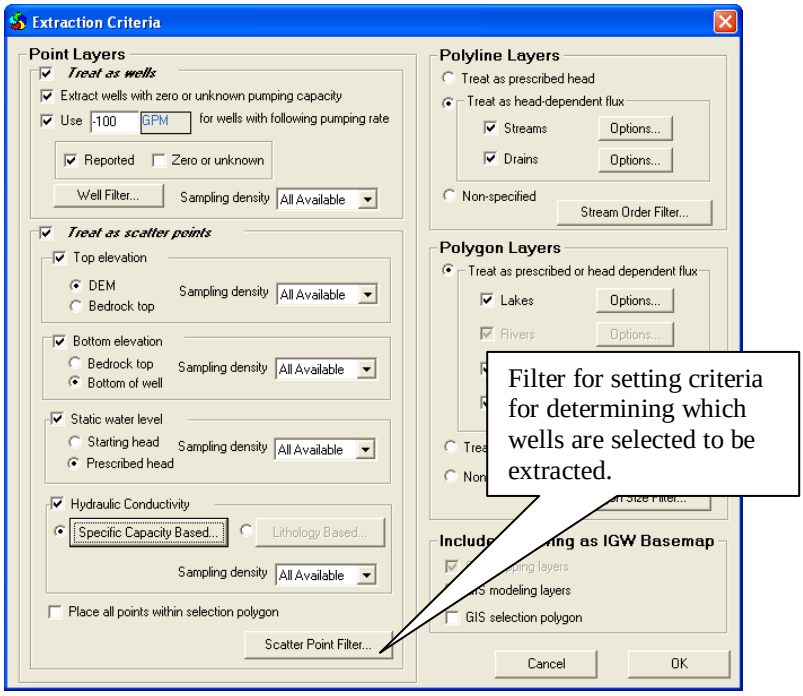


Figure 22-40 Scatter Point Extraction Filter Button

The **Filter Setting** window for the scatter points is the same as the window for the **Well Filter** (**Figure 22-33**).

22.17.4. Polyline Data

Rivers, streams, and drains are specified in the **MIV** database as polyline shape files. The user has the option of importing polylines as a prescribed head boundary (**Treat as prescribed head),** a head-dependent river or drain (**Treat as head-dependent flux**), or as an unspecified boundary (**Non-specified**), (**Figure 22-41**).

If the user imports the polyline as a prescribed head, a head value is assigned to the prescribed head boundary, taken from an average of the Digital Elevation Model (DEM) values for the beginning and end of the polyline. If no DEM values exist, a prescribed head value of zero is assigned. Within the **IGW Modeling Environment**, the user may assign a site-specific value that is more representative of stage elevation within that river or drain to these prescribed head polylines.

Polyline shapefiles that are imported as rivers or drains are assigned values for stage, depth to stream or drain bottom, riverbed, or drain leakance. These are obtained from a post-processed form of the USGS 100K NHD river shapefiles. These contain the stream order, estimated river stage (from 30m DEM), estimated hydraulic gradient (from 30m DEM), and USGS baseflow estimates that were derived as part of the **GWIM** project.

The user may select the (**Options…**) lookup table for **Streams** or **Drains** (**Figure 22-41**) to vary the riverbed or drain leakance on the basis of stream order. This will determine how the stream or drain segments are to be modeled, river bottom, and drain bottom elevation (**Figure 22-42**). At this level, the user may specify the leakance value for all stream segments on the basis of stream order number. At any time during the modeling process, the modeler may change the value for any stream segment, independent of all other stream segments in the model. The larger the stream order number, the farther downstream the polyline. A user may make the assumption that the

width of the stream is greater and the streambed leakance should be greater. It is also possible to determine how the exchange of water between the polyline and the adjoining aquifer is calculated.

> **!** The user may treat each stream segment, by order, as a river (two-way) or drain (one-way), but not both. The difference is that specifying the polyline as a river allows water to move back and forth between the polyline and the aquifer. With a "drain" specification, water may only move from the aquifer to the polyline (i.e. a drain can obviously not be a source of water for the aquifer).

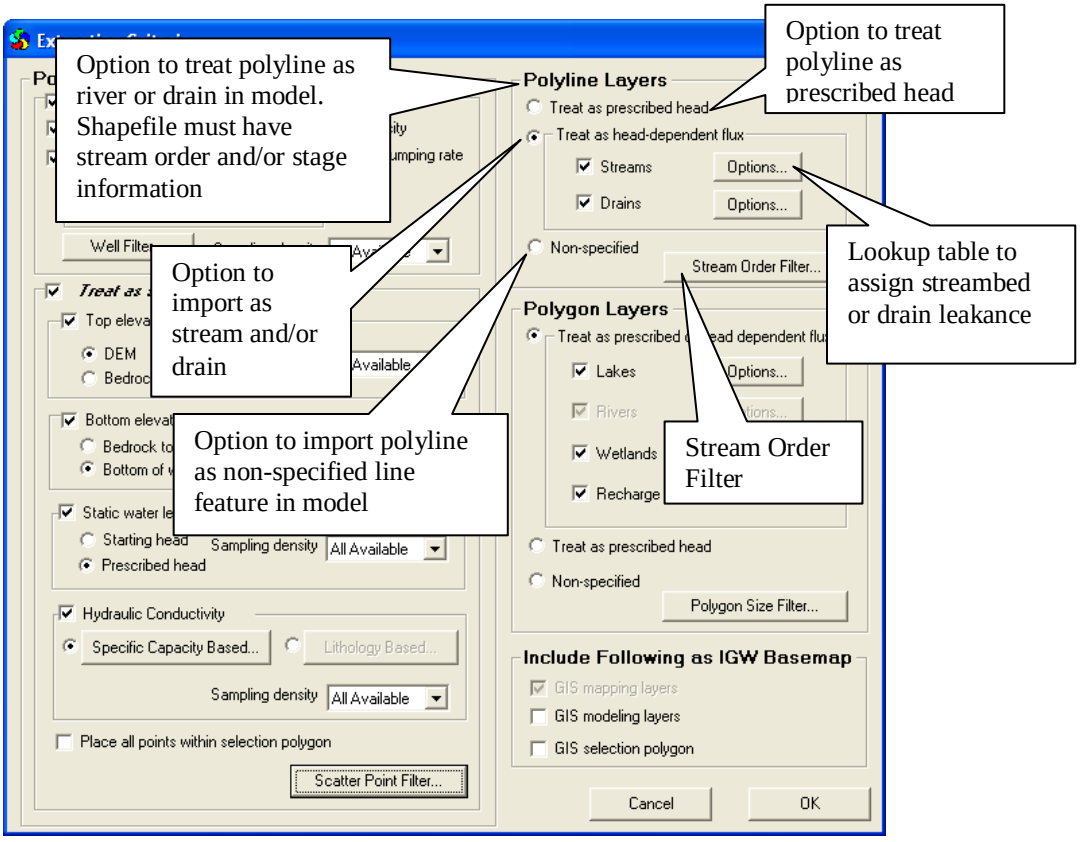


Figure 22-41 Polyline Extraction Criteria

It is also possible to treat all stream orders the same by assigning the same leakance or specifying that the stream functions as a "river" or "drain" in the model. The user may also assign a constant leakance value to all polylines and ignore stream order.

The stage for the polyline is read automatically from the polyline shapefile (the user has no ability to control this feature). This value is derived from 30 meters DEM polygons for the area. The user can assign a uniform stream bottom depth. The bottom elevation is calculated by subtracting this stream bottom depth from the stage assigned to that location.

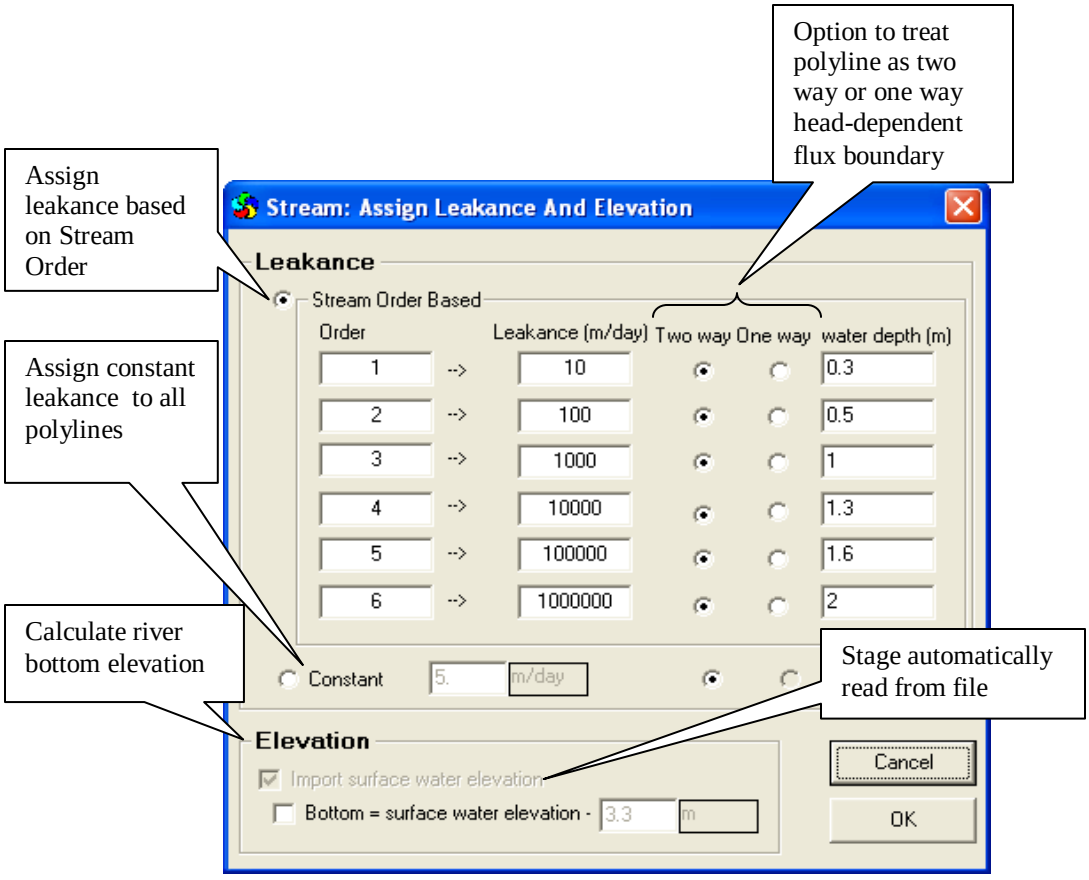


Figure 22-42 Leakance and Stream Order Lookup Table

With the **Stream Order Filter** (**Figure 22-43**) it is possible to use stream order as a criterion for extracting polylines. It is possible to simply extract all polylines, extract only polylines that have a stream order greater than a determined value, or simply equal to a single stream order.

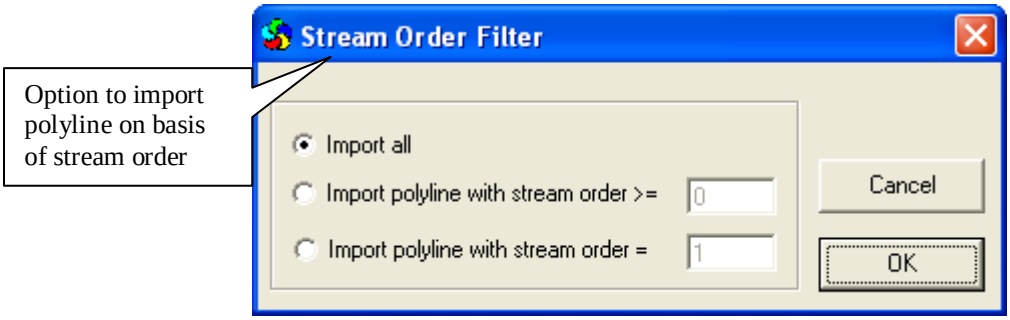


Figure 22-43 Stream Order Filter

22.17.5. Polygon Data

Surface-water bodies, wetlands, and estimated groundwater recharge are represented in the **MIV** and **GWIM** databases as polygon shapefiles. The extraction filter options for polygons are shown in **Figure 22-44**.

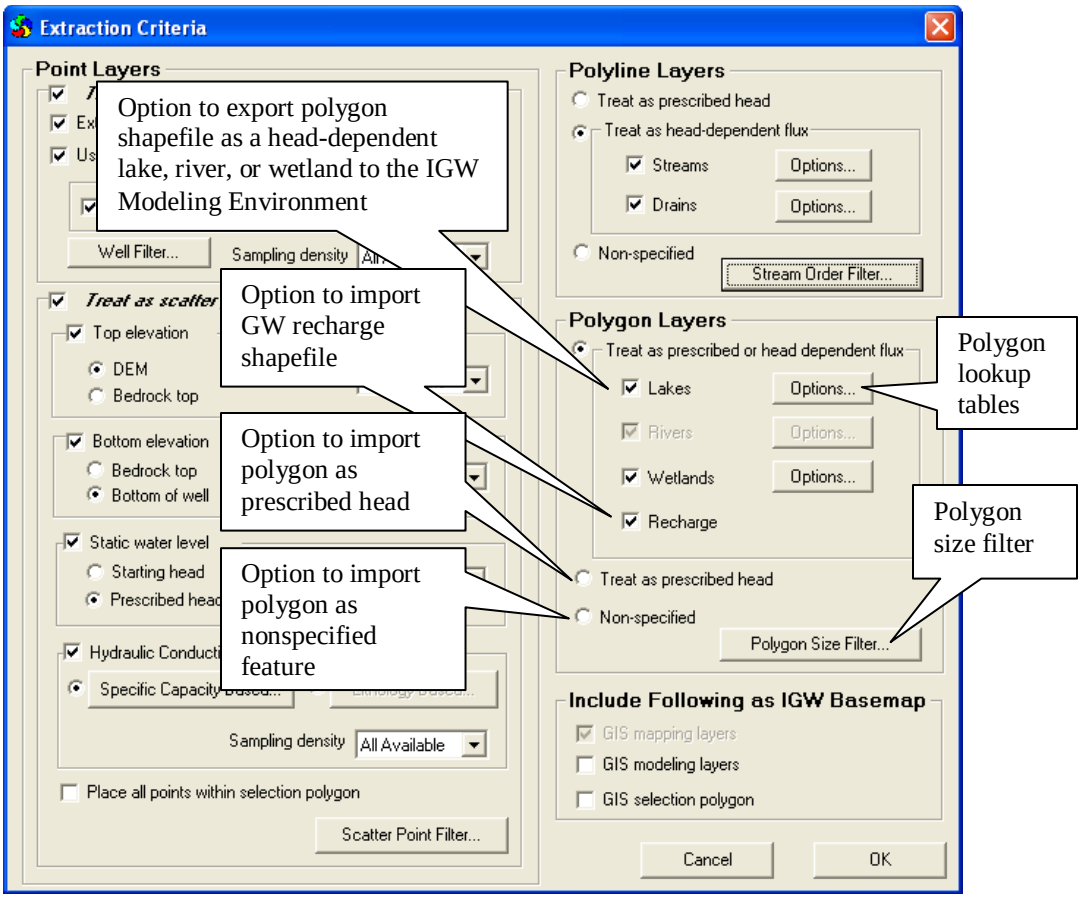


Figure 22-44 Extracting Polygon Data

The user has the option of treating these polygons as a prescribed or head-dependent flux boundary (**Treat as prescribed or head-dependent flux**), as a prescribed head feature (**Treat as prescribed head**), or as a non-specified feature (**Non-specified**). There are three options for defining head-dependent flux features: **Lakes, Rivers,** and **Wetlands**. Specific polygon shapefiles from the **MIV** database must be imported into the **GIS Model Importer** to select these options. These are the polygon shapefiles that describe all lakes, rivers, and wetlands.

The recharge polygon shapefile must be imported to select **Recharge**, as this is the only shapefile that may be used to define a prescribed flux boundary feature. During the extraction process, the **GIS Extractor** searches for specific field identifiers in each shapefile. If these specific field identifiers are missing from the shapefile, the polygons are read in as **Non-specified** polygons. Lake, river, and wetland polygon shapefiles may be read in as prescribed head features. The **GIS Extractor** will select the head value from these polygons to use as the prescribed head value. At any time during the modeling process any unspecified polygon boundary may be changed to head-dependent, specified flux, or a prescribed head boundary.

The lookup table for setting leakance values for the surface area of a lake is shown in **Figure 22-45**. The rationale for subdividing lake polygons on the basis of size is that relatively small surface-water bodies may be perched above the regional water table, and the bottom sediments may have a low hydraulic conductivity relative to much larger lakes. At this level, the user may specify the leakance value for all lake polygons on the basis of the area of the polygon, or treat them all the same. As always, at any time during the modeling process, the user may change the value for any lake polygon independent of all other lake polygons in the model.

The table is formatted so the user may divide the lake polygons on the basis of either relative-area percentile or actual area. For relative size, 10% represents the polygons whose area exceeds 10%

or less of all of the lake polygons in the model (these are the smallest polygons). Conversely, 100% represents the polygons with the largest area. The leakance values for these different percentiles are placed in the middle column. The user has the option of specifying a different leakance value for each percentile or area, or using the same value for all polygon sizes. The user also has the option of using different head-dependent flux equations (**Two-Way** or **One-Way**) for different size of polygons, or using the same equation for all polygons.

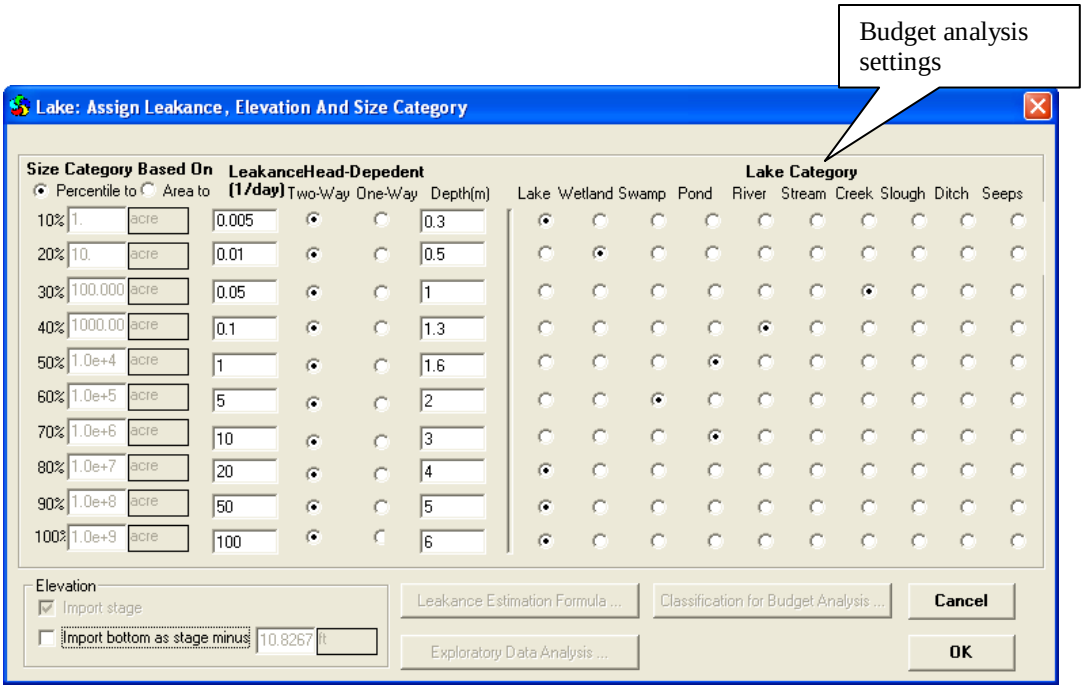


Figure 22-45 Lake Polygon Lookup Table

There is additionally the option of importing a surface-water bottom that is a specified distance below the stage. This is done by selecting **Import bottom as stage minus**. The user specifies this distance and the measurement units. If this option is not specified, a default distance to the surface-water bottom is assigned. It may be desirable to group lake polygons on the basis of size for analyzing regional water budgets.

The right hand side portion of the window titled 'Lake Category' is useful for water budget calculations. The flux into and out of the model is grouped according to these different arbitrary column headings when displaying water-balance information. These groupings have the sole purpose to help with tracking overall water budgets for different area-based groupings of lake polygons. No matter what name is assigned to a lake polygon, it will not have any impact on the modeling calculations.

The same procedures are used if polygons representing rivers are imported. The same windows as displayed for the lake polygons will open for these river polygons.

Selecting the **Options** button opposite the **Wetlands** polygon layer in **Figure 22-44** will open the window shown in **Figure 22-46**. There are five categories that generally describe the frequency of flooding of wetlands, listed under **Description**. The user has the option of specifying the leakance rate and the head-dependent equation that will be solved for that wetland "category." As always, any of these settings may be changed for individual polygons within IGW.

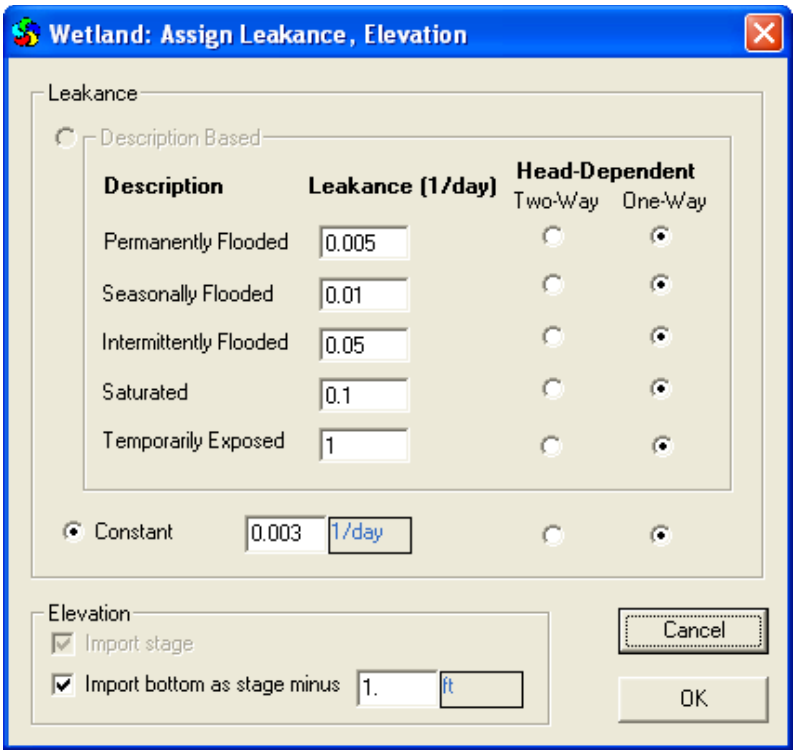


Figure 22-46 Wetland Lookup Table

With the **Polygon Size Filter** it is possible to use size of the polygon area as a criterion for extracting additional polygons. The user can extract all polygons, extract only polygons that have an area greater than a determined value, or extract polygons having an area greater than a specified percentage of all polygons in that layer. Selecting the **Polygon Size Filter** button shown in **Figure 22-44** will open the window shown in **Figure 22-47**.

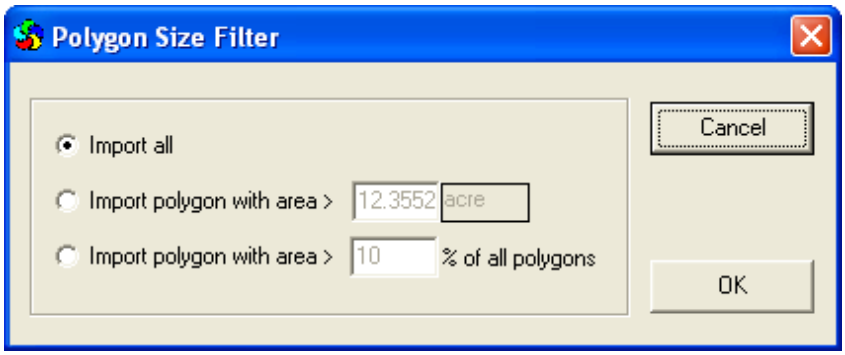


Figure 22-47 Polygon Filter

Finally, the user has the option of extracting the groundwater recharge rates estimated by the U.S. Geological Survey by selecting **Recharge**. These are square mile polygons that cover the entire state, and there are no manipulative options for these polygons in IGW at this time.

All line work for the polygon shapefiles may be exported without attributes as **Non-specified** features. Within the **IGW Model Environment**, the user may specify the leakance, head, bottom elevation, or flux at any time if they choose to model any **Non-specified** polygons as prescribed flux, head-dependent flux, or prescribed head boundaries.

22.17.6. Mapping Criteria

Prior to extracting data to the **IGW Modeling Environment**, the user has the option of determining how information will be displayed within the **IGW Modeling Environment** (see **Include Following as IGW Basemap** in lower right corner of **Figure 22-48**).

The first option (**GIS mapping layers**) is to display all mapping layer features. This option is mandatory and the user has no control over its selection. The second option (**GIS modeling layers**) will display any model features as mapping features within the **IGW Modeling Environment**. The advantage of doping this is that the shapefile lines are retained, even if the layer is not used in the model. If this option is not selected, no modeling layer lines will be displayed if the layer is not used in the model.

The third option is to display either the selection rectangle or polygon (e.g. watershed boundary) that was used to select the model and mapping layers exported into the **IGW Modeling Environment.** The selection box will be displayed within the **IGW Modeling** workspace.

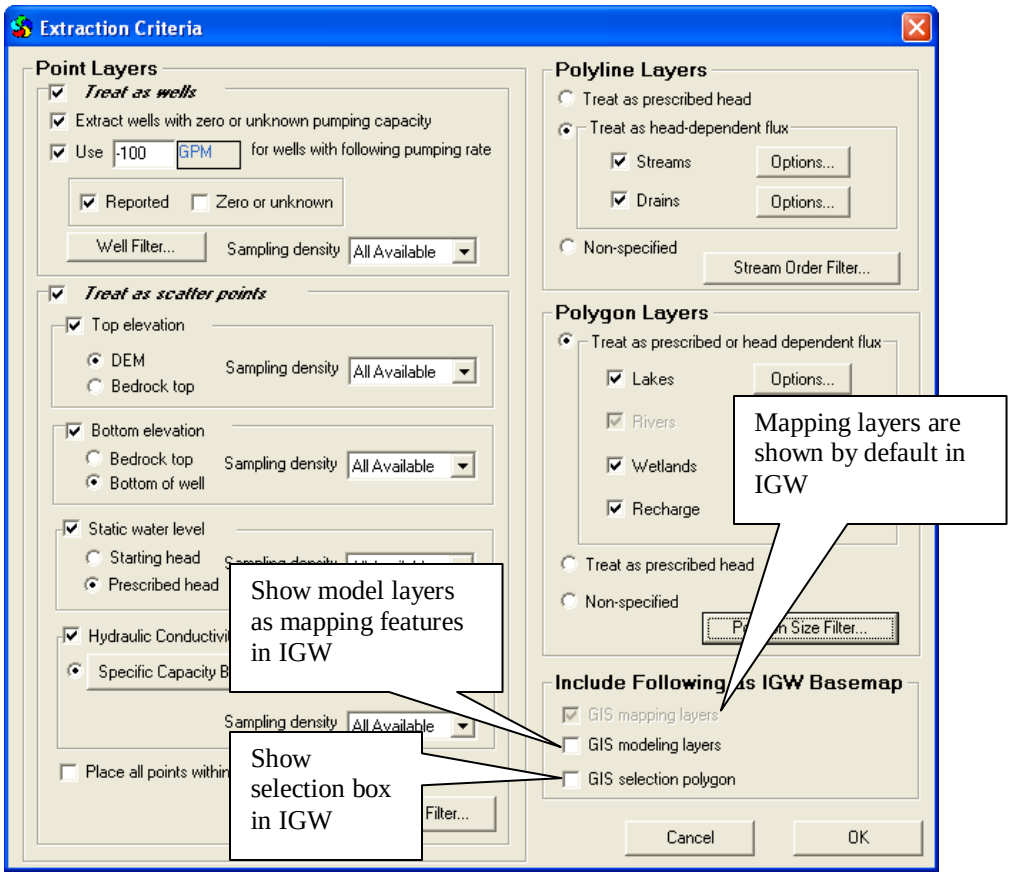


Figure 22-48 Selecting Mapping Criteria

22.17.7. Placing all Points in a Single Polygon

It is very likely that WHPA delineations may cross county or watershed boundaries, or involve the selection of scatter point data from more than one county or watershed. These point layers are contained within separate polygons, one for each county or watershed. These will show up in the **GIS Layer Explorer** as separate point layer shapefiles under **Layers for Modeling**. In order to interpolate across polygons (county or watershed boundaries), it is necessary to place all point

layer shapefiles in the same polygon. This is done by selecting the **Put all points within selection polygon** option at the bottom of the **Extracting Criteria** window (**Figure 22-49**).

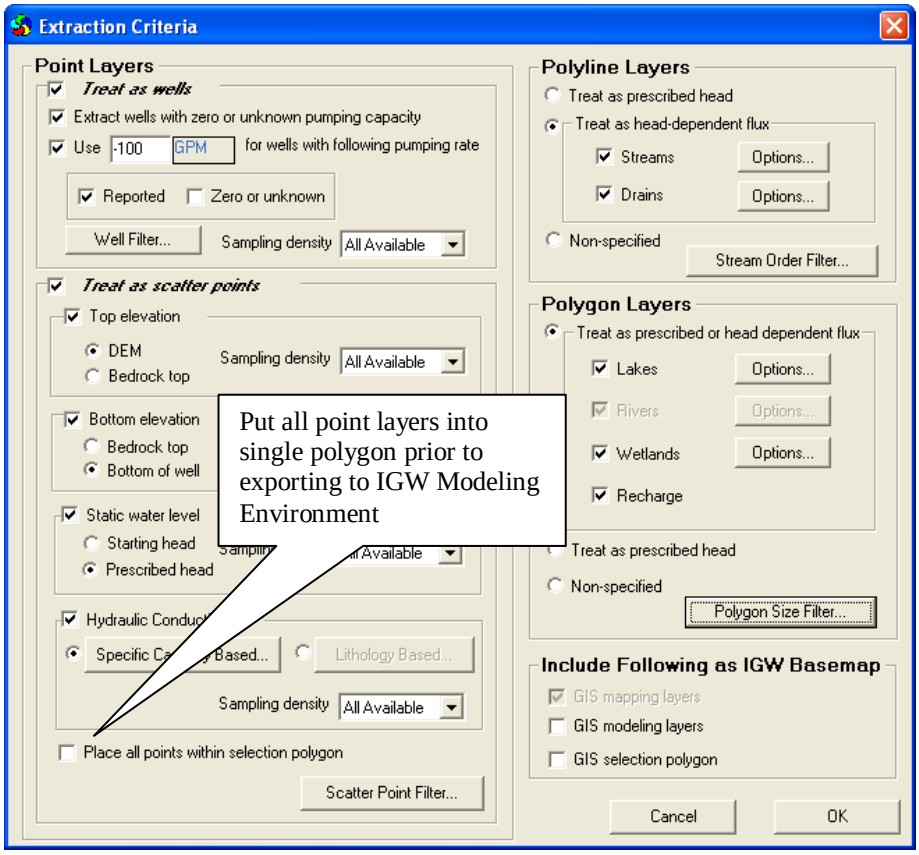


Figure 22-49 Put All Point Layers Into a Single Polygon Feature

Select **OK** to close the **Extraction Criteria** window. The selected data and mapping features are exported to the **IGW Modeling Environment** (**Figure 22-50**).

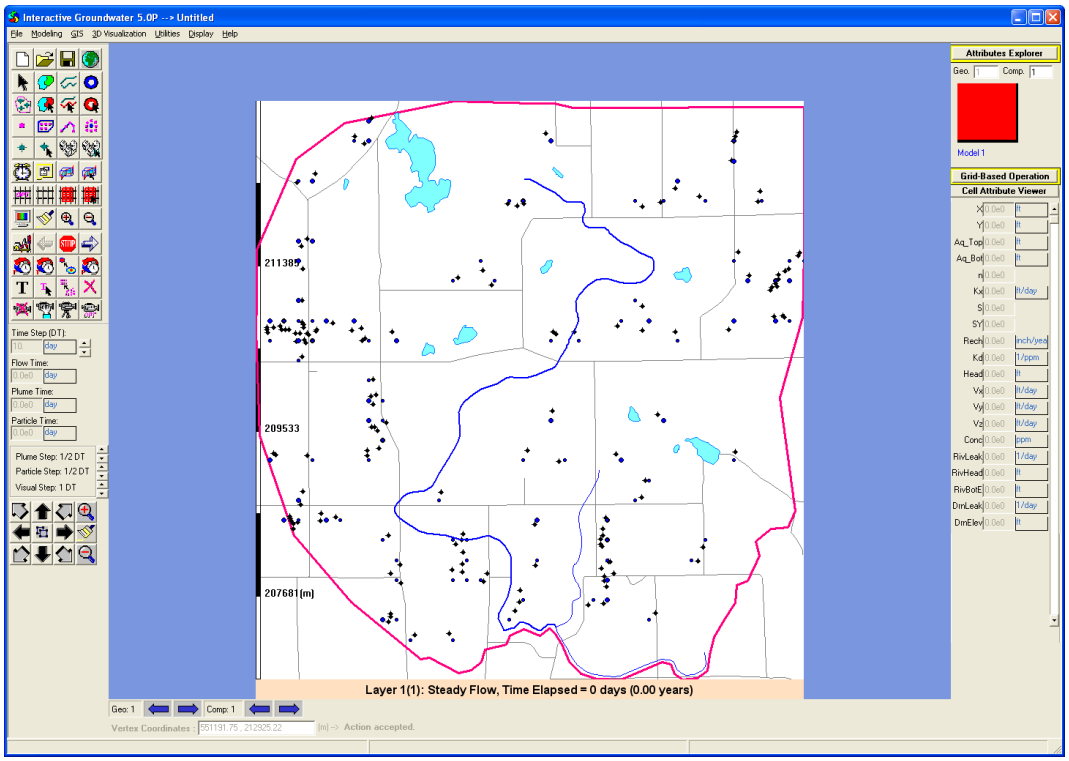


Figure 22-50 GIS Data imported to IGW Modeling Environment

22.17.8. Multiple Data Extractions Using the Same Selection Box or Polygon

Once in the **IGW Modeling Environment**, the user may find that it is necessary to extract additional data from the **MIV or GWIM databases**. While this is not recommended, it is possible to make multiple data extractions using the **GIS Model Importer** to invoke the same selection area as the first extraction. When a second selection is made using the same box or polygon, a warning appears asking the user that 'Some of the selected polygons are already in IGW. Do you want to add all duplicate polygons?' (**Figure 22-51**). The user should click 'No' to avoid creating another duplicate zone within the model. one of the polygons already exists.

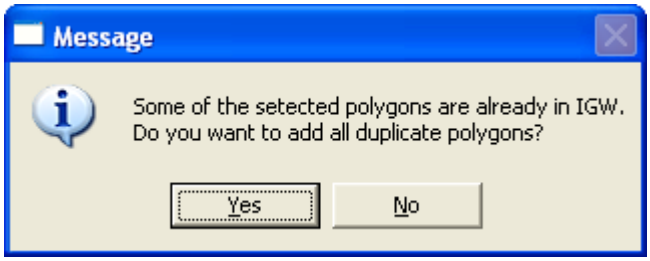


Figure 22-51 Duplicate polygon warning message

If the user clicks 'Yes', the data from the second extraction will be placed in a second zone that overlays the first zone. This may or may not be desired, as it can significantly increase the time needed to interpolate scatter point data or to process polyline / polygon data in the IGW Modeling Environment.

R In general, multiple extractions are to be avoided for model areas where there are several hundred scatter points, polylines or polygons. It is best to "re-do" the extraction, selecting all appropriate data over a large area so that a single extraction will be sufficient.

After making the selection in the **GIS Model Importer**, depress **OK**. The **Extraction Destination** window (**Figure 22-52**) opens. This gives the user the following options:

1) Add selected data to current model layer,
2) Add selected data to a new model layer, or
3) Delete existing data.

Under the first option, it is possible to delete all data from the previous data extraction (**Delete all existing data**), or to delete part, or none, of the data in the active model layer (**Select existing data to delete** area in the **Extraction Destination** window). All shapefile types that are checked will be deleted. If no shapefile types are checked, no data will be deleted and the selected data will be added to the existing data.

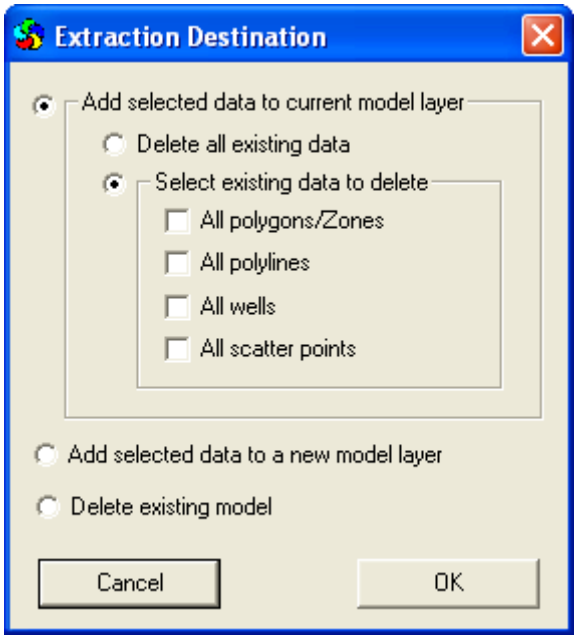


Figure 22-52 Option to Delete Selected Layers in the IGW Modeling Environment

After making the appropriate selections, select **OK** to complete the exportation to the **IGW Model Environment**.

! If the user decides to perform a second extraction, it is important to uncheck all data types previously exported to IGW. Otherwise, a second set of the same data will be exported to IGW. Duplicate data may create problems in the conceptual model.

R The user may find that it is more efficient in terms of data processing time to simply delete all data from the previous extraction, and perform a new extraction that includes the previously extracted data and new data. This will depend on the size of the area and the number of data involved. The more data involved, the more likely that it will be more efficient to delete all previously extracted data and perform a new extraction.

22.17.9. Importing GIS Files From an Anonymous Database

In order to transfer GIS data to IGW modeling environment, IGW needs two different types of GIS files:

1) Shape file (*.shp), and
2) Database file (*.dbf)

The **IGW GIS Interface** has been designed to recognize the GIS files **from Statewide Groundwater Database for Michigan**. In order to import a GIS file from another database, the user should match the database structure of IGW GIS protocol. If the user has a readily available database for a groundwater system other than that of Michigan, then they can simply use that database after making slight modifications to the spreadsheet file.

Chapter 23 FILE FEATURES

IGW Version 5.0P offers the user many options for saving files, data, and pictures. This chapter describes these options.

23.1. Graphics Capture

Capturing graphical output is very useful for subsequent use in papers and presentations. The following subsections describe the options available to capture IGW Version 5.0P screens and output as images.

23.1.1. Screen Capturing

The 'Screen Capture Options' window is the main interface for setting up screen capture functions. It can be accessed by clicking the 'Set Capture Option' button, or by selecting 'Screen Capture' from the **Utility** menu at row 12, column 4 of the button palette. It is pictured in **Figure 23-1.**

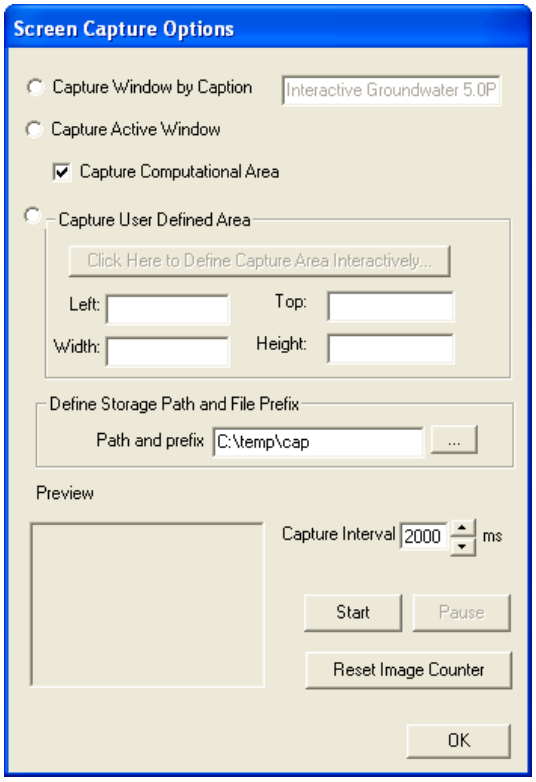


Figure 23-1 Screen Capture Options

The user has various options for capturing the entire screen or portions thereof. These are discussed in the following subsections. Once the desired option has been defined, clicking the 'OK' button sets the desired function in the software.

CAPTURE WINDOW BY CAPTION

Selecting this allows the user to enter the name of the window for capturing in the 'Caption:' field. When capturing commences, this window alone will be captured.

CAPTURE ACTIVE WINDOW

Selecting this allows the user to select the desired window interactively by making it active (this method will capture inactive windows that are in front of the active window). As capturing

commences, the user may change the active window by clicking them in series therefore sequentially changing the active window and capturing them in the order of the clicking sequence (only one window can be active at a time).

CAPTURE USER-DEFINED AREA

Selecting this allows the user to define a specific area of the screen to capture. After selecting this, the user subsequently clicks 'Click Here to Start Capture…' and then uses the special cursor to define a rectangular area to capture (click and hold the mouse at one corner of the rectangle, drag to the opposite corner, release the mouse button). The fields 'Left:', 'Top:', 'Width:', and 'Height:' update to show associated numerical values for the desired capture area (alternatively, the user may type in the desired coordinates in the field). The specific portions of any windows falling within that area are captured.

CAPTURE INTERVAL: MS FIELD

This field indicates how many milliseconds should pass between captures when using the timing capture mode, where the default value is 2000 (see **Section 23.1.2**). The user may enter a desired value or may adjust it using the associated arrow buttons. This field will accept values between 1 and 65535.

PATH AND PREFIX

This field indicates the desired path and file prefix to be used when saving the capture images. The user may type the desired path in the box or may surf to it using the '…' button (the default is c:\temp\). The default prefix is 'cap'.

START BUTTON

This button begins the capture timer (set in the 'ms' field). Capture will occur only if the 'External Calling Capture' button is activated beforehand.

STOP BUTTON

This button stops the capture timer after it is started using the 'Begin' button (as described above – also see **Section 23.1.2**).

RESET IMAGE COUNTER

This button resets the image counter. The counter starts with the number 0001 and counts up. This number is added to the prefix as the file name for a particular image. Resetting the counter will cause the software to overwrite preexisting files unless the path or prefix has been changed.

PREVIEW AREA

This is the gray rectangle near the bottom left-hand corner of the window. It displays a preview of the last image captured.

23.1.2. Motion Capture

There are two ways to capture pictures using the settings in the 'Automatic Capture' window.

The first is activated by clicking the 'Timing Capture' button. When this is selected, the software will capture an image at every interval specified in the 'ms' field (in the 'Automatic Capture' window). This continues until the 'No Capture' button is clicked.

The second is activated by clicking the 'External Calling Capture' button. When this is selected, the software will capture an image each time the model is solved or iterates. This continues until the simulation ends or the user stops the simulation. Capturing will commence the next time the model is solved unless the 'No Capture' button is clicked.

Also, when this button is clicked, the manual timer in the 'Automatic Capture' window can be activated using the 'Begin' and 'Stop' buttons described in the previous section.

Every time the software captures an image the computer will emit an audible beep (the sound defined as 'Default Beep' in the operating system).

23.1.3. Saving Display as a Picture

An alternative to using the capture option is to manually save the current Working Area state as a picture[27].

On the **File** menu, the user selects 'Export Picture' and then 'as BMP' or 'as 'JPEG' from the cascading menu. The user then selects a path, types in a desired file name in the 'Save As' menu, and then clicks 'Save' to complete the procedure.

Using the 'Print Screen' function associated with the computer (usually a key on the keyboard) also saves the current screen in the Windows™ clipboard. The user can subsequently edit the picture with a variety of graphics editing programs, ranging from the relatively simple 'Paint' included in Windows™, to powerful third-party software packages such as Adobe™ PhotoDeluxe™.

23.2. Saving IGW Model Files

The user may save the current IGW Version 5.0P file by selecting 'Save Model to File' on the **File** menu[28]. This opens the 'Save As' window in which the user defines the desired path and types the desired file name. Clicking the 'Save' button closes the 'Save As' window and opens the 'Message' window with the text 'Do you want a description explaining the parameters in the IGW file?'. Clicking the 'Yes' button saves the file with extra explanation text as comments in the file. Clicking the 'No' button saves the file without the extra comments.

Clicking the 'Cancel' button in the 'Save As' window closes the window and aborts the file saving process.

IGW Version 5.0P files are saved with the extension '*.igw' and are in American Standard Code for Information Interchange (ASCII) text format.

23.3. Editing IGW Model Files

Advanced users may wish to edit *IGW Version 5.0P* files using a text-editing program such as Windows™ Notepad. This may be useful in instances where batch file processing is desired to reduce redundancy and save time. This feature may also be necessary in such instances as changing the location of the basemap.

IGW Version 5.0P files may be saved with extra parameters explaining comments to assist in manual file editing (see **Section 23.2**).

23.4. Exporting Data

Head and concentration data may be stored by selecting 'Export Model Results', for the current computational layer/whole model, from the **File** menu. Selecting this opens the 'Save As'

[27] Refer to **Section 17.2** of the ***IGW Version 5.0P Tutorials*** document for an example.
[28] Refer to **Section 17.1** of the ***IGW Version 5.0P Tutorials*** document for an example.

window and the user may browse to the desired path and type in the desired file name for the data store. Clicking the 'Save' button then opens the 'Output Data to File' window pictured in **Figure 23-2**. Clicking the 'Cancel' button aborts the data exporting process.

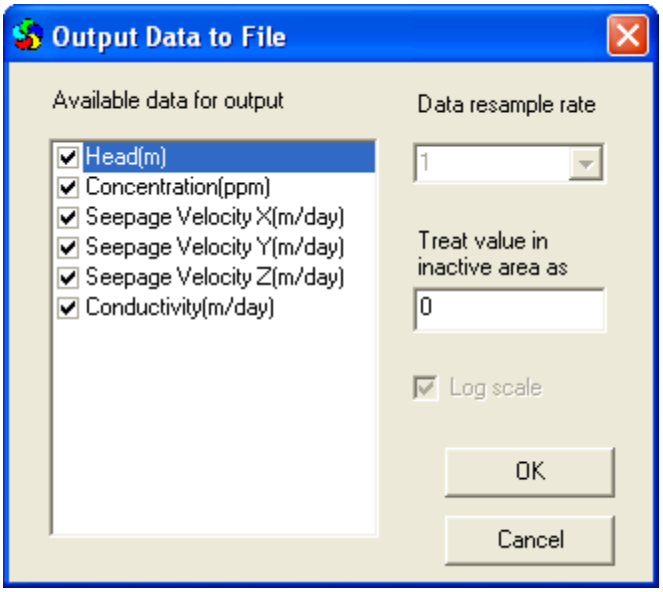


Figure 23-2 Output Data Types

In the 'Available data for output' area, the user places a checkmark in the appropriate boxes to indicate which data sets should be saved ('Head (m)', 'Concentration (ppm)', 'Seepage Velocity X (m/day)', and 'Seepage Velocity Y (m/day)' are checked by default). The user may set the 'Data resample rate' (the ratio of sampled nodes to total nodes) by using the appropriate list field (available settings are 1 [default], $\frac{1}{2}, \frac{1}{3}, \frac{1}{4}, \frac{1}{5}, \frac{1}{10}$). For example, a setting of 1 samples every node; a setting of ½ samples every other node, etc. The user may also assign the default value to the inactive Working Area space by entering the desired value in the 'Treat value in inactive area as' field. Clicking 'OK' creates the data file. Clicking 'Cancel' aborts the data exporting process (after verification in a separate window).

The resulting '.igd' file is an ASCII text-formatted document with the first column being the *x*-coordinate, the second column the *y*-coordinate, and additional columns for each data set selected in the 'Output Data to File' window (top to bottom indicates the order of the columns in the text file).

The exporting data process only saves data at the indicated time of the simulation.

23.5. Random Sampling

The user may conduct a random sampling of the Working Area by selecting 'Random Sampling' from the **Utilities** menu (**Section 3.3.5**). This opens the 'Data Random Sampling' window as shown in **Figure 23-3**.

R	The user should first select a zone in the Working Area and then select 'Random Sampling from the Utilities menu.

The '**Total Nodes in Current Zone:**' number indicates the total number of nodes available in the currently selected zone.

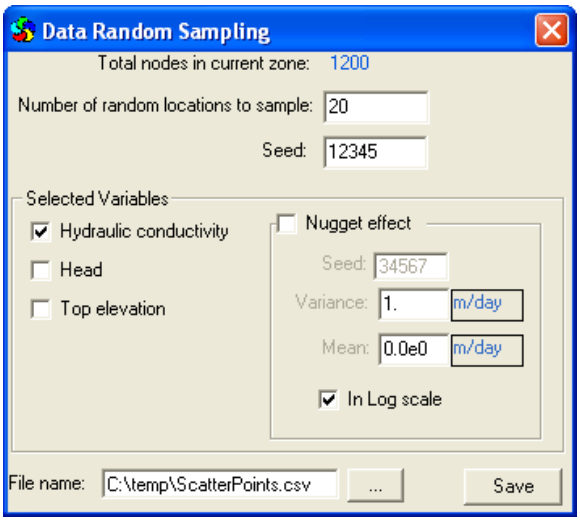


Figure 23-3 Random Sampling

The user may specify the number of points to be sampled in the '**Number of random locations to sample**' field (this number may be more than the 'Total Nodes in Current Zone:' number, however, repeat samplings will occur in this case). The 'Seed:' field allows the user to change the random field by entering different numbers.

The user may also specify the location and name of the file in the 'File name:' field (and using the ... button to browse). There are three options available in the 'Selected Variables' section: 'Hydraulic Conductivity', 'Head', and 'Top Elevation'. Information about the 'Nugget Effect' can be found in **Section 20.4**. Please also refer to **Figure 20-15**.

Clicking the 'Save' button creates the data file. Closing the window aborts the process.

The data is saved in the comma separated value (.csv) file format. The specific format is identical to that of scatter point input files discussed in **Section 7.7.4.**

23.6. Opening a Process File

Statistical data analysis and visualization tools of *IGW Version 5.0P* can be used to process data for a stochastic process (please refer to **Chapter 18** for stochastic modeling). A stochastic process, based on various realizations of logarithm of hydraulic conductivity (lnK), can generate data for heads and concentration in a model. Data can be stored for such stochastic processes. IGW can accept the process data in a *.DAT file format as explained below. A text editor, such as Microsoft® Notepad can be used to set the required format.

23.6.1. Monitoring Well Process File

Stochastic processes at a monitoring well are discussed in detail in **Section 18.5.2.2**. The processes at a monitoring well may contain data for hydraulic conductivity K (usually expressed as log of K, or lnK), head and concentration. A *.DAT file using the general layout shown in **Figure 23-4** can be opened and processed in IGW for statistical parameters and process visualization. The first row in the file is general information about the well in text form. The second row is also text. Third row contains the numeric value for the number of realizations. Fourth row is column names separated by space. Fifth row is empty. From sixth row onwards, respective numeric values for each column are entered, separated by space (the values can be entered in any numeric format). The first column is the realization number, the second column is lnK, the third is observed head, and the fourth is observed concentration.

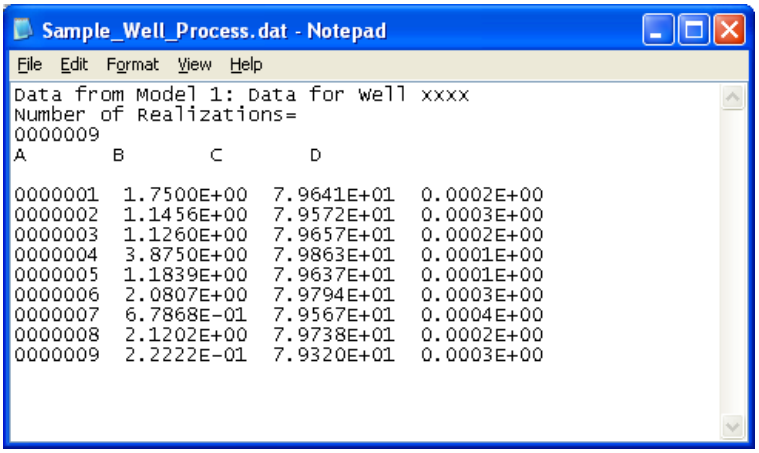


Figure 23-4 Example data of stochastic processes at a monitoring well

Once the data is imported into *IGW Version 5.0P*, for each process (lnK, head, concentration) the software can generate various plots shown in **Figure 23-5** [the process, mean ± standard deviation, histogram, PDF, CDF, mean realization, coefficient of variation]. The software also generates tables for statistical parameters of each process data including max, min, mean, median, mode, average error, standard deviation, skewness and kurtosis. This table can be seen on the right side of each plot in **Figure 23-5**.

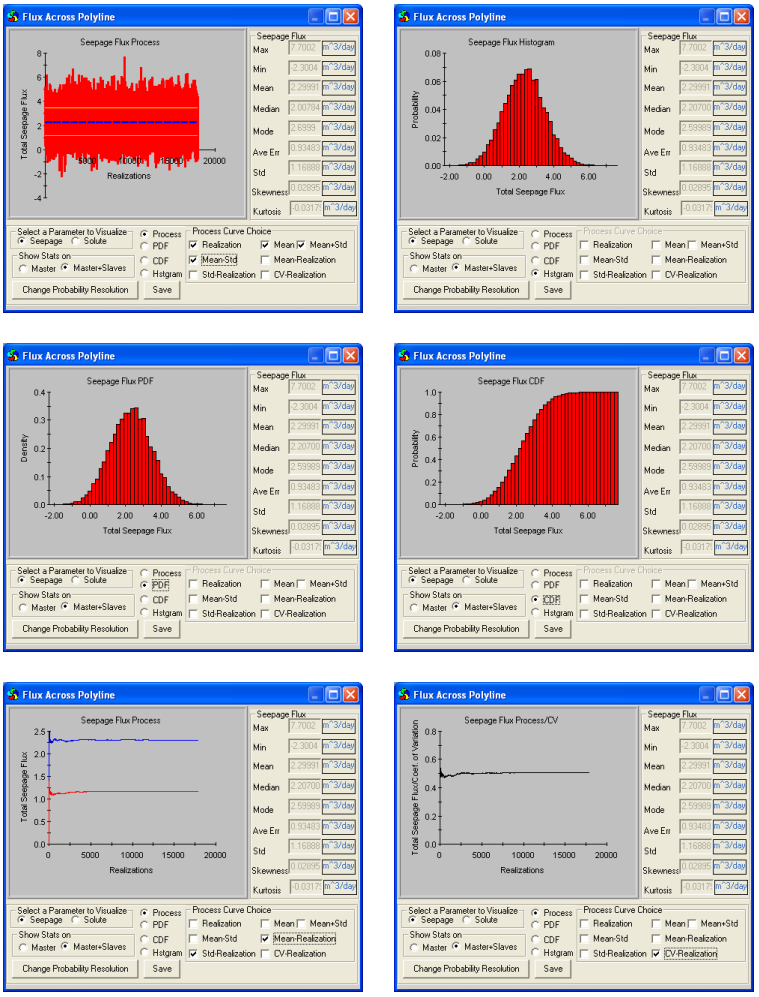


Figure 23-5 Plots from process files at monitoring wells

23.6.2. Polyline Flux Process File

Stochastic process at polyline includes total solute flux and total seepage flux across a polyline for a number of realizations. Data file in *.DAT format on these processes can be imported to IGW. The required layout of the data file is shown in **Figure 23-6**.

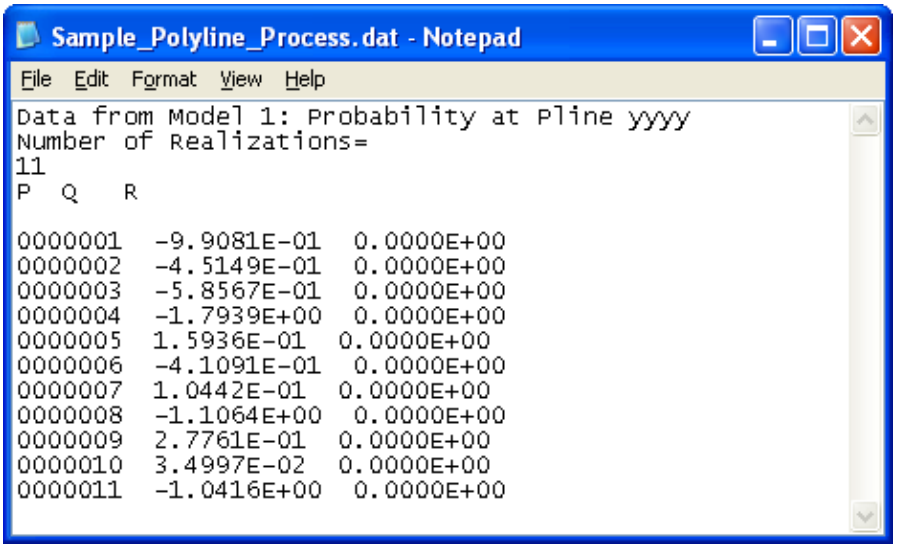


Figure 23-6 Example data of stochastic processes across polylines

A *.DAT file using the general layout shown in **Figure 23-6** can be opened and processed in IGW for statistical parameters and process visualization. The first row in the file is general information about the polyline in text form. The second row is also text. Third row contains the numeric value for the number of realizations. Fourth row is column names separated by space. Fifth row is empty. From sixth row onwards, respective numeric values for each column are entered, separated by space (the values can be entered in any numeric format). The first column is the realization number, the second column is seepage flux and the third one is solute flux.

Once the data is imported into IGW 5.0P for each process (seepage flux and solute flux), the software can generate various plots shown in **Figure 23-7** [the process, histogram, PDF, CDF, mean realization with ± standard deviation, coefficient of variation]. The software also generates tables for statistical parameters of each process data including max, min, mean, median, mode, average error, standard deviation, skewness and kurtosis. This table can be seen on the right side of each plot in **Figure 23-7**.

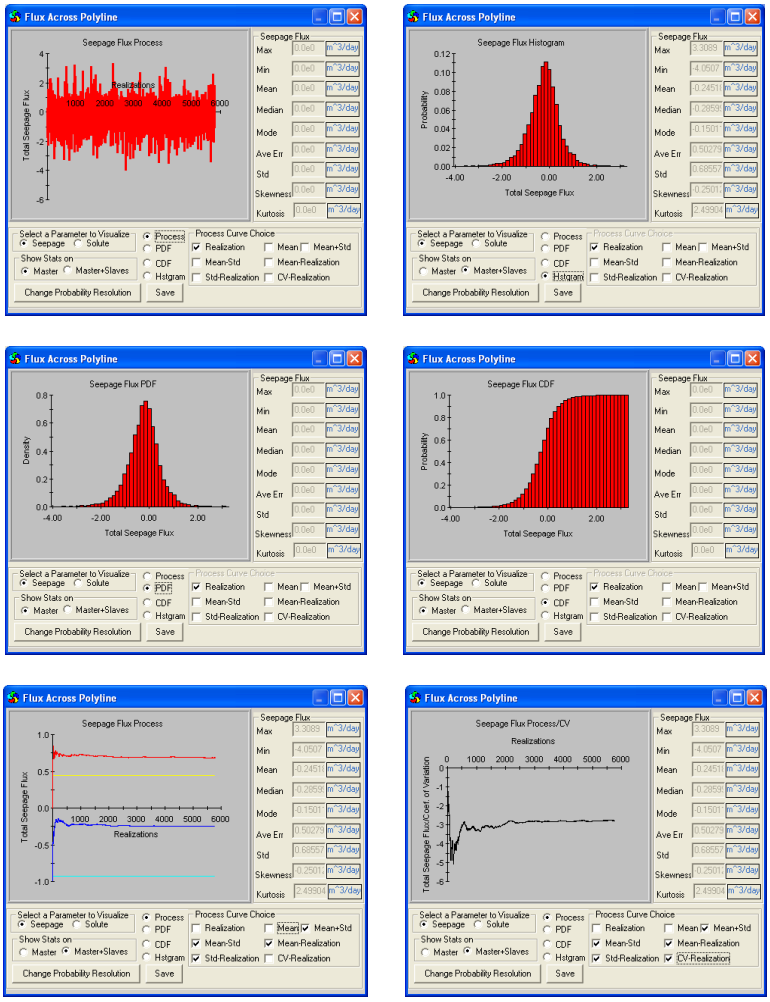


Figure 23-7 Plots from process files across polylines

Chapter 24 CONCLUSION

This user's manual was designed to give the user the ability to master all of the features and functions of *IGW Version 5.0P*. However, as *IGW Version 5.0P* is constantly evolving, so is this manual. Therefore, small discrepancies or inconsistencies might be encountered as the authors attempt to keep the software and its documentation robust and complete. Consult the tutorial and program help file for more information and stay alert for periodic software and documentation updates.

Feel free to contact the authors with any comments or concerns regarding this document or *IGW Version 5.0P* in general. If more information is desired regarding the *IGW Version 5.0P* file format or the parameters therein, please contact Dr. Shu-Guang Li, at lishug@egr.msu.edu .

Thank you for taking the time to examine *IGW Version 5.0P*. We hope you find it a powerful and empowering tool.

APPENDIX A: IGW VERSION 5.0P SOFTWARE INTERNAL MARKERS

This appendix explains certain IGW Version 5.0P internal software considerations that may be of interest to the user.

A-I Aquifer Type Parameter

The aquifer type parameter specifies the aquifer status of a cell. The software may assign one of nine classifications to each cell in a model. They are listed in **Table A-I-1**.

Table A-I-1 Classification Codes and Explanations for Aquifer Type

CODE	EXPLANATION	DISPLAY IN CAT
1	Confined - locked	Confined
2	Confined or Unconfined	Confined or Unconfined
3	Confined or Unconfined – never dry	Confined or Unconfined
4	Confined or Unconfined – transmissivity not head dependent	Confined or Unconfined
5	Unconfined – never dry	Unconfined
6	Unconfined – transmissivity not head dependent	Unconfined
7	Inactive	Inactive
8	None – not currently utilized	-
9	Unconfined – locked	Unconfined

The display in the CAT will read 'Inactive' if the cell state (see **Appendix A-II**) is set to 1 (meaning the cell is dry). This will occur even if the aquifer type is 1,2,4,6 or 9.

I At this point in time, IGW Version 4.7 assigns a code of 2 to each cell in the model. The original impetus to assign separate codes to each cell stemmed from the desire for increased computational efficiency, but given the processing speed of current computers, this distinction no longer yields measurable efficiency increases.

A-II Cell State Parameter

The cell state parameter specifies the hydraulic condition of a cell. The software may assign one of four classifications to each cell in a model. They are listed in **Table A-II-1**.

Table A-II-1 Classification Codes and Explanations for Cell State

CODE	EXPLANATION	DISPLAY IN CAT
0	The cell is wet	Wet
1	The cell is dry (temporarily inactive)	Dry
-1	The cell is permanently inactive	Inactive
None	No cell state has been assigned	N/A

The display in the CAT will read 'Inactive' if the Ibound parameter (see **Appendix A-III**) is set to –1.

A-III IBOUND Parameter

The Ibound parameter specifies the general activity of a cell. The software may assign one of three classifications to each cell in a model. They are listed in **Table A-III-1**.

Table A-III-1 Classification Codes and Explanations for 'IBound' Parameter

CODE	EXPLANATION	DISPLAY IN CAT
0	The cell is permanently inactive	Inactive
1	The cell is active	Active
-1	The cell is assigned a constant head value	ConstH

The display in the CAT will read 'Inactive' if the cell state parameter (see **Appendix A-II**) is set to 1.

APPENDIX B: RANDOM SETTINGS INTERFACES

B-I Option of Unconditional Random Field Window

The option of **Unconditional Random Field** window is shown in **Figure B-I-1**. This window is the common interface for most random settings in the software.

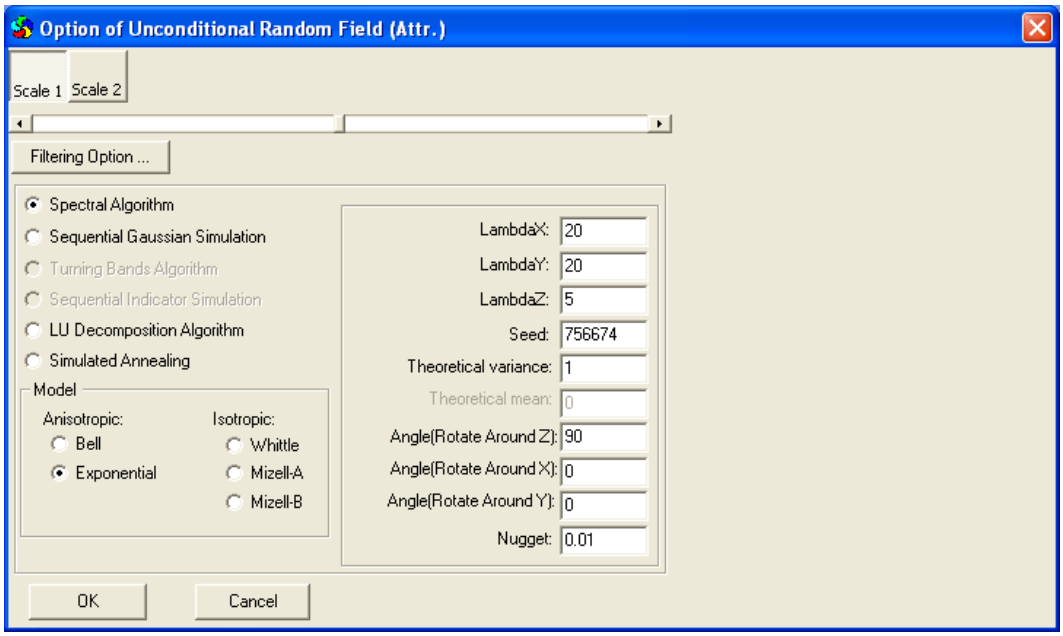


Figure B-I-1 Unconditional Random Field Attributes

The **Scale 1** and **Scale 2** toggle buttons allow the user to access two simultaneous but independent scales. Additional scales can be added by right-clicking in the upper portion of the window and selecting **Add New Scale** (the scale is added after the currently depressed toggle button). The currently selected scale can be deleted by right-clicking in the upper portion of the window and selecting **Delete Current Scale**. The effects of each scale on the overall random field are linearly additive. The scale buttons can be scrolled by using the arrows in the upper right-hand corner.

Note that the unit of measure associated with all **Lambda** parameters is meters. All other parameters are dimensionless.

Clicking the **OK** button sets any changes in the software. Clicking the **Cancel** button aborts any changes made and closes the window (after a verification prompt).

The user is given a choice of four different random field generation methods in the upper left corner of the window. They include:

1) Spectral Algorithm,
2) Sequential Gaussian Simulation,
3) LU Decomposition Algorithm, and
4) Simulated Annealing.

These are described further in the following sub-sections. Refer to the ***IGW Version 5.0P Reference Manual*** for more in-depth mathematics for each method and model.

B-I-I Spectral Algorithm

If this method is chosen (it is the default for all scales), a window appears similar to that shown in **Figure B-I-1**.

The user may subsequently select in the **Model** area whether to employ an **Anisotropic** model (**Bell** [default for **Scale 2** and all subsequently added scales], **Exponential** [default for **Scale 1**]), or an **Isotropic** variation (**Whittle**, **Mizell-A**, or 'Mizell-B').

LambdaX, **LambdaY**, **Seed**, **Theoretical Variance**, **Angle** (Rotation angle around coordinates), and **Nugget** are available. The default values can be observed by switching between **Scale 1** and **Scale 2**. For all subsequent scales, the default lambda values increase by an order of magnitude.

B-I-II Sequential Gaussian Simulation

If this method is chosen the main portion of the **Option of Unconditional Random Field** window appears as shown in **Figure B-I-II-1**.

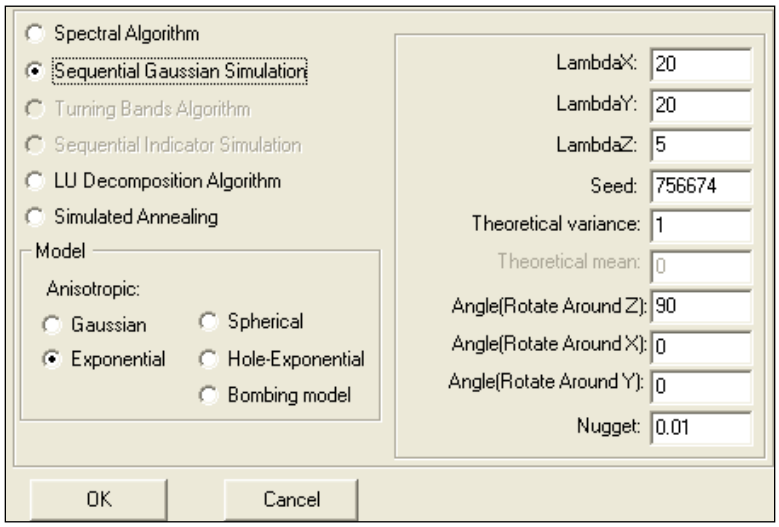


Figure B-I-II-1 Sequential Gaussian Simulation Window

The user may subsequently select in the **Model** area which **Anisotropic** model to employ. The options are **Bell**, **Exponential** (the default for all scales), **Spherical**, **Hole-Exponential**, or **Bombing model**.

Available fields are **LambdaX**, **LambdaY**, **Seed**, **Theoretical Variance**, **Angle** (Rotation angle around coordinates), and **Nugget** are available. The default values can be observed by switching between **Scale 1** and **Scale 2**. For all subsequent scales, the default lambda values increase by an order of magnitude.

B-I-III Turning Bands Algorithm

If this method is chosen, the main portion of the 'Option of Unconditional Random Field (Attr.)' window appears as shown in **Figure B-I-III-1**.

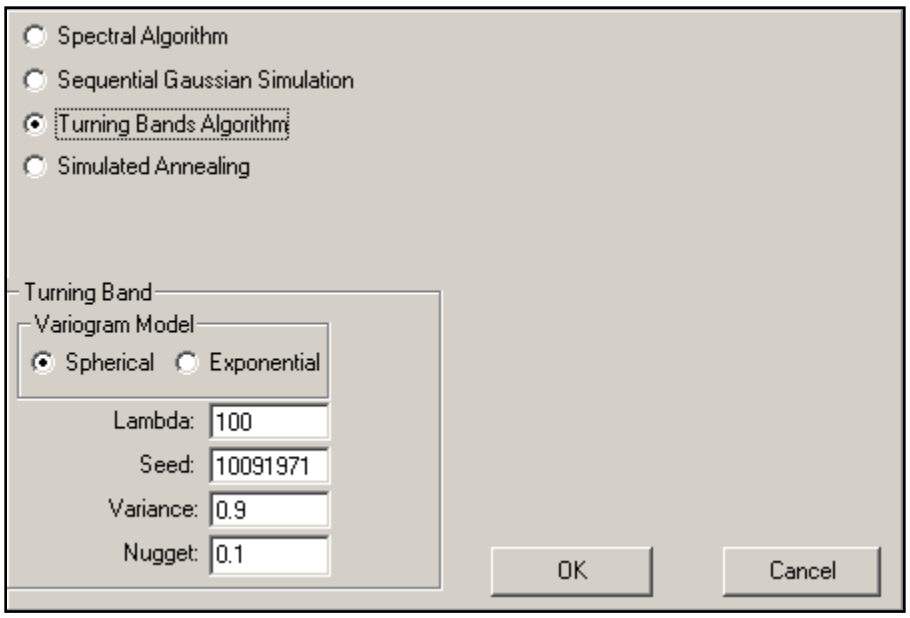


Figure B-I-III-1 Turning Bands Algorithm Options

The user has the option in the 'Variogram Model' area of choosing either a 'Spherical' model (the default for all scales), or an 'Exponential' model. The four available parameter fields are 'Lambda', 'Seed', 'Variance', and 'Nugget' with default values of 100, a random seed number, 0.9, and 0.01, respectively, for all scales.

B-I-IV Simulated Annealing

If this method is chosen, the main portion of the **Option of Unconditional Random Field** window appears, as shown in **Figure B-I-IV-1**.

The user may subsequently select in the **Model** area which **Anisotropic** model to employ. The options are **Gaussian**, **Exponential** (the default for all scales), **Spherical**, **Hole-Exponential**, or **Hole-Gaussian**.

Available fields are **LambdaX**, **LambdaY**, **Seed**, **Theoretical Variance**, **Angle** (Rotation angle around coordinates), and **Nugget**. The default values can be observed by switching between **Scale 1** and **Scale 2**. For all subsequent scales, the default lambda values increase by an order of magnitude.

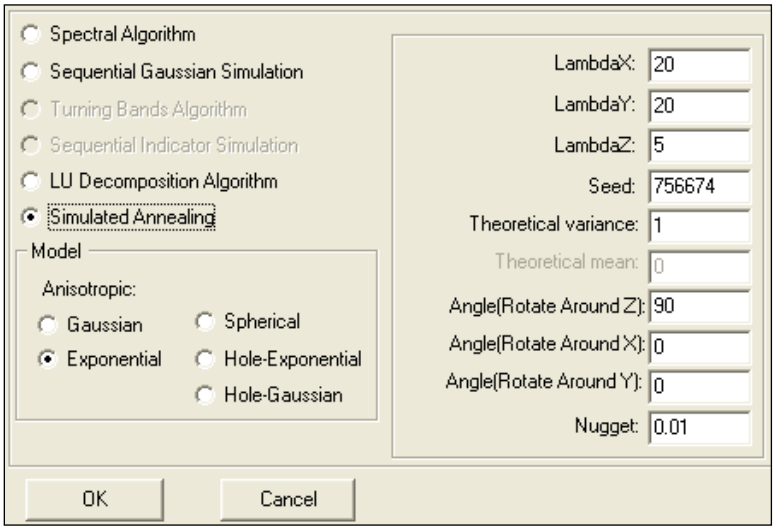


Figure B-I-IV-I Simulated Annealing Options

B-II Random Parameters Window

The **Random Parameters** window is shown in **Figure B-II-1**.

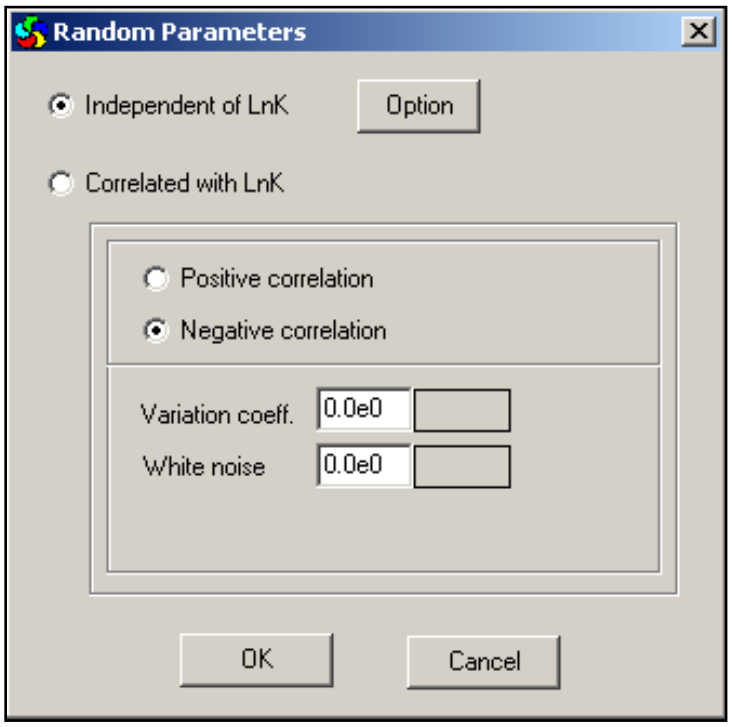


Figure B-II-1 Random Parameters

In the **Random Parameters** window the user may make two distinctions as to the relation to ln K:

1) Independent of LnK
2) Correlated with LnK

Independent of LnK is the default selection. This selection sets the target parameter as being completely independent of the conductivity. If no further action is taken, the defaults from the unconditional random field options are used. These defaults may be edited by clicking the **Option** button to open the **Option of Unconditional Random Field** window (see the **Appendix B-I**).

If **Correlated with LnK** button is selected, the user then may specify whether the relationship is a (+) or (-) . The correlations are specified by the following formula, with the plus sign applicable for the correlation and the $(-)$ sign applicable for the $(-)$ correlation:

$$x = x_m(e^{\pm \alpha f'} + w) \qquad \text{(B-II-I)}$$

$$\begin{aligned} &\text{where } x: \text{the variable of interest}\left[\text{applicable dimensions}\right], \\ &\quad x_m: \text{the mean of the variable of interest}\left[\text{applicable dimensions}\right], \\ &\quad a: \text{the correlation factor}\left[T/L\right], \\ &\quad f': lnK\left(\text{the natural log of conductivity}\right), \left[L/T\right], \text{and} \\ &\quad w: \text{white noise value}\left[\text{dimensionless}\right] \end{aligned}$$

The user specifies the coefficient of variation (in the 'Variation coeff.' Field). This is defined as the standard deviation divided by the mean value $\left(\sigma_x / x_m\right)$, and is used to determine the α value.

The user also specifies the white noise value. This value (typically the standard deviation) then sets the upper boundary for the distribution of the randomly selected white noise values, while the lower bound is set as the $(-)$ of this value). The white noise exhibits a normal distribution.

For either of these, the user may set fields to reflect absolute values (leaving the units list field blank – default), or to relative values (by setting the units list field to '%').

Therefore, for each f' value that exists in the model, an x-value is calculated from **Equation B-II-I**, given the software-determined α value, a randomly selected w value (selected by the computer and based on the distribution), and the user specified x_m value.

Clicking the **OK** button closes the window and sets the changes in the software. Clicking the **Cancel** button closes the window and discards any changes.

APPENDIX C: TRANSIENT SETTINGS

This appendix explains the windows that may be encountered when specifying transient settings for features in IGW Version 5.0P.

C-I The Transient Settings Window[29]

The **Transient Settings** window is shown in **Figure C-I-1**.

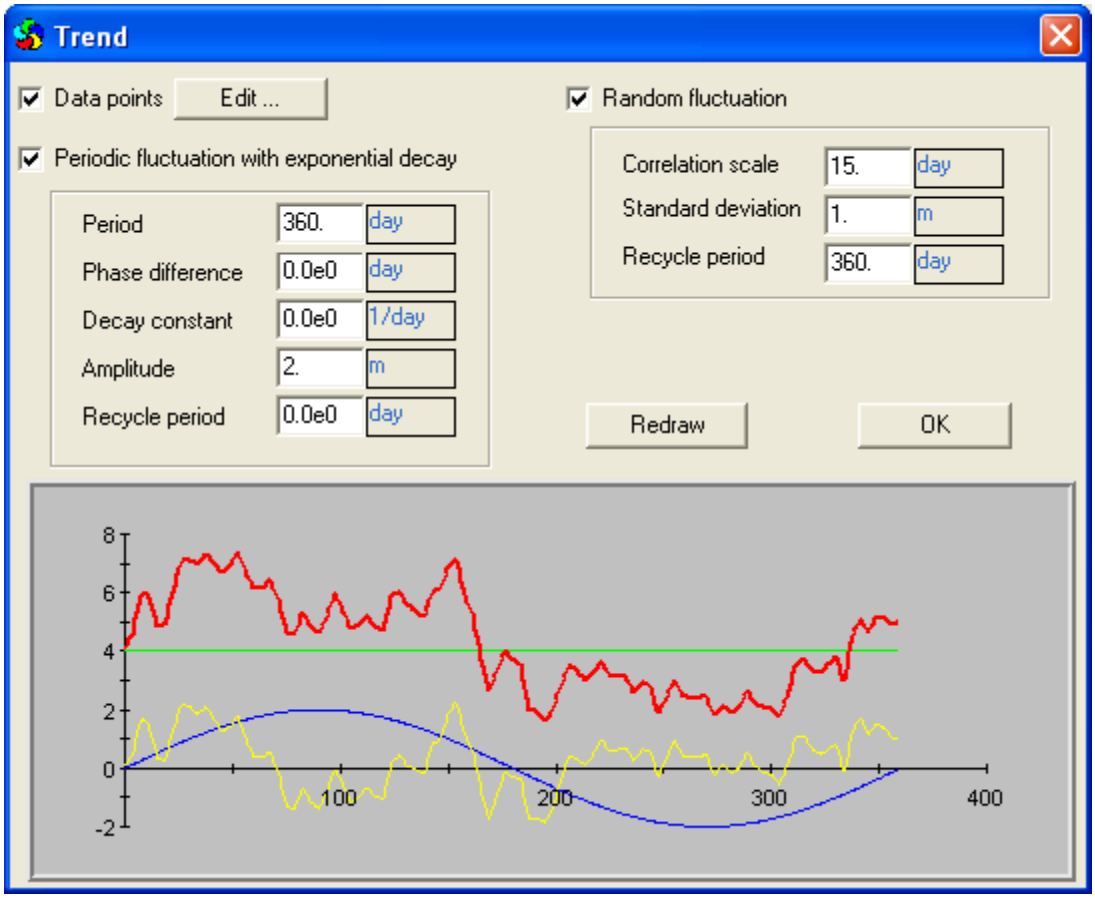


Figure C-I-1 Transient Settings

There are three parameters the user can manipulate with respect to transient conditions:

1) Data points,
2) Periodic fluctuation with exponential decay, and
3) Random fluctuation.

By default, all three are active (the boxes next to the titles are checked).

The data points can be edited by clicking the **Edit** button next to **Data points**. This opens the **Data Points** window discussed in **Appendix C-II**.

[29] Refer to **Chapter 17** of the ***IGW Version 5.0P Tutorials*** document for examples of utilizing this interface.

The settings associated with **Periodic fluctuation with exponential decay** and **Random fluctuation** areas are discussed in **Table C-I-I**.

Table C-I-I Fields Associated With Periodic and Random Fluctuations

GROUP	ATTRIBUTE	DEFINITION	DEFAULT VALUE / ENTRY	DEFAULT UNIT	OTHER AVAILABLE UNITS
Periodic fluctuation with exponential decay	Period	The amount of time it takes for the periodic fluctuation to repeat	360	day	hour, sec, month, year
	Phase difference	The amount of time the start of the periodic cycle is offset from day 0	0	day	hour, sec, month, year
	Decay constant	A value quantifying how quickly the periodic fluctuation attenuates	0	1/day	1/hour, 1/sec, 1/month, 1/year
	Amplitude	The greatest amount of fluctuation experienced in the periodic cycle	2	m	cm, ft, inch
	Recycle Period	The amount of time it takes for the periodic fluctuation to reset (independent of period)	0	day	hour, sec, month, year
Random fluctuation	Correlation scale	The segment of time that each random point should be correlated with	15	day	hour, sec, month, year
	Standard deviation	A measure of the magnitude of deviation	1	m	cm, ft, inch
	Recycle Period	The amount of time it takes for the random fluctuation to reset	360	day	hour, sec, month, year

Clicking the **Redraw** button updates the visualization of the trend and fluctuations in the preview pane (the lower half of the **Transient Settings** window). Clicking the **OK** button saves any changes made and closes the window.

C-II The Data Points Window

The **Data Points** window is shown in **Figure C-II-I**.

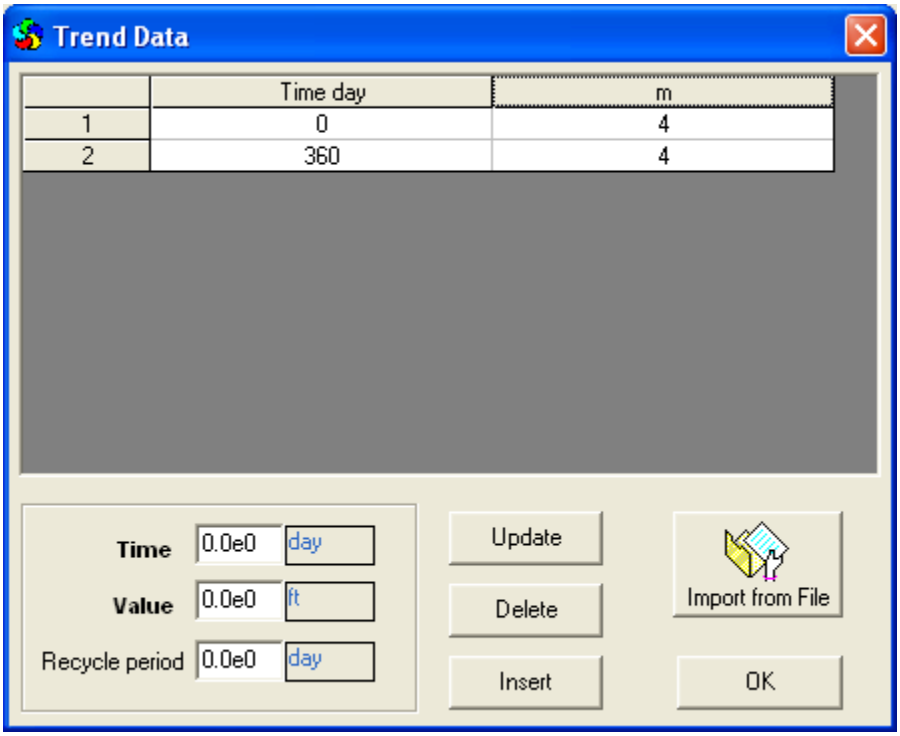


Figure C-II-I Data Points

The upper portion of the window lists the data. The left-hand column lists each datum's index, the **Time day** column gives its time value in days, and the **m** column gives its value in meters.

The lower portion of the window is for data entry.

To select a specific datum click on any of its columns in the upper portion of the window. The lower portion of the window updates to show the current values.

Clicking the **Insert** button inserts a copy of the current datum at that point (forcing the current one down the list).

Clicking the **Delete** button deletes the current datum.

To edit a datum, simply enter new values in the **Time**, **Value** and/or **Recycle Period** fields (and select desired units) then click the **Update** button.

Import from File button imports cumulative data points in *.csv format, instead of entering the values manually.

When all desired changes have been made, click the **OK** button to close the **Data Points** window and return to the **Transient Settings** window**.**

APPENDIX D: IGW VERSION 5.0P PLOT TYPES

IGW Version 5.0P offers a number of plot types available for displaying model results. These plot types are discussed in the following sections.

D-I Zone Mass Balance Plots

There are two main plots that are available with respect to zones. The first is the **Water Balance** window. It displays water mass balance data and is discussed in **Section D-I-I**. The second is the **Plume Mass Balance** window. It displays contaminant mass balance data and is discussed in **Section D-I-II**.

D-I-I The Water Balance

The **Water Balance** window is shown in its initial state in **Figure D-I-I-1**. Please refer to **Section 7.6.2.4** for information on how to activate **Water Balance** window.

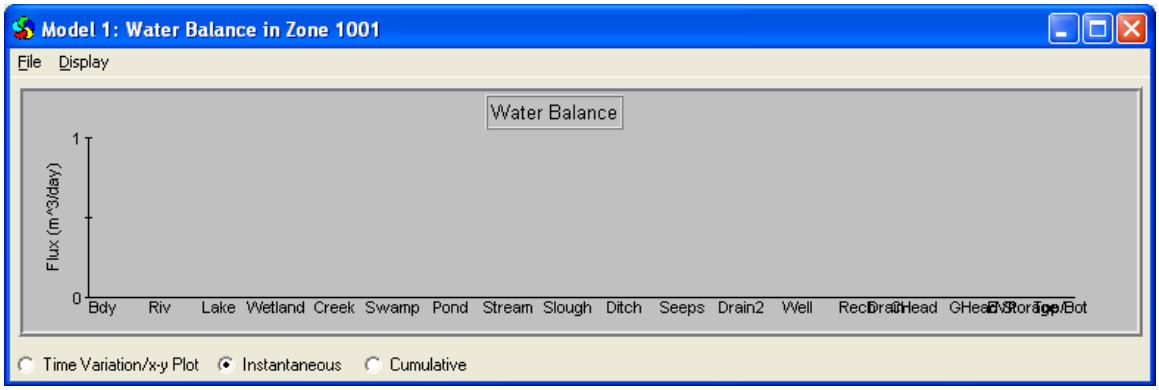


Figure D-I-I-1 Water Balance Window at Initial State

There are three display types for the plot in this window (chosen at the bottom of the window):

1) Time Variation/x-y plot,
2) Instantaneous (default), and
3) Cumulative.

These plots are discussed in the following subsections.

On the menu bar in this window, the user may access the **File** menu and subsequently choose **Export Picture** or **Close**.

Placing the cursor on **Export Picture** opens a cascading menu with **As BMP** or **As JPEG**. Selecting either opens the **Save As** window in which the user may select the path and type in a filename for the picture of the window that is to be saved. The software saves the file in the previously chosen format.

Choosing **Close** simply closes the window.

The user may also access the **Display** menu and subsequently choose **Change Draw Property…** option. This opens the **2D Chart Control Properties** window in which the user may customize the display of the water balance chart area. Details are given in **Section 14.3**. The user may also open this window by right-clicking in the window area.

THE TIME VARIATION/X-Y PLOT

The **Time Variation/x-y Plot** shows the flux plotted on the y-axis versus time plotted on the x-axis as a continuous variation. An example is shown in **Figure D-I-I-2**.

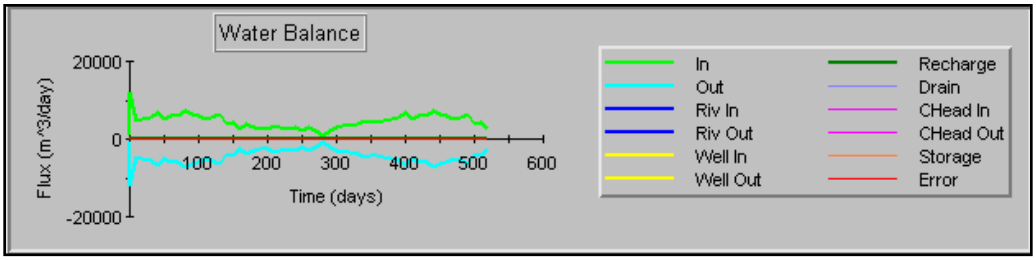


Figure D-I-I-2 An Example of the Time Variation/X-Y Plot

THE INSTANTANEOUS PLOT

The **Instantaneous** plot shows the instantaneous flux plotted on the y-axis for each feature type defined on the x-axis. An example is shown in **Figure D-I-I-3**.

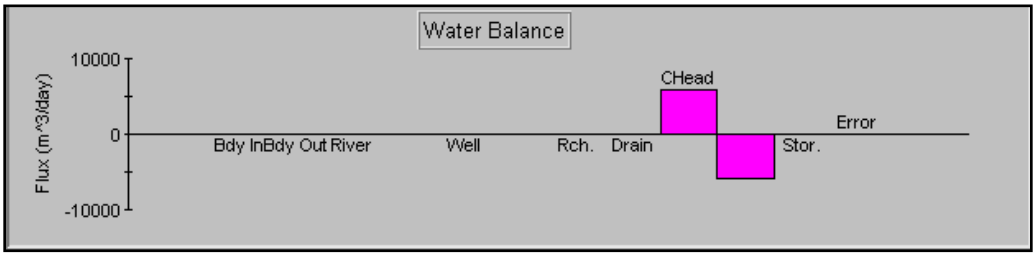


Figure D-I-I-3 An Example of the Instantaneous Plot

THE CUMULATIVE PLOT

The **Cumulative** plot is similar in display to the **Instantaneous** plot (above) except that it shows the cumulative flux (measured from the time the monitoring was activated – not when the plot window is opened) instead of the instantaneous flux.

D-I-II The Plume Mass Balance

This window functions in the same manner as the **Water Balance** window (see **Section D-I-I**) except it monitors contaminants. It is opened by checking the **Plume Mass Balance** checkbox in the lower left window of Attributes Explorer. **Zone Budget** checkbox should be selected before activating the plume mass balance. Refer to **Section 7.6.2.4** for further details.

D-II Well Monitoring Plots

There are two main plots that are available with respect to wells. The first is the **Time Process** window. It displays temporal head or concentration data and is discussed in **Section D-II-I**. The second is the **Probability** window. It is available when stochastic modeling is enabled and is discussed in **Section** Error! Reference source not found..

D-III-I The Time Process for Wells

The **Time Process** window is shown in its initial state in **Figure D-III-I-1**. It is opened by double-clicking on a **Time Process** entry in the TPS. An entry will only appear after a well has been set to **Monitoring Head and Concentration** in the Attributes Explorer (**Section 9.4.2** and Error! Reference source not found.).

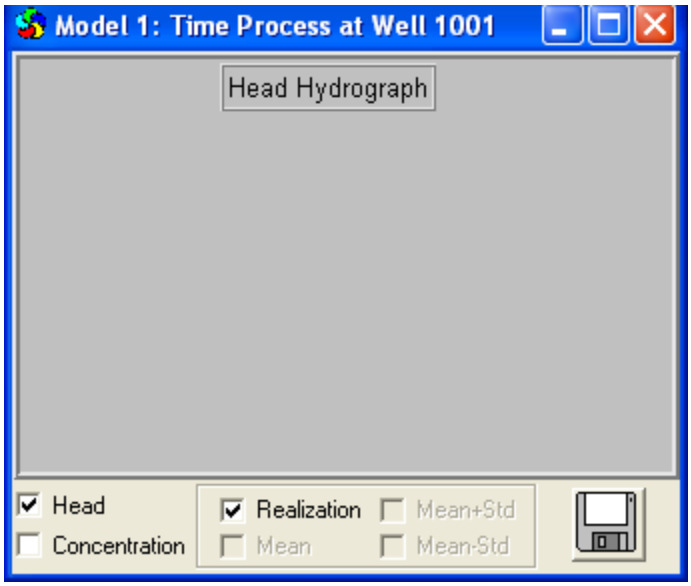


Figure D-III-I-1 Head Hydrograph at Initial State

There are two display options for the plot in this window (chosen at the bottom left-hand corner of the window:

1) Head, or
2) Concentration.

If **Head** is chosen, the plot area will show a **Head Hydrograph** curve in which the head in the well is plotted on the y-axis and time is plotted on the x-axis. An example is shown in **Figure D-III-I-2**. If **Concentration** is chosen, the plot area will show a **Concentration Breakthrough** curve in which contaminant concentration in the well is plotted on the y-axis and time is plotted on the x-axis. The plot appears similar to that shown in **Figure D-III-I-3** (except the title reads **Concentration Breakthrough**).

Notice that the **Realization** box is checked. This indicates that the software will plot the current model solution. If this is unchecked, the plot will appear empty. The other options in this area are specific to stochastic modeling and are discussed in **Section** Error! Reference source not found..

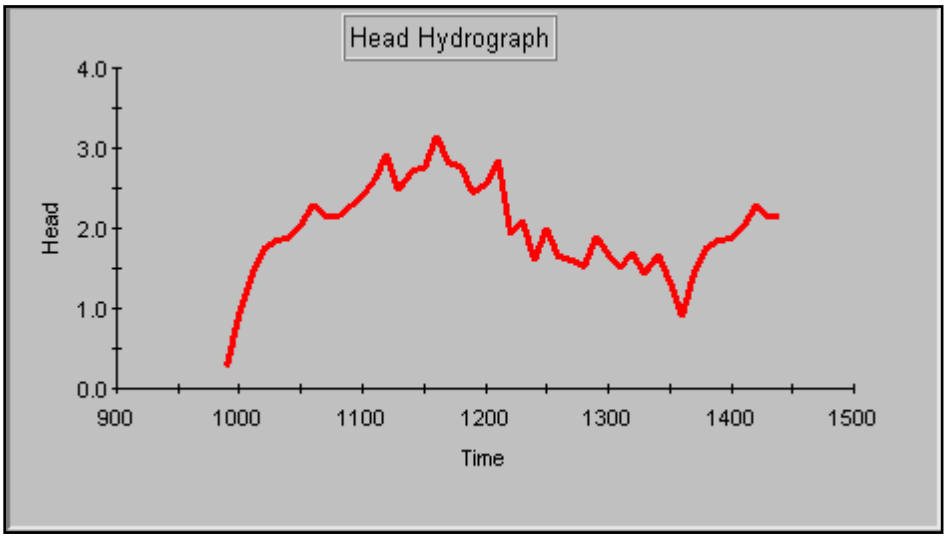


Figure D-III-I-2 An Example of the Head Plot

Using the list in the lower left field of Attributes Explorer, the user may select which model the plot will be associated with if the well exists within the boundaries of a submodel. The default is **Main Model**. The field will list all available options. After changing models, click the **Refresh** button to update the display.

> **I** All IGW plots update continuously as the model solution proceeds.

Clicking the disk button begins the process of exporting the data displayed in the window by opening the **Save As** window. This is discussed in **Section 20.4**. If head or concentration data was defined in the **Input Head / Concentration Data** window (**Section 9.4.2**), they will appear in the respective plot types as discrete points.

> **I** The plots will show a higher resolution if they are resized to have larger dimensions.

D-III-II Other Well Plots

Refer to **Section** Error! Reference source not found. for more information concerning stochastic odeling well plots.

D-IV Changing Plot Displays

IGW Version 5.0P data plots may have their display characteristics changed by the user. These characteristics are controlled through the **2D Chart Control Properties** window (see **Figure D-IV-1**). This window is common to all of the plots, however the settings are independent of each other. It is accessed by right-clicking in the plot area of the respective window (some plots may provide additional alternate methods of accessing the window). Guidance in changing the settings is available by clicking the **Help** button in the lower left-hand corner of the window. When clicking, a help screen will appear that is specific to the layer in the window that is currently active.

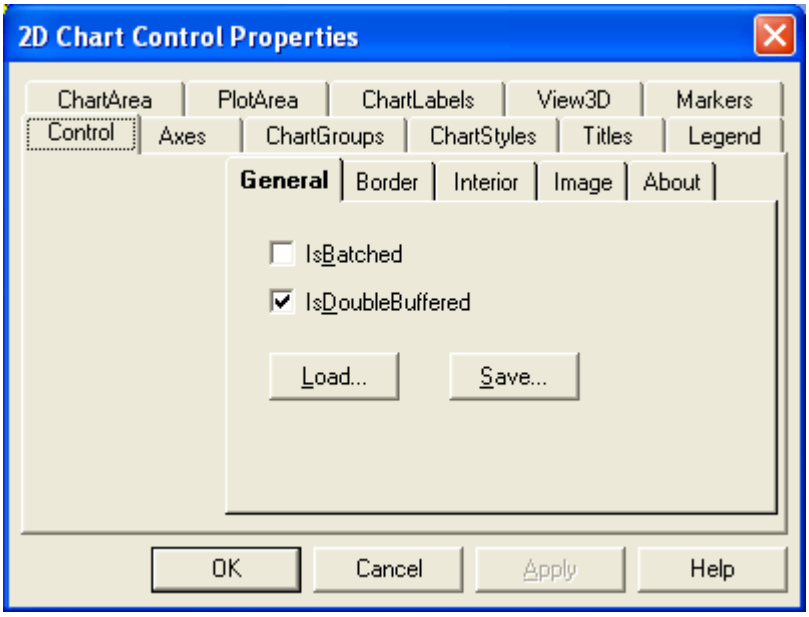


Figure D-I-I-2 2D Chart Control Properties

APPENDIX E: THE DATA ANALYSIS WINDOW[30]

This appendix describes the **Data Analysis** window that displays statistical information for a set of scatter point attributes. Its default view is shown in **Figure E-I-1**.

I The windows in this section show graphs and numbers that are associated with an example set of scatter point attributes, and therefore will not necessarily reflect those seen by the user.

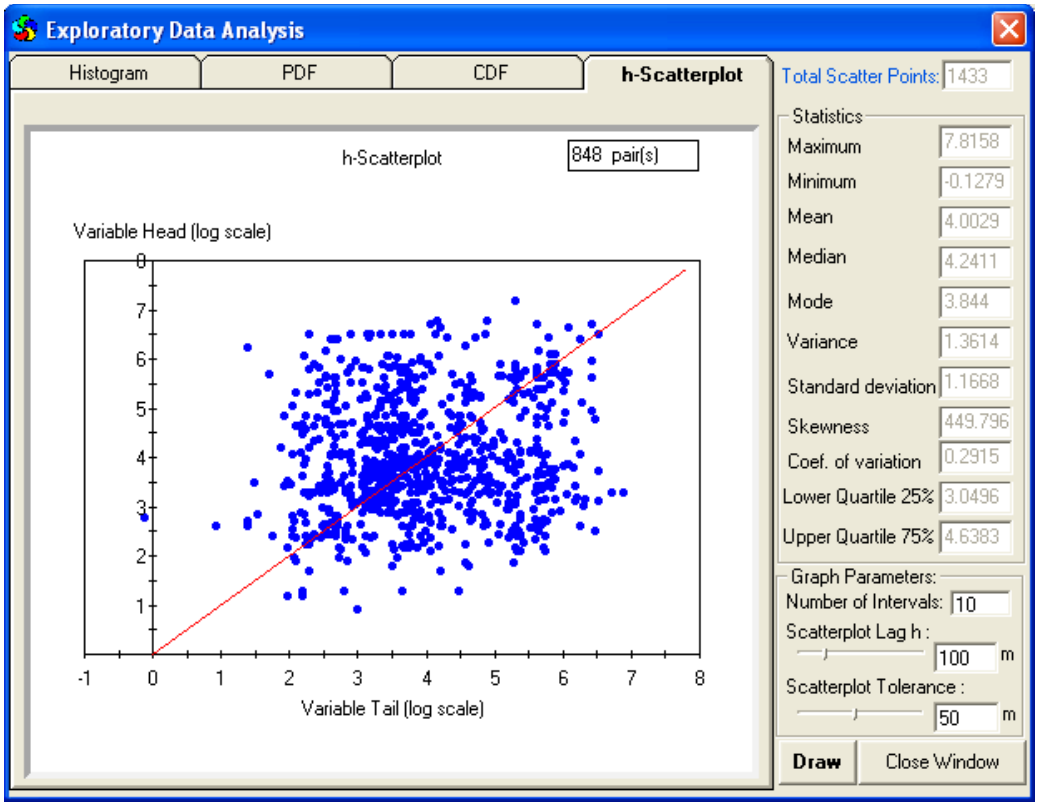


Figure E-I-1 Data Analysis

The window has two main sections:

1) **Plot Pane** (PP), the multi-layered left-hand portion of the window, and
2) **Statistics Pane** (SP), the collection of fields on the right-hand side of the window. These panes are discussed in the following sections.

[30] The user should consult the ***IGW Version 5.0P Reference Manual*** and/or a statistics textbook for more detailed information.

E-I The Plot Pane (PP)

When the **Data Analysis** window is first opened, the **h-Scatterplot** layer is displayed in the PP. The other available layers include:

1) **Histogram**,
2) **Probability Distribution Function (PDF)**, and
3) **Cumulative Distribution Function** (CDF). These layers are discussed in the following subsections.

E-I-I The H-Scatterplot Layer

The **H-Scatterplot** layer is visible in **Figure E-I-1**.

It shows a log-scale scatterplot that displays every data pair (falling within the **Scatterplot Lag h** distance, see **Section E-II-II**) for the variable at hand (redundant points are automatically excluded). The red line shows the 1 to 1 relationship.

E-I-II The Histogram Layer

The **Histogram** layer shows a histogram that displays the total number data with a specific value (over the entire value range).

E-I-III The PDF Layer

The **PDF** layer shows the PDF for the specific variable. The **PDF** shows the probability of encountering a datum in the set with a value in a specific range.

E-I-IV The CDF Layer

The **CDF** layer shows the CDF for the specific variable. The **CDF** shows the probability of encountering a datum in the set with a value that is less than or equal to a given value.

E-II The Statistics Pane (SP)

The **SP** can be seen in **Figure E-I-1**. The top field of the **SP** is labeled **Total Scatter Points** and lists exactly how many data (scatter points) are defined in the model.

At the bottom of the **SP** there are 2 buttons: **Draw** and **Close Window**. The **Draw** button allows the user to update the display of the PP graphs to reflect any changes made in the **Graph Parameters** area (see **Appendix E-II-I**). The **Close Window** button closes the **Data Analysis** window.

The **Statistics** and **Graph Parameters** areas are discussed in the following subsections.

E-II-I The Statistics Area

There are a number of parameters listed in the **Statistics** area. A brief definition of the parameters is presented below:

- MAXIMUM: The greatest value encountered in the data set.
- MINIMUM: The smallest value encountered in the data set.

- MEAN: The arithmetic mean of the values in the data set.
- MEDIAN: A value that divides the data set such that an equal number of data have values greater than and less than it.
- MODE: The most frequently occurring value in the data set.
- VARIANCE: This is the average squared difference of the observed values from their mean.
- STANDARD DEVIATION: The square root of the variance.
- SKEWNESS: It is the average cubed distance between the data values and their mean. It is a parameter used to quantify the shape of the value distribution.
- COEFFICIENT OF VARIATION: This is another parameter used to quantify the shape of the value distribution. It is defined as the standard deviation divided by the mean.
- LOWER QUARTILE 25%: Similar to the median except this is the value range that divides the data set such that ¾ of the data have values greater than it while ¼ of the data have values less than it.
- UPPER QUARTILE 75%: This is the value range that divides the data set such that ¼ of the data have values greater than it while ¾ of the data have values less than it.

E-II-II The Graph Parameters Area

In the **Graph Parameters** area, the user may adjust settings that affect the plot and statistical analysis of the data. They are briefly described in the following subsections.

NUMBER OF INTERVALS

This value sets the number of intervals that the data is grouped into when displayed in the **PDF**, **CDF**, and **Histogram**. This number will also slightly affect the value of the Median, Mode, and Quartile values (discussed above). This is due to the fact that these values are calculated based upon the groupings of the data.

SCATTERPLOT LAG H

This distance sets the range at which the data correlation analysis should occur. The default value is 100 m. This implies that the correlation analysis will be done for all pairs that are 100 m apart. The tolerance is set at the value divided by 2 (or in the default case, 50 m). Instead of only analyzing the pairs that are exactly the lag distance apart (which realistically will be very few), the analysis is performed for all pairs that lie within the lag distance, plus the tolerance and the lag distance minus the tolerance (in the default case, between 150 and 50 m).

E-II-II The Graph Parameters Area

APPENDIX F: SPATIAL STATISTICS PARAMETERS WINDOWS

This appendix discusses the interface windows that are encountered when defining scatter point kriging or simulation settings.

F-I The Input Parameters Window

The 'Input Parameters' window is used to manually define the spatial statistics when kriging is being implemented. An example of the default view of the window is shown in **Figure F-I-1**.

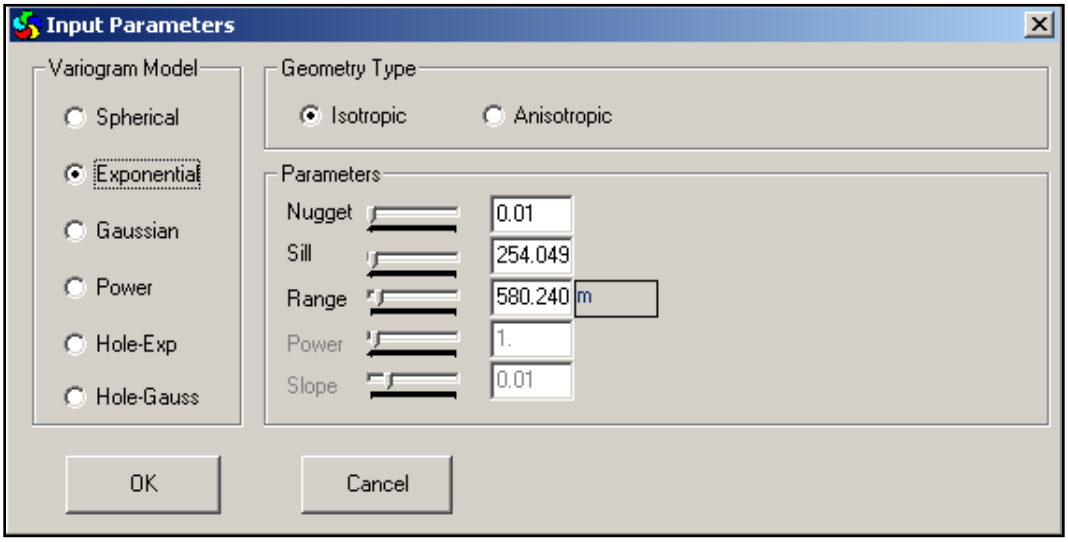


Figure F-I-1 An example of the default view of the 'Input Parameters' window

When the window is opened, the software supplies the initial values in the fields based upon an automatic variogram analysis[31].

In the 'Variogram Model' area, the user has the option of 'Spherical', 'Exponential' (the default), 'Gaussian', 'Power', 'Hole-Exp', and 'Hole-Gauss'. 'Power' employs all but the 'Sill' parameter. The rest employ only the top three parameters.

In the 'Geometry Type' area, the user may specify whether to use an 'Isotropic' (the default) or 'Anisotropic' model form. If 'Anisotropic' is selected, the window updates to show an extra range field and an angle field. The modified window is shown in **Figure F-I-2**.

In the 'Parameters' area, the user may adjust the settings for 'Nugget', 'Sill', 'Range' (or 'Range1' if anisotropic), 'Power', and 'Slope' (and 'Range2' and 'Angle' if anisotropic). These settings may be adjusted by entering specific numbers in the fields, or by using the slide bars that appear between the field and the field name. Parameters not available for a specific model will be displayed in gray text and not be accessible. The parameter definitions are discussed in the ***IGW Version 5.0P Reference Manual.***

[31] The variogram analysis is shown in the 'Variogram' window (see **Appendix F-II**).

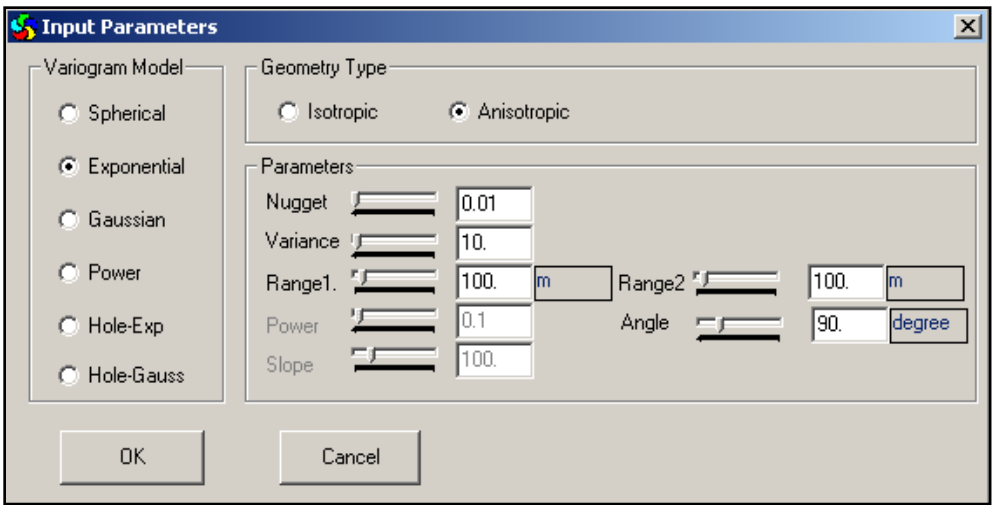


Figure F-I-2 Modified View of 'Input Parameters' Window

Clicking the 'OK' button closes the window and sets the changes in the software. Clicking the 'Cancel' button ignores any changes and closes the window.

F-II The Variogram Window

The 'Variogram' window is used to view and edit the software determined variogram parameters. An example of the default view of the 'Variogram' window is shown in **Figure F-II-1.**

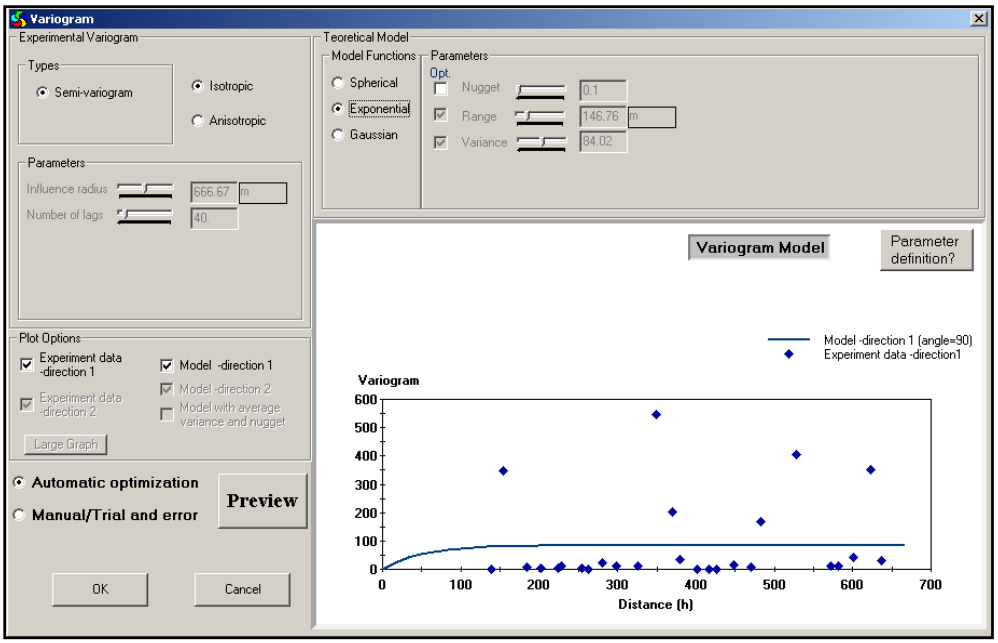


Figure F-II-1 A Sample of the Variogram Window

The 'Variogram Model' portion of the window (the large white area with the plot) shows the current model shape with respect to the data. Clicking the 'Parameter definition?' button in this area opens a window that graphically describes a number of parameters that are utilized in this window[32].

[32] All of the parameters are fully described in the ***IGW Version 5.0P Reference Manual***.

Notice that in the lower left-hand corner of the 'Variogram' window, the user has the option of selecting either 'Automatic optimization' or 'Manual/Trial and error'. By default, 'Automatic optimization' is selected and most parameters are determined automatically by the software. Even so, the user may still define whether the model is 'Isotropic' (the default) or 'Anisotropic' (in the 'Experimental Variogram' area) and whether to employ a 'Spherical', 'Exponential' (the default) or 'Gaussian' model type in the 'Model Functions' area (in the 'Theoretical Model' area). Also, by default, 'Nugget' is not clicked in the 'Theoretical Model' – 'Parameters' area. The user can manually adjust the nugget value, or choose to check the box and set it to be determined automatically. After any of these settings have been adjusted, the user should click the 'Preview' button to update the 'Variogram Model' display.

If the user selects 'Manual/Trial and error', all of the other model parameters (in addition to those previous) are available to be adjusted. This includes those in the 'Parameters' area within the 'Experimental Variogram' area ('Influence of radius' and 'Number of lags' [and 'Principal Angle', 'Bandwidth', and 'Angle Tolerance' when anisotropic is selected]), and those in the 'Parameters' area within the 'Theoretical Model' area ('Nugget', 'Range', and 'Variance'). Note that when anisotropic is selected, this area will display additional fields for the second direction values for these settings (adjustable) along with the average of the two directions (not adjustable).

The 'Plot Options' area provides for control of the display in the 'Variogram Model' area. The user may choose to deactivate (uncheck) or activate (check) the display of the data ('Experiment data –direction 1') and the model ('Model –direction1') as desired. If anisotropic is selected, the user may also adjust the display for the second direction ('Experiment data –direction 2' and 'Model –direction 2') and the average ('Model with average variance and nugget'). The 'Large Graph' button can be clicked to open a separate larger window with these plots displayed in it.

Clicking the 'OK' button sets the changes in the software and closes the window. Clicking the 'Cancel' button closes the window and discards the changes.

F-III The Random Field Option Window

The 'Random Field Option' window is used to manually define the spatial statistics when unconditional or conditional simulation is being implemented. The window will appear slightly differently depending on which method is chosen in the 'Simulation Methods' area in the Alternate RHP (**Section** Error! Reference source not found.). The three window variations are hown in **Figure F-III-1**, **Figure F-III-2**, and **Figure F-III-3**.

For all three windows, software supplies the initial values in the fields based upon an automatic variogram analysis[33].

Also, clicking the 'OK' button closes the window and sets the changes. Clicking the 'Cancel' button discards the changes and closes the window.

Refer to **Appendix B-I-I** for a discussion of the 'Spectral Algorithm' parameters. Refer to **Appendix B-I-II** for a discussion of the 'Sequential Gaussian Simulation' parameters. Refer to **Appendix B-I-III** for a discussion of the 'Turning Bands Algorithm' parameters.

[33] The variogram analysis is shown in the 'Variogram' window (see **Appendix F-II**).

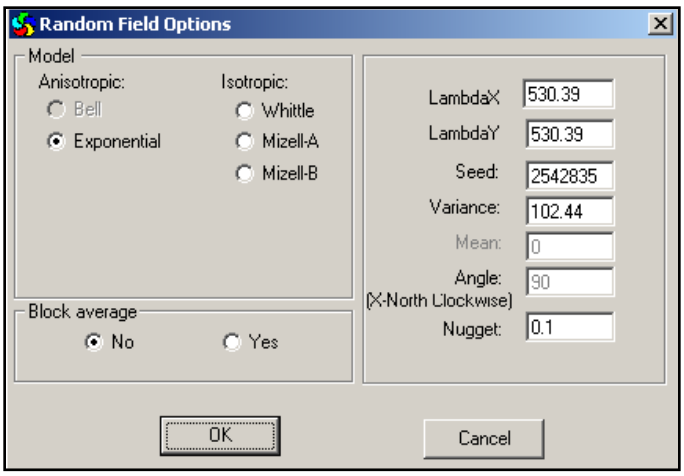


Figure F-III-1 **The 'Random Field Options' Window – Spectral Algorithm Variation**

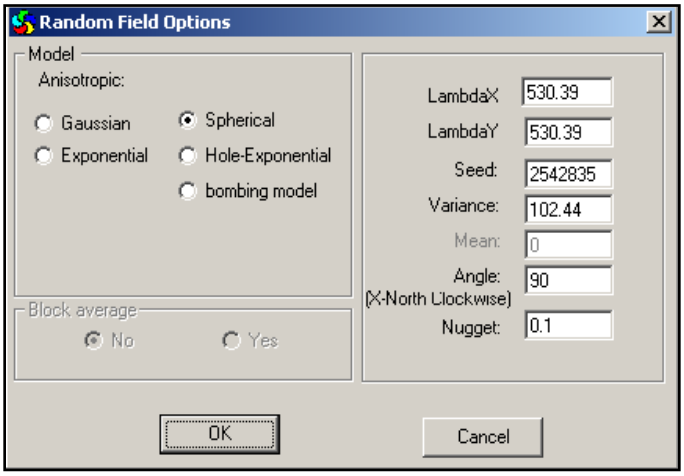


Figure F-III-2 **The 'Random Field Options' Window – Sequential Gaussian Simulation Variation**

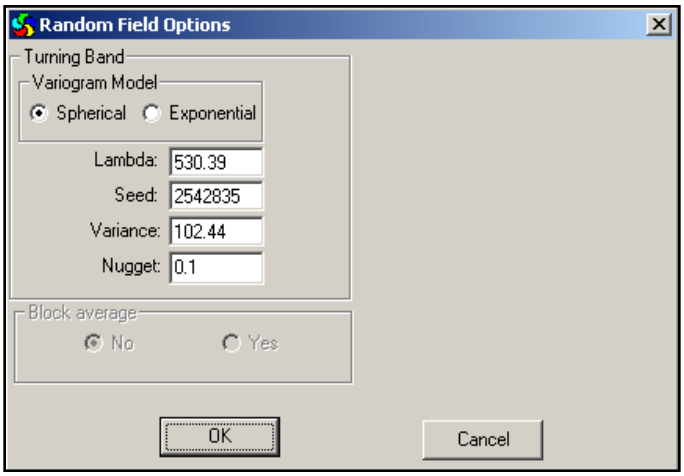


Figure F-III-3 **The 'Random Field Options' Window – Turning Bands Algorithm Variation**

REFERENCES