USER'S GUIDE
3D VISUALIZATION SOFTWARE FOR PMWIN AND RELATED GROUNDWATER FLOW AND TRANSPORT MODELS

Report to the
WATER RESEARCH COMMISSION

by

Wen-Hsing Chiang and Meintjies Bekker

Institute for Groundwater Studies
University of the Free State

WRC Project No 1007/1/01
ISBN 1 86845 782 6

JULY 2001

Acknowledgements

The research in this report emanated from a project funded by the Water Research Commission entitled:

3D Visualization Software for PMWIN and related Groundwater Flow and Transport Models

The Steering Committee responsible for the project consisted of the following persons:

Mr K.C. Pietersen	Water Research Commision (Chairman)
Prof B.E. Kelbe	Department of Hydrology, University of Zululand
Mr S. Adams / R. Titus	Dept. of Earth Sciences, University of the Western Cape
Mr J.C. Conrad	Environmentek, CSIR (Stellenbosch)
Dr H.J. van der Spuy	Water Research Commission (Committee Secretary)
Dr G.C. Green	Water Research Commission
Mr W.E. Bertram	Department of Water Affairs & Forestry, Directorate: Geohydrology

The financing of the project by the Water Research Commission and the contributions of members of the Steering Committee are hereby gratefully acknowleged.

The authors would also like to take this opportunity to express the sincere thanks to the following institutions and people, without whose help the research could not have been carried out:

The University of the Free State for facilities provided conducting the research.
The Institute for Groundwater Studies, in particular the Director, Prof F.D.I. Hodgson, and Prof G.J. van Tonder for their assistance and encouragement during the research and Me C. Bitzer for editing of the report.

Executive Summary

1 Background and Motivation

It is well known that groundwater is one of the most important water resources in South Africa. However, it is a difficult task to locate points of abstraction and to abstract groundwater from its environment. It is imperative for these points of abstraction to be well managed in order to guard against dewatering and to prevent the pollution of groundwater as a whole. Groundwater flow and transport models constitute an important tool for the management of these points and ultimately, the conservation and management of this valuable fresh water resource. However, groundwater flow and transport models require and produce a large amount of hydrogeological data and the human brain cannot possibly grasp all that information without the help of modern technology such as 3D-Visualization.

Taking advantage of the natural abilities of the human vision system, 3D visualization is often the tool of choice for understanding and communicating conceptual models, verifying model input, understanding model output, explaining and communicating conclusions and recommendations, and motivating expenses. A 3D visualization software has therefore been developed to enhance conceptualisation of model data**.** The software is based on the groundwater modeling system Processing MODFLOW (Chiang and Kinzelbach, 2001) and is capable of using results from existing groundwater flow and solute transport models, viz. MODFLOW (McDonald et al, 1988), MT3DMS (Zheng and Wang, 1998), PHT3D (Prommer, 2000) and RT3D (Clement, 1997). As these computer models are widely used by local and international groundwater researchers, the present 3D Visualization software could certainly make a great contribution.

2 Statement of Objectives

The objective of this research project is to enable 3D-Visualization from the following data:

- 3D PMWIN Output:

Processing Modflow for Windows (PMWIN) is an integrated system for modeling groundwater flow and solute transport with the USGS groundwater flow model MODFLOW, the particle tracking model PMPATH (Chiang and Kinzelbach, 1995), the solute transport model MT3DMS (Zheng and Wang, 1998) and the parameter estimation program PEST (Doherty, 1994).

Model output used by the 3D-Visualization software are simulated groundwater levels, pollution plumes and the spatial distribution of model parameters (T and S values).

- 2D AQUAMOD for Windows

  The 2D finite element model was sponsored by the WRC and is used by many groundwater modelers in SA. Visualization of the following parameters is possible: simulated water levels, pollution plumes and the spatial distribution of model parameters.
- Water level contours; surface contours; geological surfaces

  3D Visualization of the interpolated groundwater levels as well as the surface topography will be possible. Visualization of geological logs and geophysical data are also helpful in better understanding an aquifer

3 Summary of Methods and Results

All the objectives of this project were met. The geohydrological objects, which can be displayed by Groundwater Explorer, are given in the following list:

- **Geospatial model** displays the topography of the aquifer surface as well as the aquifer structure.
- **Potentiometric Surface** displays the model-calculated groundwater surface.
- **XYZ-Surface** uses xyz data files (such as result files from AUQAMOD) to display surfaces.
- **Isosurface** uses the concentration values calculated by MT3DMS, RT3D and PHT3D to display pollution plumes in the form of isosurfaces (i.e., surfaces of the same concentration values).
- **Hydraulic Components** such as wells, drains, rivers or boundary conditions used in a MODFLOW model can be displayed.

- **Parameters** displays the distribution of all geohydrological parameters used in a MODFLOW model, such as transmissivity T, storage coefficient S, or hydraulic conductivity K.

The 3D-Visualization software, called Groundwater Explorer, is included in the companion CD-ROM of this report.

Groundwater Explorer is written in the C++ programming language. The Microsoft Visual C++ (version 6.0) and the Microsoft Visual Studio have been used as the programming environment. The Visualization Toolkit (Schroeder et al., 1998b), a C++ class library for visualization, was used by the Groundwater Explorer to render 3D geohydrological objects.

4 Conclusions

For the purposes of management and conservation of groundwater, the recommendations derived from groundwater models are communicated and explained to the responsible managers. However, managers are not always familiar with geohydrological concepts and groundwater models. As a result it is often difficult for managers to understand the recommendations in whole. The same problem presents itself in motivating the public.

Taking into account of the human visual ability, 3D-Visualisation is one of the best tools for helping people understand model input/output and helping professionals to give explanations. In this regard, Groundwater Explorer will play an important role in the application of groundwater models and will be a good supplement to Processing Modflow, which is included in the book “3D-Groundwater Flow Modeling with PMWIN” (Chiang and Kinzelbach, 2001).

The present version of Groundwater Explorer is linked to Processing Modflow directly, as the later has been developed over more than 10 years and has been used by thousands of groundwater modellers world-wide. As stated in Recommendations below, Groundwater Explorer should be further developed to support the standard input/output files of MODFLOW, which is a public program developed by the U.S. Geological Survey.

5 Recommendations

As a consequence of its many advantages, 3D visualization is recommended as a supplementary tool to groundwater modelling tasks as well as to understand hydrogeological conditions.

Groundwater Explorer is merely a beginning of developing a versatile and yet sophisticated 3D-Visualization tool for groundwater modellers. It is recommended to develop the following features in future:

1. Cross-sectional planes
2. Streamlines
3. Mapping of aerial photographs (either using raster or vector graphics)
4. Animation of time-varying data, such as concentration plumes or potentiometric surfaces.
5. Construction of geological surfaces using geological logs.
6. Importing models from standard MODFLOW input files.

The list for future research can be endless because of the evolving technology, software and ideas; making visualization an exciting challenge.

6 Publication

A dissertation presented for the degree of Magister Scientiae in the Department of Computer Science and Informatics in the faculty of Science of the University of the Free State by Meintjes Bekker resulted from the research done for this project. It is entitled “3D Visualization of Data from Groundwater Flow and Transport Models” (November, 2000) and was supervised by Prof. H.J. Messerschmidt and Prof. W.H Chiang.

Contents

List of Figures

Chapter 1

Introduction

1.1 Groundwater Explorer

Groundwater Explorer (GE) is a software tool for the visualization of data from groundwater flow and transport models. The present version of GE visualizes models created by the simulation system, PMWIN (Chiang and Kinzelbach, 2001). This guide explains the different components of GE and guides the user to easily use GE. At the end of the guide a step-by-step tutorial explains how to use GE for the Wetland model.

1.2 Documentation

Formatted text in this guide is used as follows:

- A **Bold** typeface is used for menus, dialog box names and tab labels.
- An *Italic* typeface is used for folder paths and filenames.
- ....(dotted lines) are used to show where pictures were cut.

Terms used in this guide are as follows:

- Visualization Object (Figure 1.1) – An object that represents an aspect of groundwater model data in a 3D space.
- Scene (Figure 1.1) – Collection of visualization objects, visible on the display. GE can add arbitrary numbers of visualization objects of the following types to a scene. Refer to Chapter 3 for details.
    - Model Outline – The bounding box of a groundwater model.
    - Geospatial Model – Topography of each model layer.
    - Potentiometric Surface – Calculated hydraulic head values.
    - XYZ Surface – Measured values.

- Isosurface of concentration values calculated with MT3DMS (Zheng, 1990), MT3D99 (Zheng and Wang, 1999), PHT3D (Prommer, 2000) and RT3D (Clement, 1997).
- Hydraulic Components, for example pumping wells.
- Model parameter values.

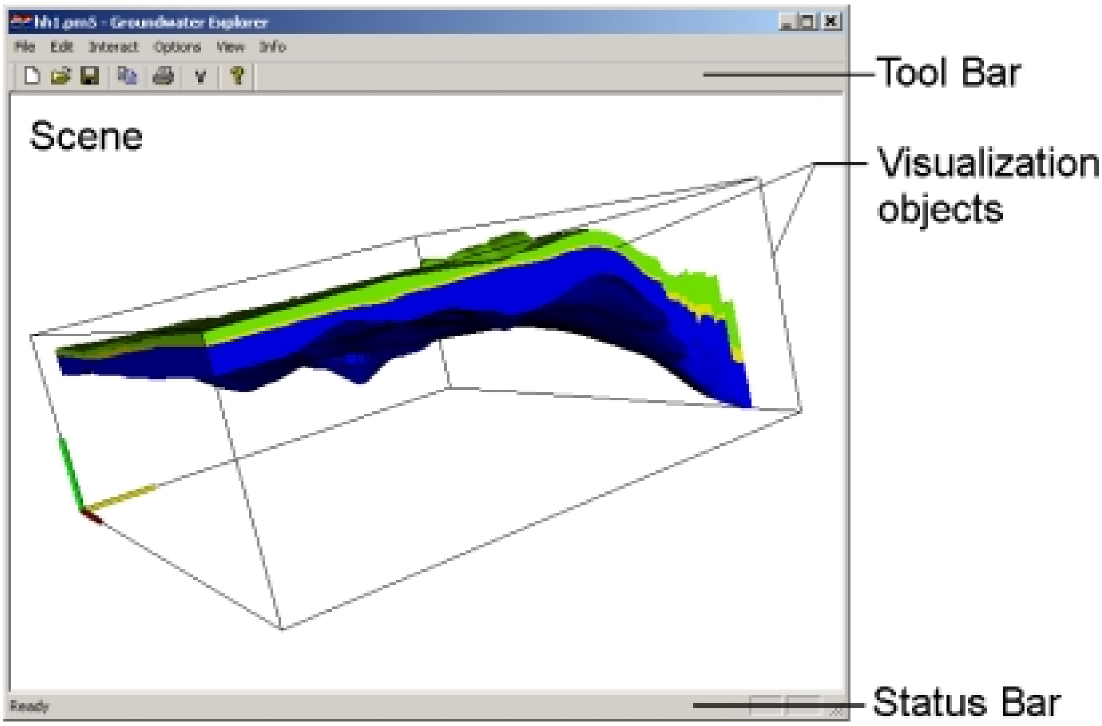


Figure 1.1: Visualization objects and scene.

1.3 System Requirements

GE runs on a personal computer (PC) with Windows 98/2000 or Windows NT as the operating system. The recommended hardware is as follow:

- Pentium III Processor
- 128 MB RAM
- 16MB AGP Graphics Adapter
- SVGA Monitor
- Mouse

1.4 Installation

Insert the GE CD into the CD-ROM drive. Use Windows Explorer to browse for the installation program *Setup.exe*, double click the file and follow the instructions to install GE.

1.5 Structure of the Following Chapters

Chapter 2 helps you to get started. Chapter 3 explains the details of the visualization objects. It also explains how to create and change the visualization objects for certain visual effects. The last chapter is a step-by-step tutorial.

Chapter 2

Getting Started

2.1 Menus

GE's menus are described in the following sections. Most of the menus are dimmed and cannot be used until a groundwater model is opened.

2.1.1 File

The **File** menu (Figure 2.1) consists of the following items:

- **New** – Clean the visualization tool's scene for a new scene.
- **Open…** – You can open a PMWIN model (with a *pm5* file extension). GE searches for all available information (input data, calculated results etc.) within the specified file folder.
- **Save as VRML…** – Displays the **Save As** dialog box (Figure 2.2) for saving the visualization scene as a VRML file. VRML stands for Virtual Reality Modeling Language. Most web browsers can display VRML files and it is a fast and easy way to put interactive visualization scenes on the World Wide Web.

  ► **To save a scene as a VRML file:**

  1. Specify the filename to be used in the **File name** box.
  2. Choose **VRML File (*.wrl)** as the **Save as type**.
  3. Click the  button. The *wrl* extension is automatically added to the given filename. In Figure 2.2 for example, the current scene will be saved as *Wetland.wrl* in the wetland folder.
- **Print…** – Prints the scene.
- **Print Preview** – Displays a preview of how the scene is going to be printed on a page.
- **Print Setup…** – Changes the print setup.
- **Exit** – Exits the program.

2.1.2 Edit

Currently the **Edit** menu contains only one item:

- **Copy** – Copies the current scene as an image to the clipboard. The image can be pasted into most word- or graphics-processing software environments by using Ctrl+V.

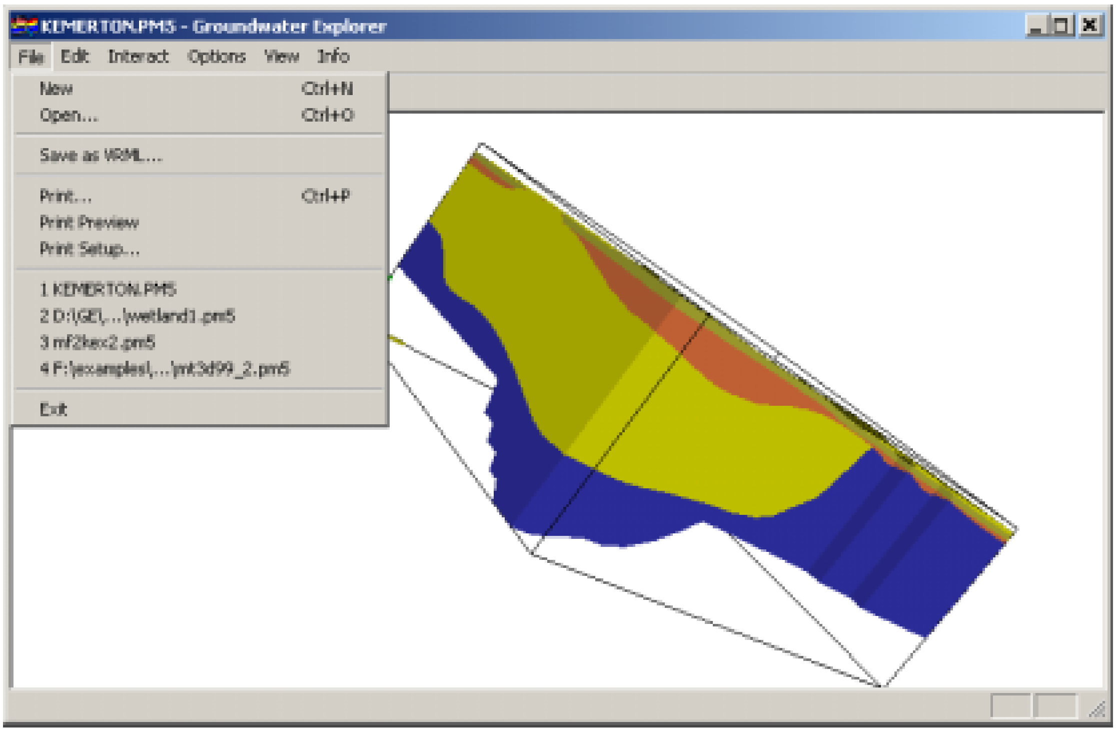


Figure 2.1: The **File** menu.

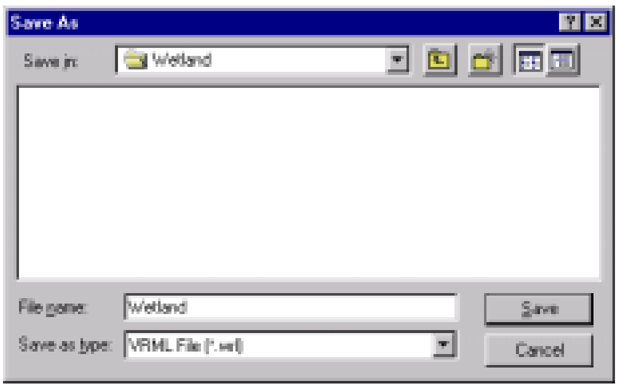


Figure 2.2: The **Save As** dialog box for saving VRML files.

2.1.3 Interact

The **Interact** menu (Figure 2.3) groups items that have an influence on all visualization objects and contains the following menu items:

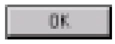
- **Clip…** – Displays the **Clip** dialog box (Figure 2.4) for clipping visualization objects of the scene. The clipping is done when the **Clip** box is checked () and  is pressed. The **Plane Origin** is the origin of the plane used for clipping and the **Plane Normal** is a normal vector on the plane, which determines the direction of clipping. Valid values for **X**, **Y** and **Z** of the **Plane Normal** range between -1 and 1.
- **Scale…** – Displays the **Scale** dialog box (Figure 2.5).
  Visualization objects can be scaled in the positive or negative axes' directions. Valid scaling values range between 0 and 100 for all directions.

Figure 2.3: The **Interact** menu.

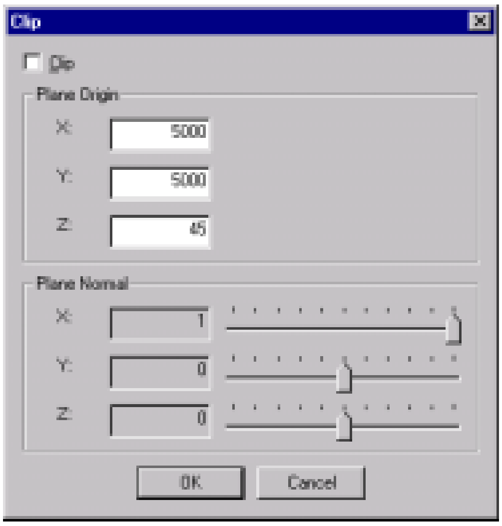


Figure 2.4: The **Clip** dialog box.

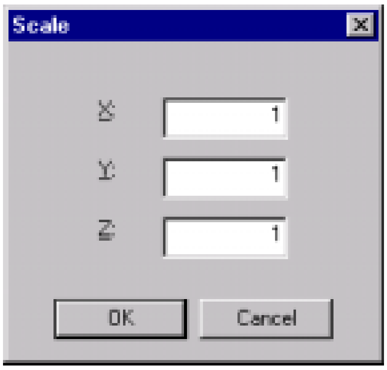


Figure 2.5: The **Scale** dialog box.

2.1.4 Options

The **Options** menu (Figure 2.6) is used to make changes to the axes and visualization objects. The **Options** menu consists of the following items.

- **Axes…** – Displays the **Axes** dialog box (Figure 2.7). The **Axes** dialog box is used to set the properties of the **Label Axes** and **Color Axes**. The **Label Axes** object displays axes with tick marks and corresponding coordinates. The displayed coordinate values are scaled by the factors set in the **Scale** dialog box (Figure 2.5). The **Color Axes** object displays axes, which are fixed at the bottom, left and lower corner of the model. Check (☑) the **Visible** check boxes to make the axes visible and clear (☐) the check boxes to make the axes invisible. In Figure 2.7, the **X** and **Y** axes of the **Label Axes** and the **Color Axes** are checked (☑) as visible and the **Z** axis of the **Label Axes** is invisible.
- **Visualize…** – Displays the **Visualize** dialog box (Figure 2.8), which is used to add or remove visualization objects, or to change properties of existing visualization objects.

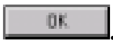
► **To add a visualization object:**

1. Click the  button to display the visualization objects menu (Figure 2.9).
2. Select an appropriate item from the visualization objects menu. A dialog box for the specific visualization object appears.
3. Type in a description and specify other data for the visualization object (refer to Chapter 3 for details), then click .

► **To remove a visualization object:**

1. Select a visualization object from the **Visualize** dialog box.
2. Click the  button.

► **To modify the properties of a visualization object:**

1. Double click a visualization object to open a dialog box containing its available properties.
2. Modify the properties, then click . Refer to Chapter 3 for details.

2.1.5 View

The **View** menu (Figure 2.10) groups items that switch the display of the **Toolbar** and **Status Bar** on or off. The **View** menu consists of the following items:

- **Toolbar** – The toolbar is displayed if the item is checked.
- **Status Bar** – The status bar is displayed if the item is checked.

2.1.6 Info

The **Info** menu contains the following item:

- **About Groundwater Explorer…** – Displays the **About** dialog box showing a short instructions for using GE.

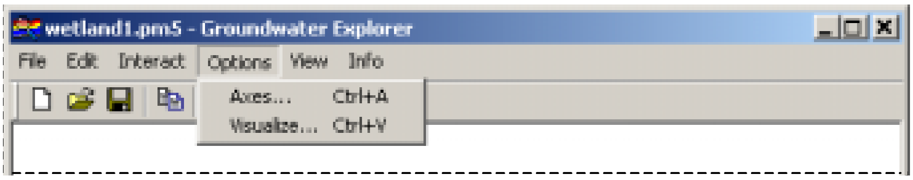


Figure 2.6: The **Options** menu.

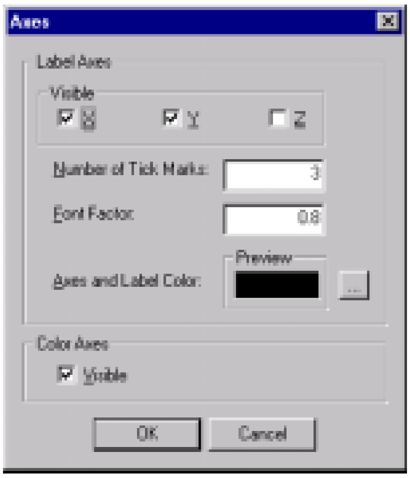


Figure 2.7: The **Axes** dialog box.

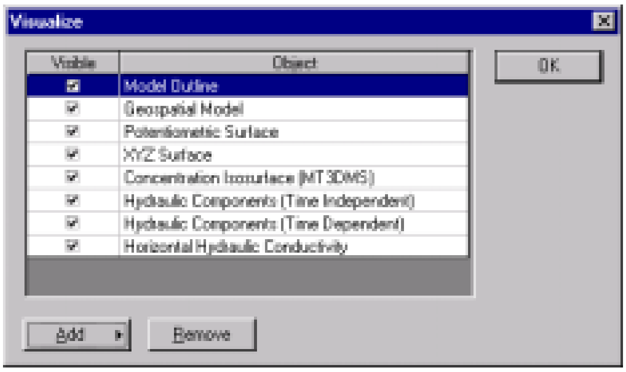


Figure 2.8: The **Visualize** dialog box.

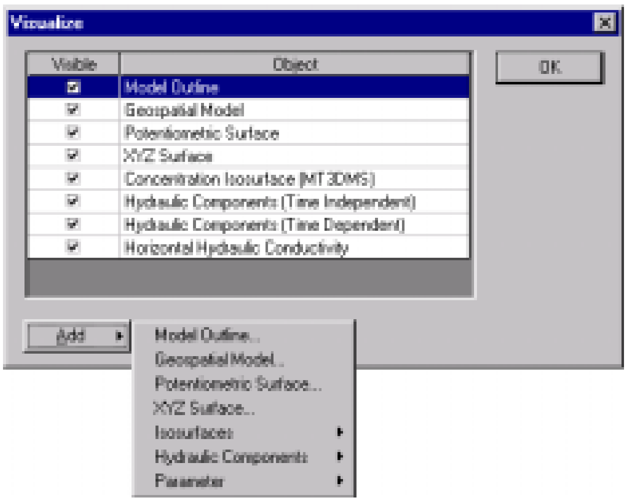


Figure 2.9: The **Add** button items.

Figure 2.10: The **View** menu.

2.2 Tool Bar

The buttons of the tool bar (Figure 1.1, page 2) are shortcuts to menus. The buttons and the corresponding menus are listed in Table 2.1.

Table 2.1: Toolbar buttons.

	**New**
	**Open…**
	**Save**
	**Copy**
	**Print…**
	**Visualize…**
	**About Groundwater Explorer…**

2.3 Status Bar

The status bar (Figure 1.1, page 2) displays processing and help information in the left corner. On the right side of the status bar, indicators show the status of the Caps Lock (**CAP**) and Num Lock (**NUM**) keys.

2.4 Control the Scene with the Mouse and Keyboard

Groundwater Explorer allows you to control the scene by using the mouse or keyboard in the following ways:

- Rotate the scene by holding down the left mouse button and moving the mouse.
- Zoom into the scene by holding down the right mouse button and moving the mouse pointer to the top half of the scene.
- Zoom out of the scene by holding down the right mouse button and moving the mouse pointer to the bottom half of the scene.
- Pressing the **w** key (without ctrl or shift keys) switches the display of all objects to the wireframe mode. The **s** key (without ctrl or shift keys) switches the display of all objects to the surface mode.

Chapter 3

Visualization Objects

3.1 Introduction

The visualization objects that can be added to a scene in GE are: Model Outline, Geospatial Model, Potentiometric Surface, XYZ Surface, Concentration Isosurface, Hydraulic Components and Distribution Parameters.

The following sections describe the groundwater model data used for each visualization object, and show examples of the objects. The components of the dialog boxes are also explained.

3.2 Model Outline

The Model Outline visualization object is a wire frame bounding box of the model domain. The bounding box is defined by the spatial extent of the model domain, including minimum and maximum x, y and z coordinates (see Section 3.2.1 below). Figure 3.1 shows a Model Outline visualization object with its minimum and maximum corners along with a geospatial model object (see section 3.3, page 15) and the (XYZ-) Color Axes, which are fixed at the bottom, left and lower corner of the model.

3.2.1 Bounds

When a model is loaded into GE, the bounds of the spatial extent of the model are determined automatically. You can change the bounds in the **Model Outline** dialog box (Figure 3.2). To open this dialog box, select Visualize from the options menu and double-click on the Model Outline object (Figure 2.8, page 10). The elements of the **Model Outline** dialog box are described below:

- **Description** is a text description by which the Model Outline visualization object is identified.
- **X Minimum**, **Y Minimum**, **Z Minimum** are the x, y and z coordinate values, which defines the *minimum* corner (Figure 3.1) of the Model Outline object.
- **X Maximum**, **Y Maximum** and **Z Maximum** are the x, y and z coordinate values, which defines the *maximum* corner (Figure 3.1) of the Model Outline object.

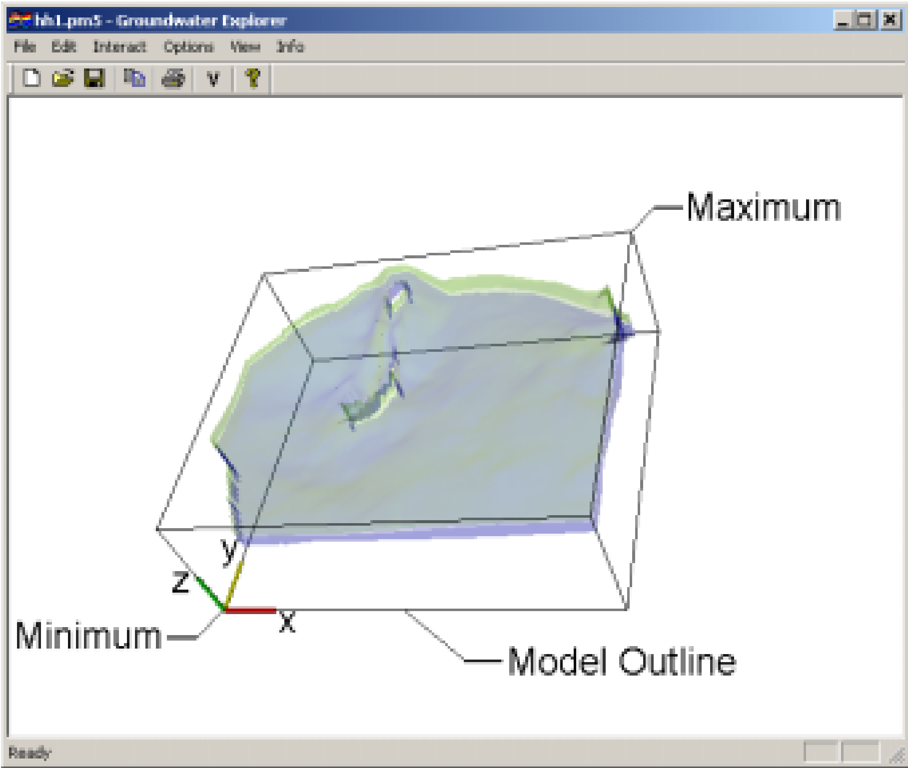


Figure 3.1: Minimum and maximum corners of Model Outline visualization object.

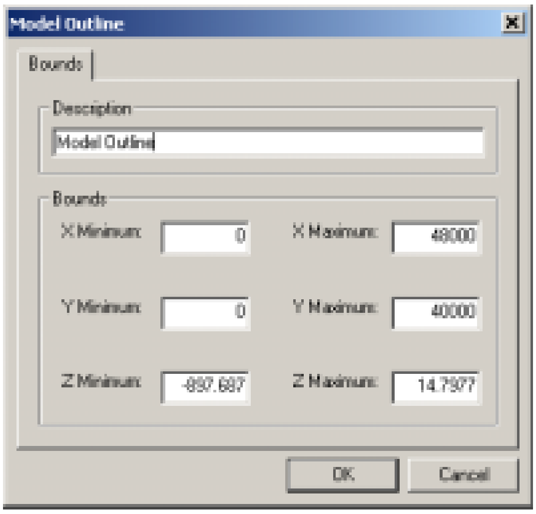


Figure 3.2: The **Model Outline** dialog box.

3.3 Geospatial Model

The Geospatial Model visualization object is a polygonal mesh of the active model cells for each layer, which is defined in the groundwater model. Inactive groundwater model cells are not displayed. Different display properties can be set for each layer. Figure 3.3 shows a Geospatial Model visualization object for a three-layer groundwater model. The polygonal mesh is built from polygonal quadrilaterals that are triangulated for decimation and smoothing. When adding a Geospatial Model object, the Geospatial Model dialog box (Figure 3.4) appears automatically. Once the object has been added, you can subsequently change the display properties in the Geospatial Model dialog box (Figure 3.4) by selecting Visualize from the Options menu and double-clicking on the Geospatial Model object (Figure 2.8, page 10).

3.3.1 General

The options of the General page (Figure 3.4) are described below:

- **Description** is a text description by which the Geospatial Model visualization object is identified.
- **Active:** Check (☑) the **Active** check box to make a layer visible, or clear (☐) it to make a layer invisible. For example in Figure 3.4 all layers are checked to be visible.
- **Color:** Double-click a cell of the **Color…** field to change the color of an individual layer or click the **Color...** header button to display the **Color Spectrum** dialog box (Figure 3.5). On the Color Spectrum dialog box, **Minimum Color** is the color used for the first layer, and **Maximum Color** is used for the last layer. Layers between the first and last layer are assigned color values using a linear scale. Click the ... button of the **Minimum Color** or **Maximum Color** to change their respective colors. The new color for **Minimum Color** or **Maximum Color** is selected from the **Color** dialog box (Figure 3.6). A color is selected by clicking on a color from the **Basic colors**, or a color is custom defined by clicking on the **Define Custom Colors**.
- **Opacity**: Opacity values range between 0 and 1. A value of 0 sets a visualization object translucent and a value of 1 opaque. A value between 0 and 1 sets a visualization object semi-tranparant. Setting an opacity value smaller than 1 makes it possible to see objects inside the Geospatial Model. For example, Figure 3.7 shows a Geospatial Model

visualization object where the opacity values for all layers were set to 0.2 and the potentiometric surface inside the Geospatial Model is visible.

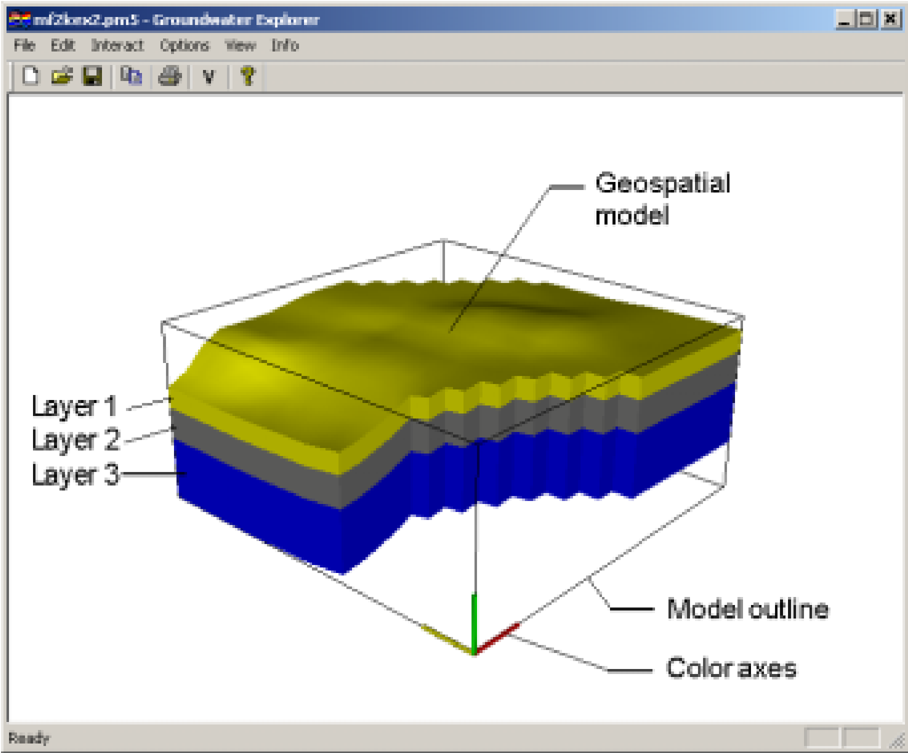


Figure 3.3: The Geospatial Model visualization object.

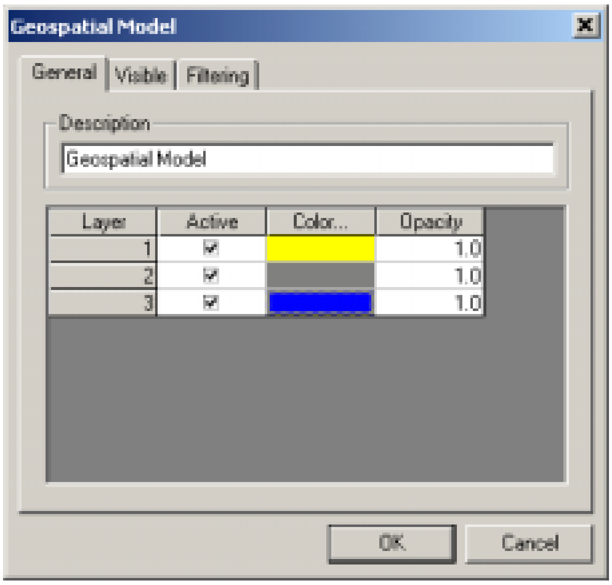


Figure 3.4: The **General** tab of the **Geospatial Model** dialog box.

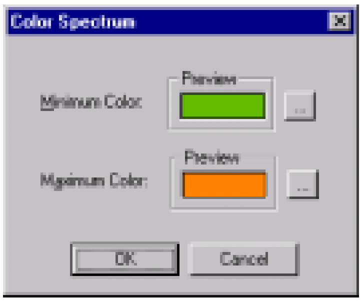


Figure 3.5: The **Color Spectrum** dialog box.

Figure 3.6: The **Color** dialog box.

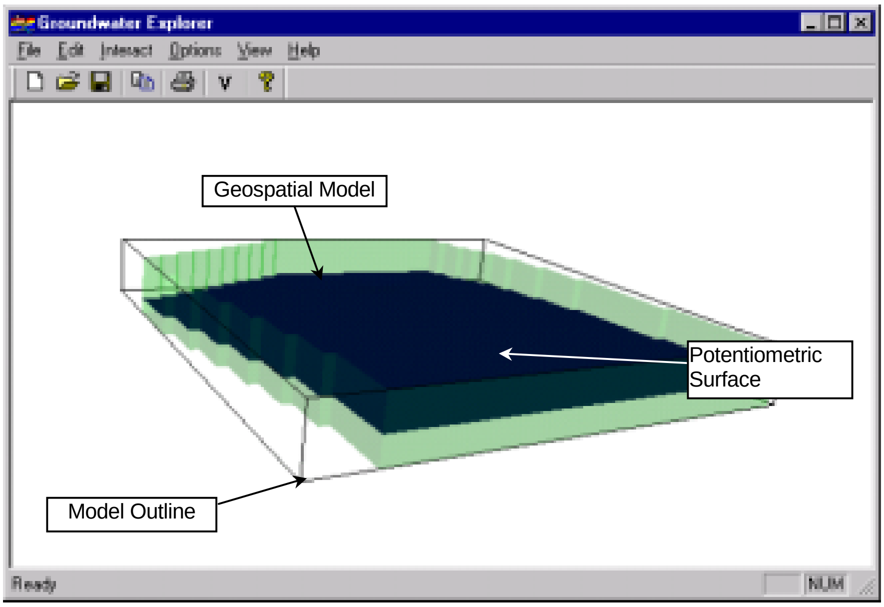


Figure 3.7: Potentiometric Surface visualization object can be seen inside the Geospatial Model visualization object of which the layer opacity values were set to 0.2.

3.3.2 Visible

Figure 3.8 shows the Visible page of the **Geospatial Model** dialog box. Check (☑) or clear (☐) the check boxes in the **Top**, **Sides** and **Bottom** fields, to set the visibility for the top, sides and bottom of a Geospatial Model, for a specific layer. The **Top**, **Sides** and **Bottom** of a Geospatial Model for a layer are shown in Figure 3.9.

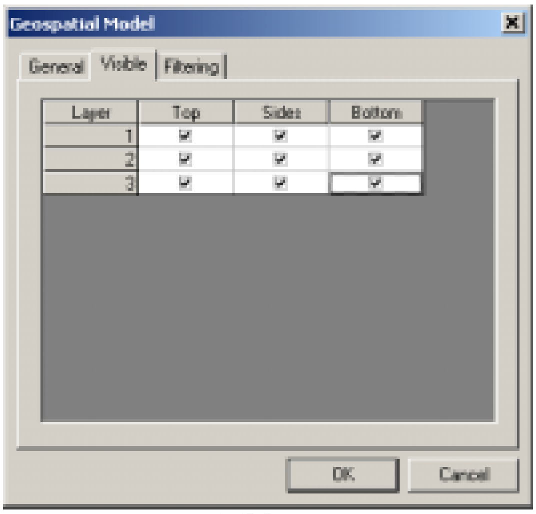
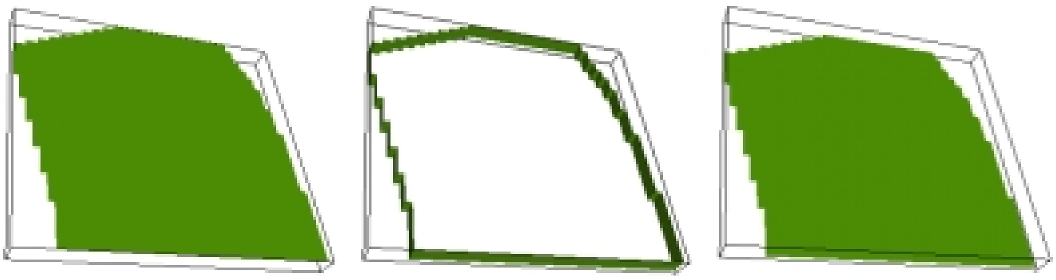


Figure 3.8: The **Visible** page of the **Geospatial Model** dialog box.



(a) (b) (c)

Figure 3.9: (a) **Top**, (b) **Sides** and (c) **Bottom** of a Geospatial Model visualization object for a layer.

3.3.3 Filtering

The options of the **Filtering** page of the **Geospatial Model** dialog box (Figure 3.10) are described below:

- **Average**: Check (☑) or clear (☐) the check boxes in the **Average** field to switch averaging on or off for a layer. Averaging applies an arithmetical smooth of the layer elevations (Figure 3.11).
- **Decimate**: Check (☑) the **Decimate** check box for decimation to be done. Decimation results in fewer triangles used to create the polygonal mesh of the Geospatial Model, and has the advantage of faster rendering and interaction times.

- **Reduction:** Reduction values range between 0 and 1. This value is only used when **Decimate** is checked. **Reduction** is the desired reduction rate in the total number of triangles that make up the Geospatial Model. Due to mathematical constraints and depend on the geometry of the model, the level of the desired reduction rate may not always be realized. Figure 3.12 shows the decimation results on the top part of a Geospatial Model visualization object. The model used to create the Geospatial Model is a single layer model with two columns and two rows. For (a) no decimation was done. For (b), (c) and (d) the target reduction rate was set to 0.3, 0.5 and 0.7, respectively.
- **Smooth, Iterations and Relaxation:** Check (☑) the **Smooth** check box to use Laplacian smoothing. Smoothing is used with **Average** and optionally with **Decimate**. The fields associated with **Smooth**, are the number of **Iterations** and the **Relaxation** factor to be specified to the right of the table shown in Figure 3.10. **Iterations** sets the number of iterations for Laplacian smoothing, and **Relaxation** specifies the relaxation factor for Laplacian smoothing. As with all iterative methods, the stability of the process is sensitive to the **Relaxation** field. In general, small relaxation factors and large numbers of iterations are more stable than larger relaxation factors and a smaller number of iterations. Figure 3.13 (a) shows the **Top** part of a Geospatial Model visualization object without smoothing. Figure 3.13 (b) shows the result of smoothing where **Iterations** was set to 10 and **Relaxation** to 0.2.

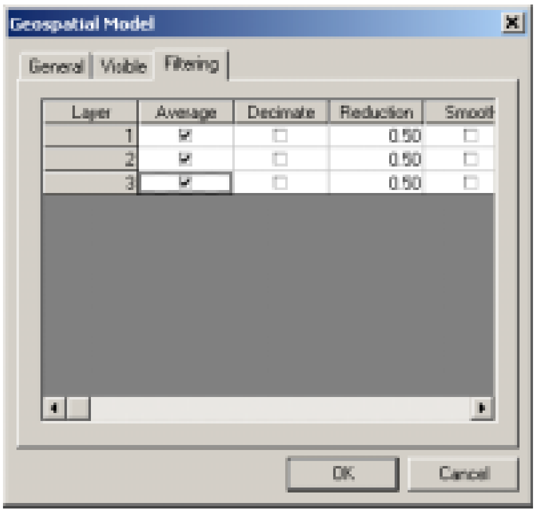
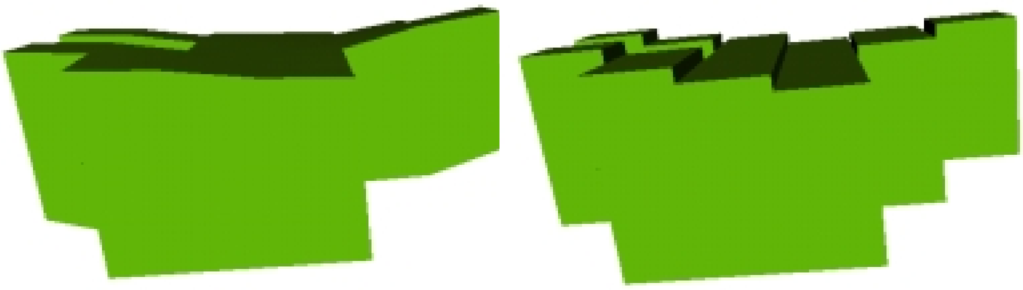
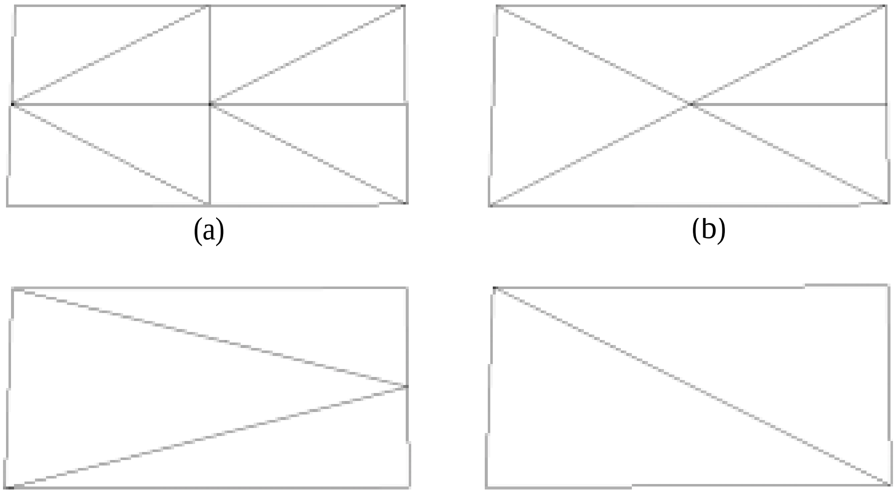


Figure 3.10: The **Average** page of the **Geospatial Model** dialog box.



(a) (b)

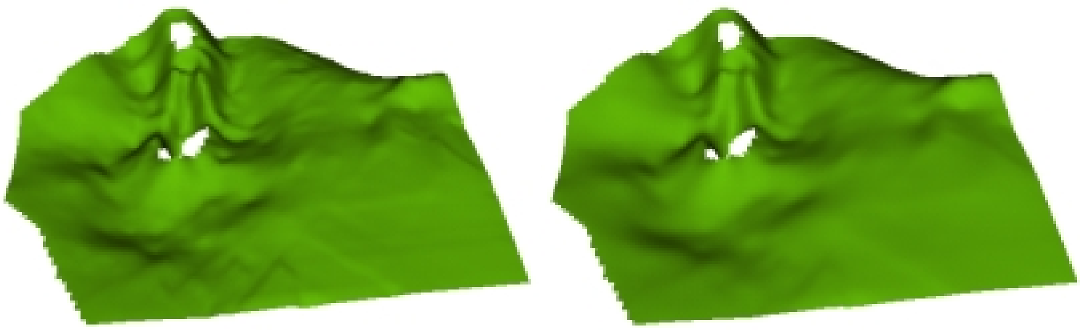
Figure 3.11: (a) Layer 1 of Geospatial Model visualization object with Average checked. (b) Layer 1 of Geospatial Model visualization object with Average not checked.



(a) (b)

(c) (d)

Figure 3.12: (a) No decimation. (b) **Reduction** = 0.3. (c) **Reduction** = 0.5. (d) **Reduction** = 0.7.



(a) (b)

Figure 3.13: (a) No smoothing. (b) Smoothing with **Iterations** = 10 and **Relaxation** = 0.2.

3.4 Potentiometric Surface

The Potentiometric Surface is a visualization object (Figure 3.14), which represents the calculated hydraulic heads in the highest active cells for all layers, or the calculated hydraulic heads in active cells for a specified layer. When adding a Potentiometric Surface object, the **Potentiometric Surface** dialog box (Figure 3.15) appears automatically. Once the object has been added, you can subsequently change the display properties in the Potentiometric Surface dialog box by selecting Visualize from the options menu and double-clicking on the Potentiometric Surface object (Figure 2.8, page 10).

3.4.1 General

The options of the **General** page of the Potentiometric Surface dialog box (Figure 3.15) are described below:

- **Description** is a text description by which the Potentiometric Surface visualization object is identified.
- **Stress Period** and **Time Step:** In PMWIN and MODFLOW, simulation time is divided into stress periods, which are in turn divided into time steps. During a groundwater flow simulation, the hydraulic heads are calculated for each stress period and time step. When adding a Potentiometric Surface visualization object, you can select a pre-defined time step and stress period, from which the calculated hydraulic heads are read and used for the visualization object.
- **Use heads in highest active cells**: Check ( ) this box, if the groundwater heads in the highest active cells are to be visualized. If the heads for a specific layer are to be visualized, clear ( ) this box and select the required layer from the **Layer** list.

3.4.2 Filtering

Figure 3.16 shows the **Filtering** page of the **Potentiometric Surface** dialog box. Refer to Section 3.3.3 (page 18) for detailed explanations of the options.

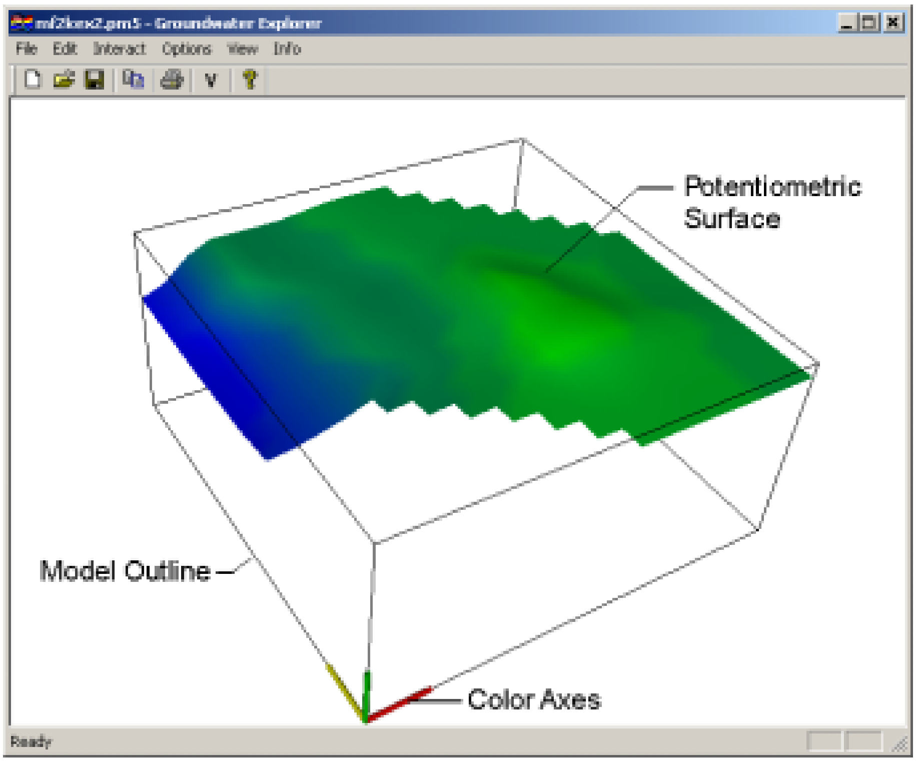


Figure 3.14: The Potentiometric Surface visualization object.

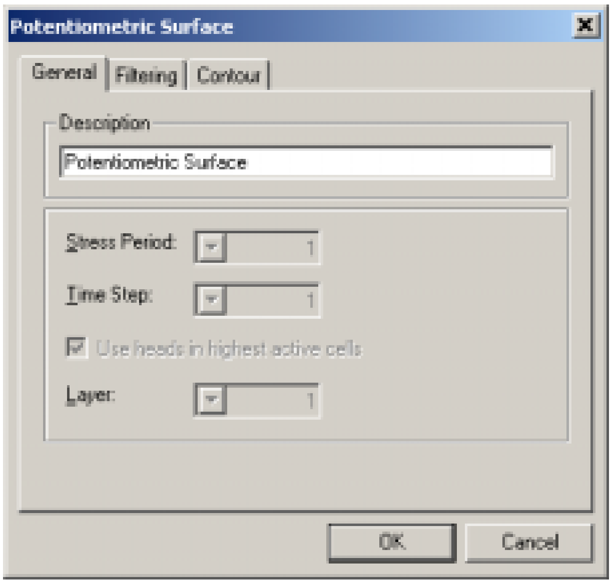


Figure 3.15: The **General** page of the **Potentiometric Surface** dialog box.

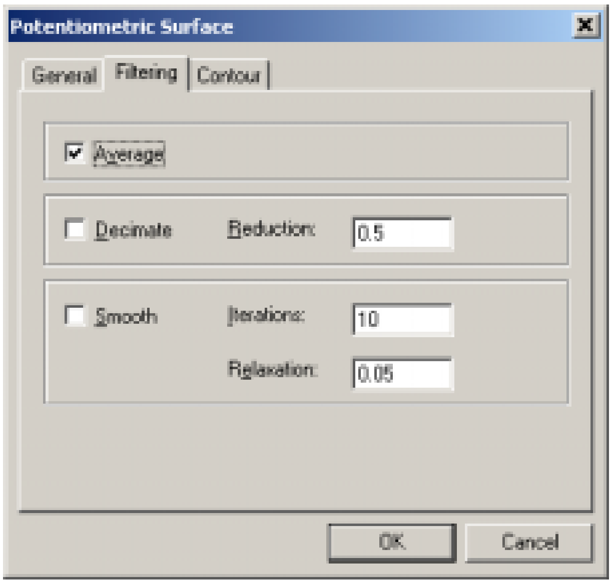


Figure 3.16: The **Filtering** page of the **Potentiometric Surface** dialog box.

3.4.3 Contour

Figure 3.17 shows the **Contour** page of the Potentiometric Surface dialog box. The components of this dialog box are described below:

- **Opacity**: The **Opacity** value for the Potentiometric Surface visualization object is set to 1. Opacity values range between from 0 and 1. A value of 0 sets a visualization object translucent and a value of 1 sets a visualization object opaque. A value between 0 and 1 sets a visualization object semi-transparent.
- **Minimum…** and **Maximum…:** Click one of these header buttons to display the **Contour Level** dialog box (Figure 3.18), which allows you to specify the contour interval and contour range using Minimum and Maximum values. In Figure 3.18 the **Minimum** head value that is to be displayed is 0 and the **Maximum** head value is 1500, with the interval specified as 200. Specific colors (defined by clicking **Color…** below) will be assigned to each interval of 200 from 0 to 1500. The result of using these values is shown in Figure 3.14.
- **Color…:** The colors are changed in the same way as the colors for the Geospatial Model object. Refer to section 3.3.1 (page 15) for details.

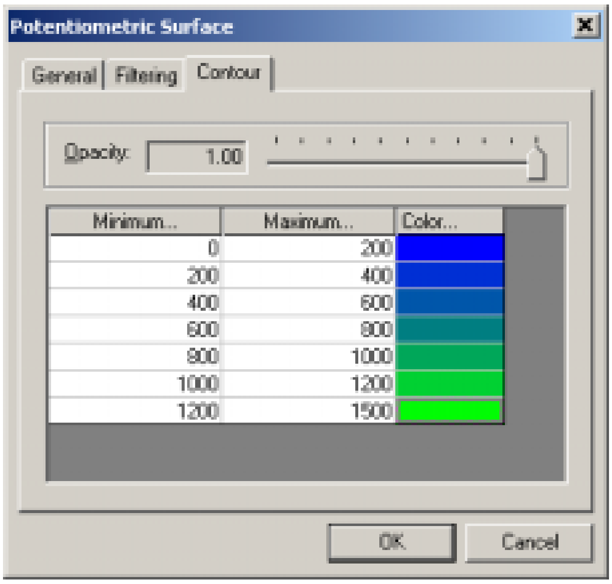


Figure 3.17: The **Color** page of the **Potentiometric Surface** dialog box.

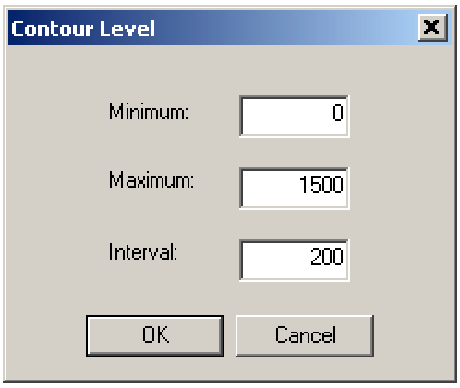


Figure 3.18: The **Contour Level** dialog box.

3.5 XYZ Surface

The XYZ Surface visualization object (Figure 3.19) is created from a file containing x, y and z real-world coordinates. When adding an XYZ Surface object, the XYZ Surface dialog box (Figure 3.20) appears automatically. Once the object has been added, you can subsequently change the display properties in the XYZ Surface dialog box by selecting Visualize from the Options menu and double-clicking on the XYZ Surface object (Figure 2.8, page 10).

3.5.1 File

The options of the **File** page of the **XYZ Surface** dialog box (Figure 3.20) are described below:

- **Description** is a text description by which the XYZ Surface visualization object is identified.
- **XYZ File** is the folder and name of the file containing x, y and z coordinates, from which a polygonal mesh representing the XYZ surface is to be generated. Type the folder and filename in the **XYZ File** box or click on the  button to select an XYZ-file from an **Open** dialog box. An XYZ-file must be saved in ASCII. Figure 3.21 shows the format of the XYZ file. The first column contains the x coordinates, the second the y coordinates and the third the z coordinates. All values are separated by "," or at least a blank. Note that you can use the Result Extractor of PMWIN (Chiang and Kinzelbach, 2001) to generate XYZ files from the simulation results.

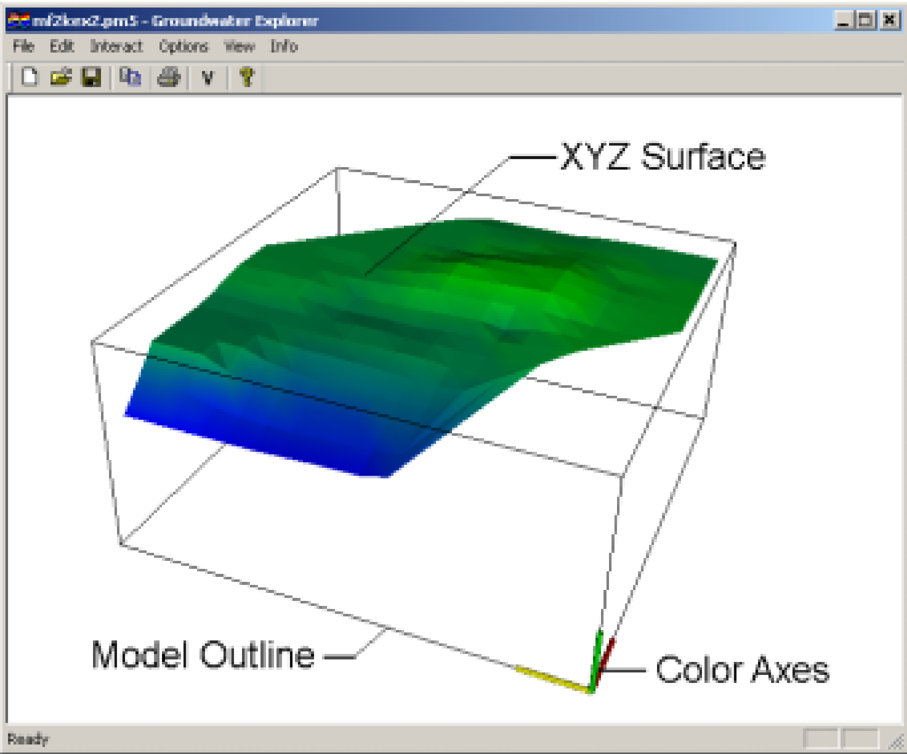


Figure 3.19: The XYZ Surface visualization object.

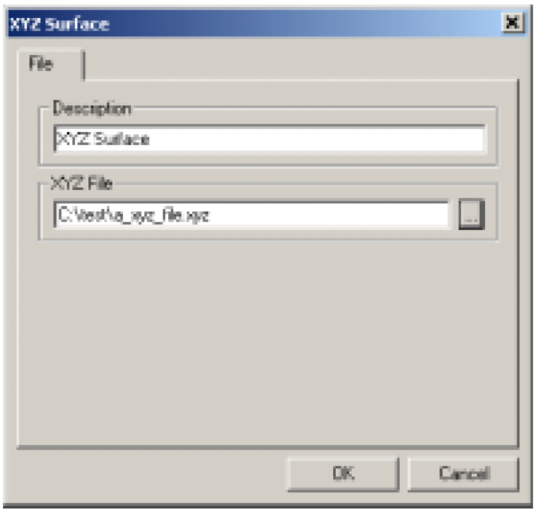


Figure 3.20: The **File** page of the **XYZ Surface** dialog box.

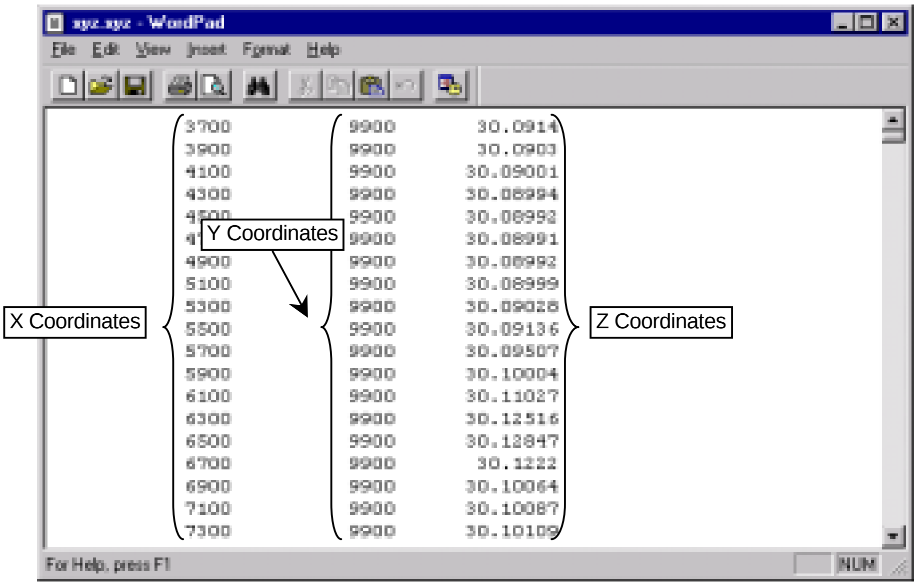


Figure 3.21: An example of an XYZ file edited in WordPad.

3.5.2 Filtering

Figure 3.22 shows the Filtering page of the **XYZ Surface** dialog box. Refer to Section 3.3.3 (page 18) for detailed explanations of the options.

3.5.3 Contour

Figure 3.23 shows the **Contour** page of the **XYZ Surface** dialog box.. Refer to Section 3.4.3 (page 23) for detailed explanations of the options.

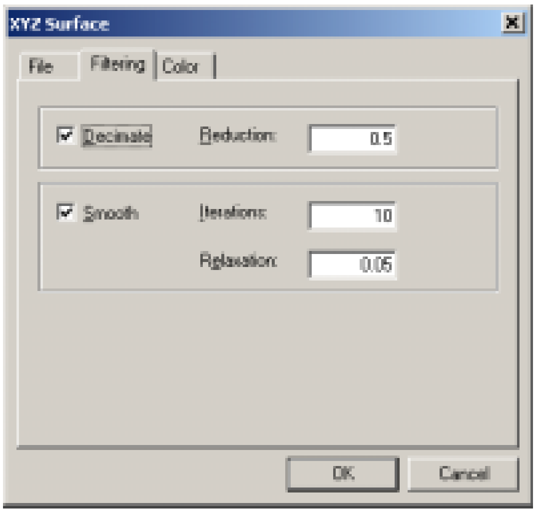


Figure 3.22: The **Filtering** page of the **XYZ Surface** dialog box.

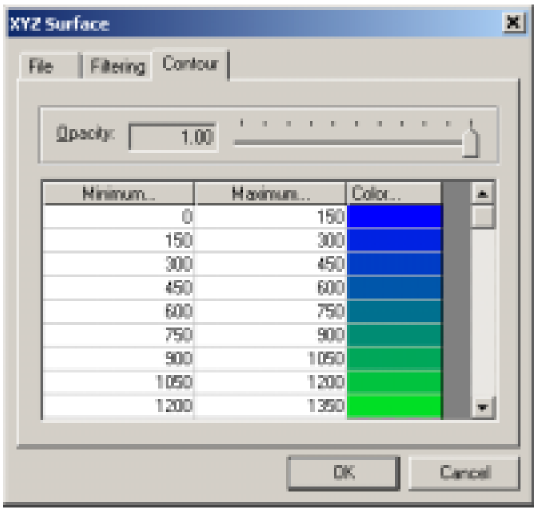


Figure 3.23: The **Contour** page of the **XYZ Surface** dialog box.

3.6 Concentration Isosurface

Concentration Isosurfaces (Figure 3.24) are created from concentration values calculated by MT3DMS, PHT3D or RT3D. In the following sections, MT3DMS will be used. The dialog components for PHT3D and RT3D are exactly the same as for MT3DMS. When adding a Concentration Isosurface object, the Concentration Isosurface (MT3DMS) dialog box (Figure 3.25) appears automatically. Once the object has been added, you can subsequently change the display properties in the Concentration Isosurface dialog box by selecting Visualize from the Options menu and double-clicking on the Concentration Isosurface (MT3DMS) object (Figure 2.8, page 10).

3.6.1 General

The options of the **General** page of the **Concentration Isosurface (MT3DMS)** dialog box (Figure 3.25) are described below.

- **Description** is a text description by which the Concentration Isosurface (MT3DMS) visualization object is identified.
- **Species Number:** In MT3DMS (as well as PHT3D and RT3D), one can define and calculate the concentration values of more than one species. When adding a Concentration Isosurface object, you can select a species from **Species Number.** The calculated concentration values of the selected species are read and used for visualization.
- **Total Elapsed Time:** MT3DMS (as well as PHT3D and RT3D) saves the calculated concentration values at times as specified by the user in PMWIN (Chiang and Kinzelbach, 2001). The elapsed time is measured from the beginning of the simulation. When adding a Concentration Isosurface object, you can select a time from **Total Elapsed Time.** The calculated concentration values of the selected time are read and used for visualization.

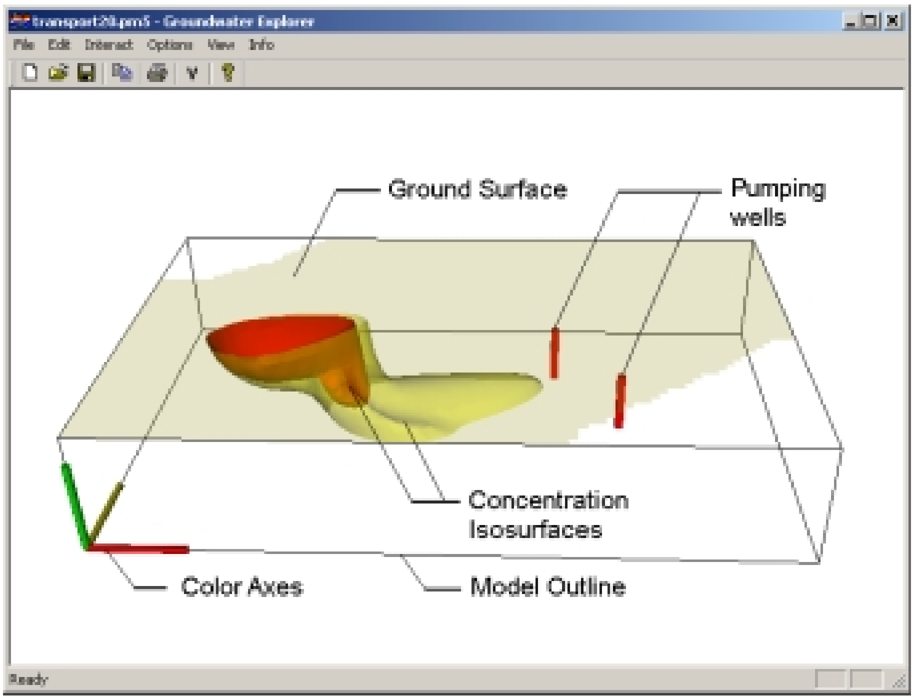


Figure 3.24: The Concentration Isosurfaces colored in red and yellow showing the development of a concentration plume at different times.

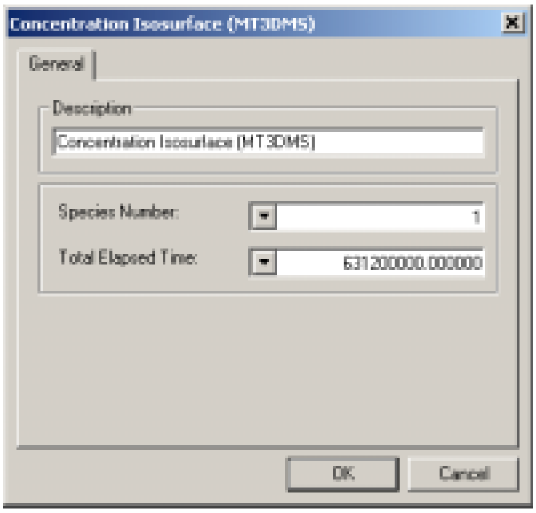


Figure 3.25: The **General** page of the **Concentration Isosurface (MT3DMS)** dialog box.

3.6.2 Contour

The components of the **Contour** page of the **Concentration Isosurface (MT3DMS)** dialog box (Figure 3.26) are described below.

- **Contour Level…:** Isosurfaces are created for the specified contour values in the **Contour Level…** field. Clicking on the **Contour Level…** header button displays the **Contour Level** dialog box (Figure 3.27) which can be used to change the contour values. The **Contour Level** dialog box is used to set the **Minimum** and **Maximum** contour values, and the interval for contour spacing.
- **Color…**: Isosurfaces are colored with the specified colors in the **Color…** field. The colors are changed in the same way as the colors for the Geospatial Model object. Refer to section 3.3.1 (page 15) for details.
- **Remove:** The row mark with  is removed when clicking on the  button.
- **Add:** A row is added by clicking on the  button.

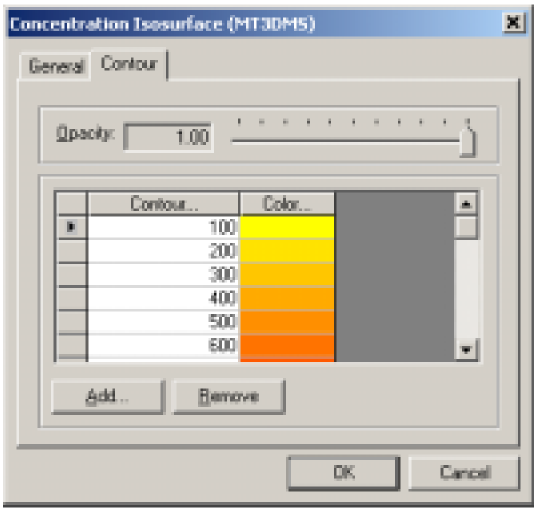


Figure 3.26: The **Contour** page of the **Concentration Isosurface (MT3DMS)** dialog box.

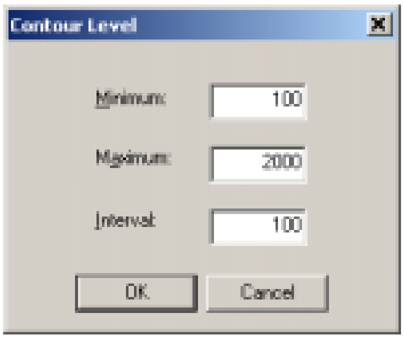


Figure 3.27: The **Contour Level** dialog box.

3.7 Hydraulic Components

The Hydraulic Components visualization object displays groundwater model cells which contain one of the hydraulic components listed below.

Time-independent hydraulic components that can be visualized are:

- Fixed Head
- Fixed Concentration
- Horizontal Flow Barriers
- Reservoir

Time-dependent hydraulic components that can be visualized are:

- Discharge well
- Drain
- General Head Boundary
- Recharge well
- River
- Time Variant Specified Concentration
- Time Variant Specified Head

When adding a Hydraulic Components object, the **Hydraulic Components (Time Independent)** dialog box (Figure 3.28) or the **Hydraulic Components (Time Dependent)** dialog box (Figure 3.29) appears automatically. Once the object has been added, you can subsequently change the display properties in the dialog box by selecting Visualize from the Options menu and double-clicking on the Hydraulic Components object (Figure 2.8, page 10).

3.7.1 General

The components of the General pages shown in Figure 3.28 and Figure 3.29 are described below:

- **Description** is a text description by which the Hydraulic Components visualization object is identified.
- **Visible:** Check (☑) a **Visible** check box to make a hydraulic component visible, or clear (☐) the box to make a hydraulic component invisible.
- **Color:** The **Color** field is used for specifying the colors for different hydraulic components. To change the color of a hydraulic component, double click the **Color** cell of the component and select a new color from a **Color** dialog box. Section 3.3.1 (page 15) explains how to use the **Color** dialog box.
- **Stress Period:** This appears only in the **Hydraulic Components (Time Dependent)** dialog box. In PMWIN and MODFLOW, external excitations or stresses (for example, pumping wells) can vary from stress period to stress period. When adding a Hydraulic Components visualization object, you can select a pre-defined stress period, from which the hydraulic components are read and used for visualization.

3.7.2 Filtering

The only available filtering method for hydraulic components is **Average.** Refer to section 3.3.3 (page 18) for details.

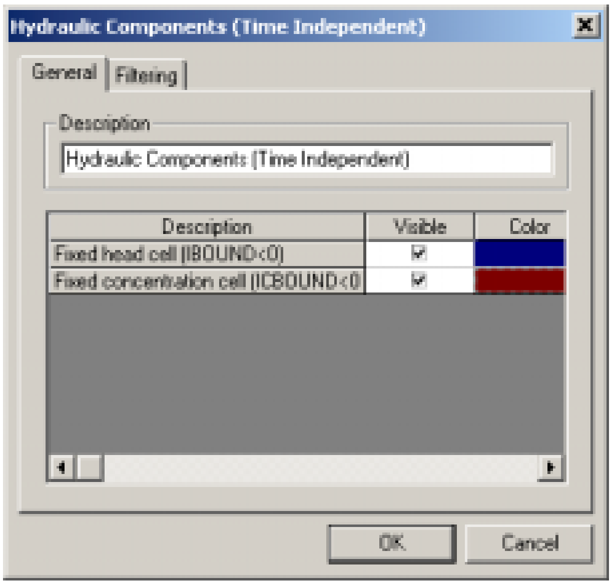


Figure 3.28: The **General** page of the **Hydraulic Components (Time Independent)** dialog box.

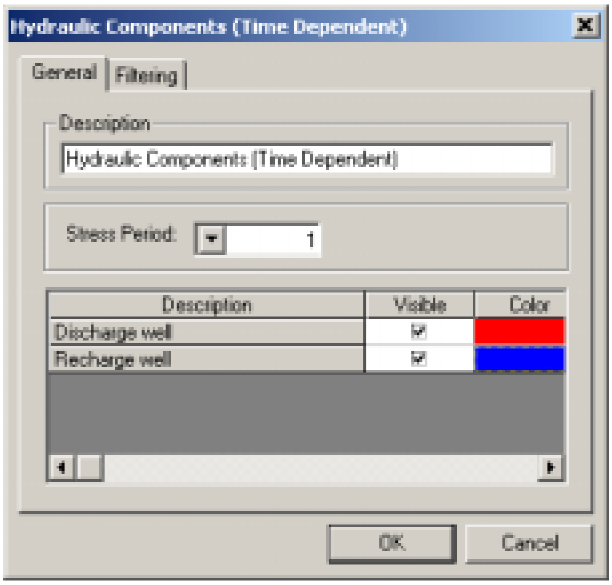


Figure 3.29: The **General** page of the **Hydraulic Components (Time Dependent)** dialog box.

3.8 Parameter

The Parameter visualization object colors groundwater model cells based on the values of the following parameters:

- Horizontal Hydraulic Conductivity
- Vertical Hydraulic Conductivity
- Specific Storage
- Transmissivity
- Vertical Leakance
- Storage Coefficient
- Effective Porosity
- Specific Yield

3.8.1 General

The options of the **General** page of the **Parameter** dialog box (Figure 3.30) are described below:

- **Description** is a text description by which the Parameter visualization object is identified.
- **Visible**: Check (☑) the check box to make a parameter visible for a layer, or clear (☐) them to make the specific layer invisible.

3.8.2 Filtering

The only available filtering method for Parameter visualization objects is **Average.** Refer to section 3.3.3 (page 18) for details.

3.8.3 Contour

Figure 3.31 shows the **Contour** page of the **Parameter** dialog box.. Refer to Section 3.4.3 (page 23) for detailed explanations of the options.

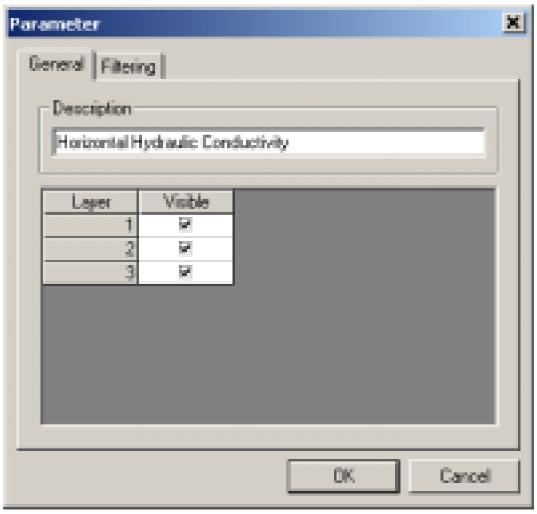


Figure 3.30: The **General** page of the **Parameter** dialog box.

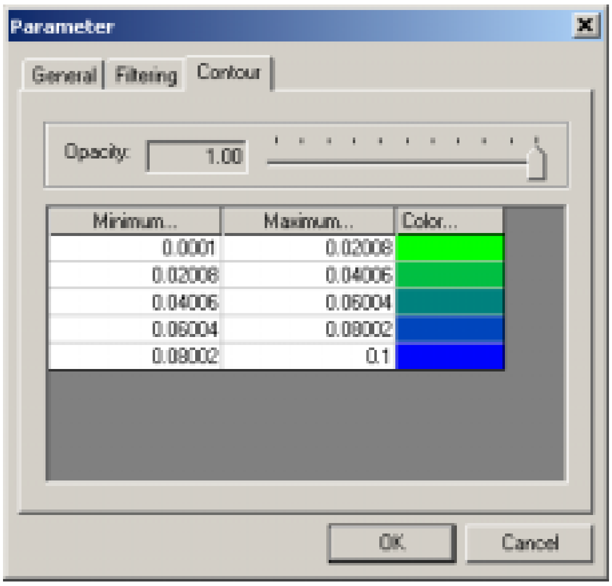


Figure 3.31: The **Contour** page of the **Parameter** dialog box.

Chapter 4

Tutorial

4.1 Introduction

This chapter gives step-by-step instructions on how to visualize existing PMWIN models. The PMWIN models used for this visualization tutorial were course material for a short course titled "Applied Groundwater Modeling using PMWIN", presented by the first author of the present text at the University of the Free State, South Africa from the 27th to the 30th of August 1999.

The synthetic models has two model layers. A well field is proposed to be built near a wetland/lake. The well field will supply drinking water and water for industrial use to a nearby community. The proposed pumping rate of the well field is 3600 m^3/d, and is evenly distributed over six model cells in the second model layer (Figure 4.1).

4.2 Geospatial Model and Potentiometric Surface

The first course task was to construct a flow model for calculating the hydraulic heads under the natural aquifer condition. The Wetland1 model resulted from this task. The following sections are instructions on how to use GE to visualize the natural condition of the aquifer, including a geospatial model and the calculated steady-state potentiometric surface.

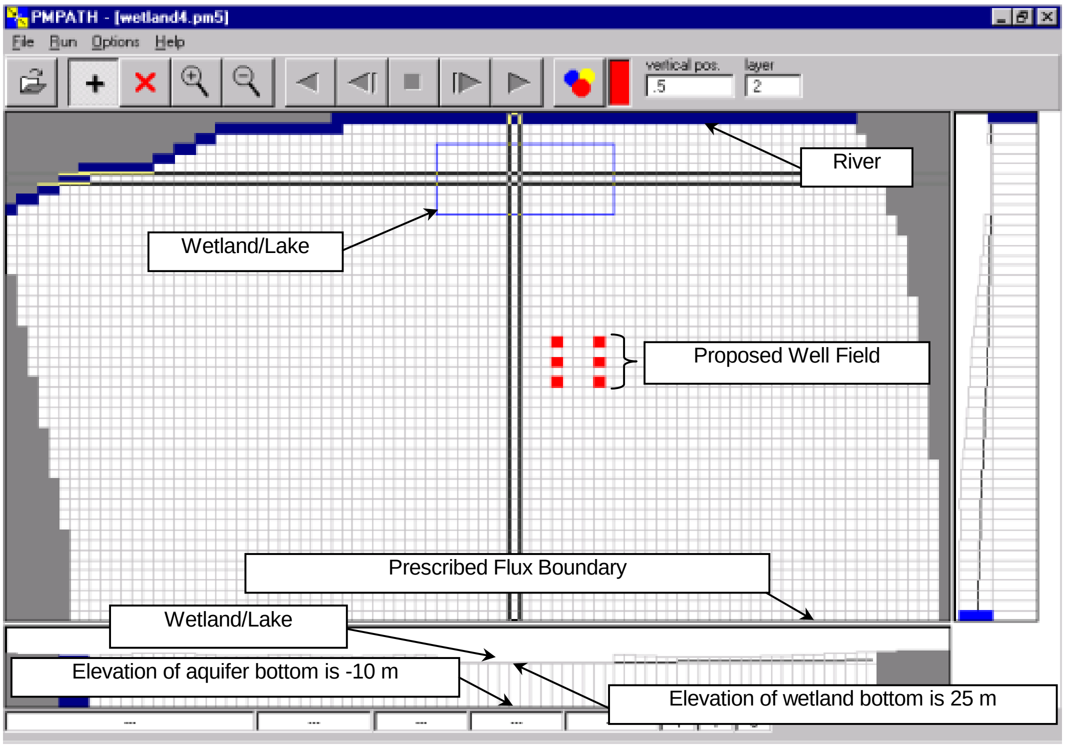


Figure 4.1: Configuration of the Wetland model

4.2.1 Step 1: Start GE

Select **Groundwater Explorer** from the **Programs** menu list.

4.2.2 Step 2: Open the Wetland Model

To start visualizing the PMWIN model, Wetland1 must be opened. From the **File** menu, click the **Open…** item (Figure 4.2) and select *wetland1.pm5* from the ...\ *Models* \ *Wetland1* \ file folder on the accompanying CD (Figure 4.3). Click the Open button to open the Wetland1 model for visualization. After opening the model, the Color Axes and Model Outline visualization objects are displayed (Figure 4.4).

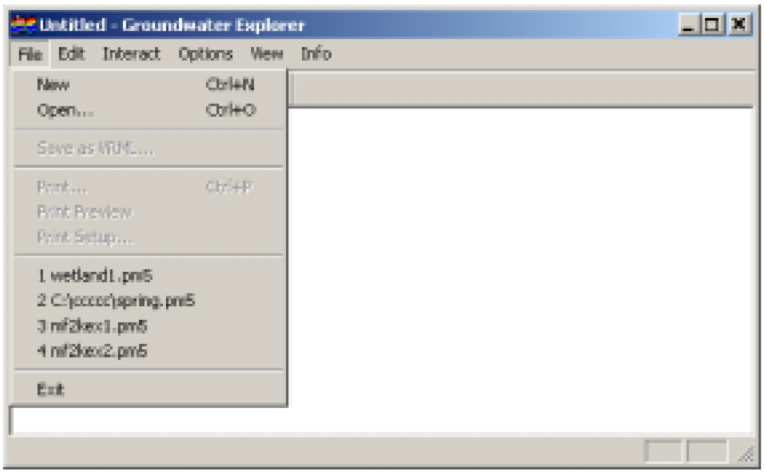


Figure 4.2: The **Open…** item for opening an existing model.

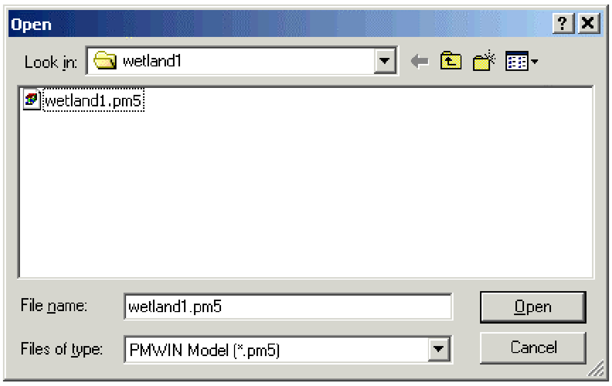


Figure 4.3: The **Open** dialog box.

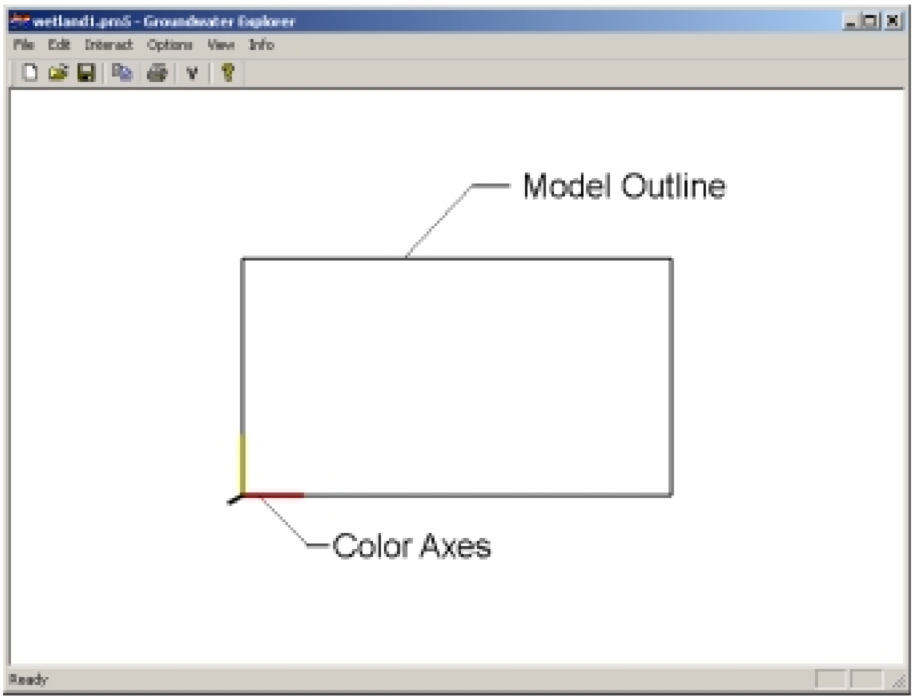


Figure 4.4: The Color Axes and Model Outline visualization objects.

4.2.3 Step 3: Add Geospatial Model

Follow the steps below to add a geospatial model:

1. Click **Visualize…** from the **Options** menu to display the **Visualize** dialog box (Figure 4.5) from which the Geospatial Model visualization object can be added. Note that GE has automatically added the Model Outline visualization object.
2. Click the button to display the **Add** button items (Figure 4.6). A new Geospatial Model visualization object is added by clicking on **Geospatial Model…**, which displays the **Geospatial Model** dialog box (Figure 4.7).
3. Keep the default values and click the button. The Geospatial Model is added to the list of visualization objects in the **Visualize** dialog box (Figure 4.8).
4. In the **Visualize** dialog box, click the button to see the result (Figure 4.9).

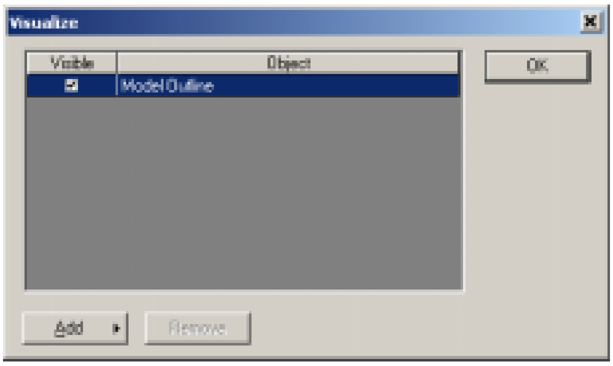


Figure 4.5: The **Visualize** dialog box.

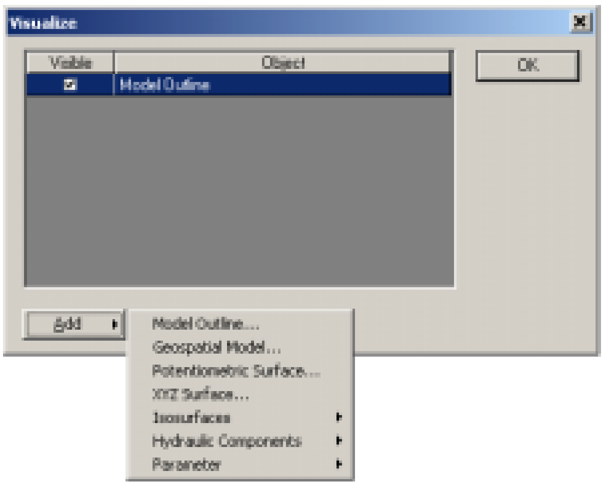


Figure 4.6: The **Add** button items.

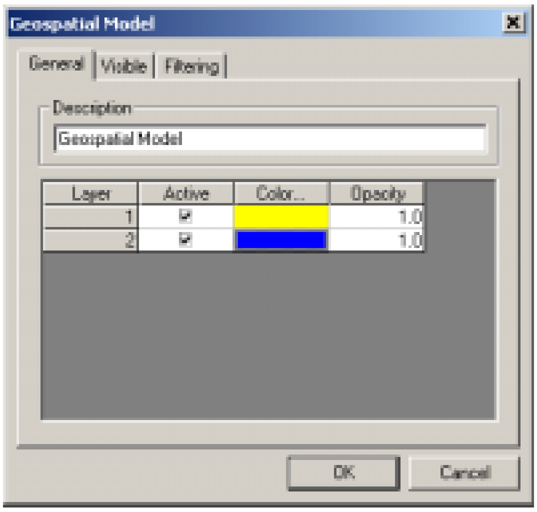


Figure 4.7: The **General** page of the **Geospatial Model** dialog box.

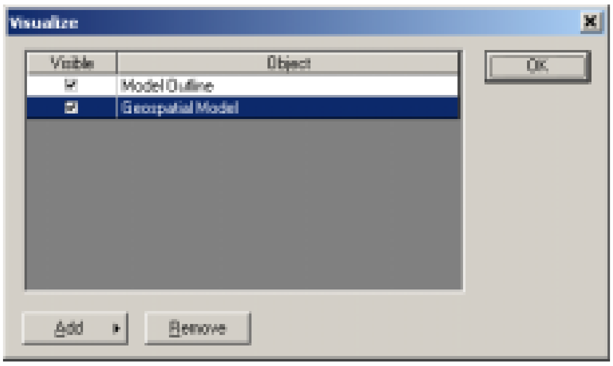


Figure 4.8: The **Visualize** dialog box with Geospatial Model added.

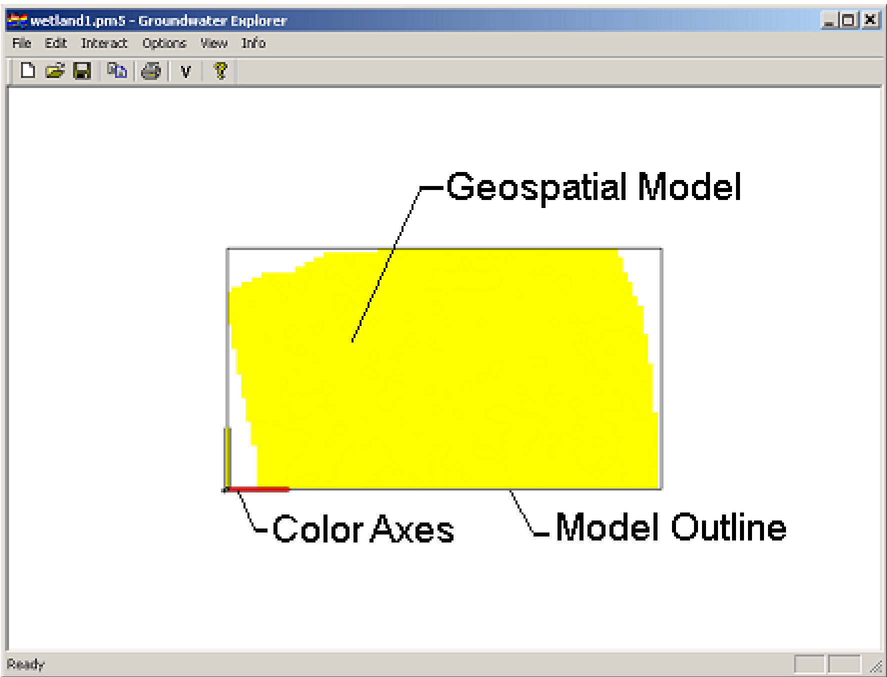
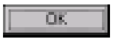


Figure 4.9: The Geospatial Model, Color Axes and Model Outline and visualization objects.

4.2.4 Step 4: Scale

The visualization objects of a scene can be rotated and zoomed using the mouse. See section 2.4 “Control the Scene with the Mouse and Keyboard” (page 11), for an explanation on how to use the mouse to rotate and zoom.

The Geospatial Model is flat in the z coordinate direction, in comparison with the x and y coordinate directions. A good idea is to scale the z coordinate direction to make changes in elevation more obvious.

To scale all the objects in a visualization scene, select **Scale…** from the **Interact** menu to display the **Scale** dialog box (Figure 4.10). Change the value in the **Z** box to 20 and click the  button. Figure 4.11 shows the scaled, rotated and zoomed Geospatial Model. As a result of scaling, the model and two model layers can be seen more clearly.

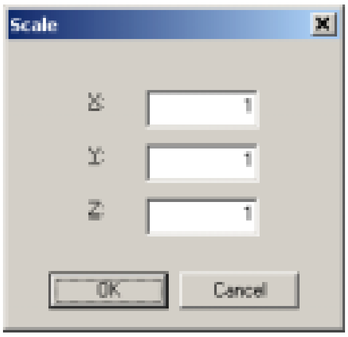


Figure 4.10: The **Scale** dialog box

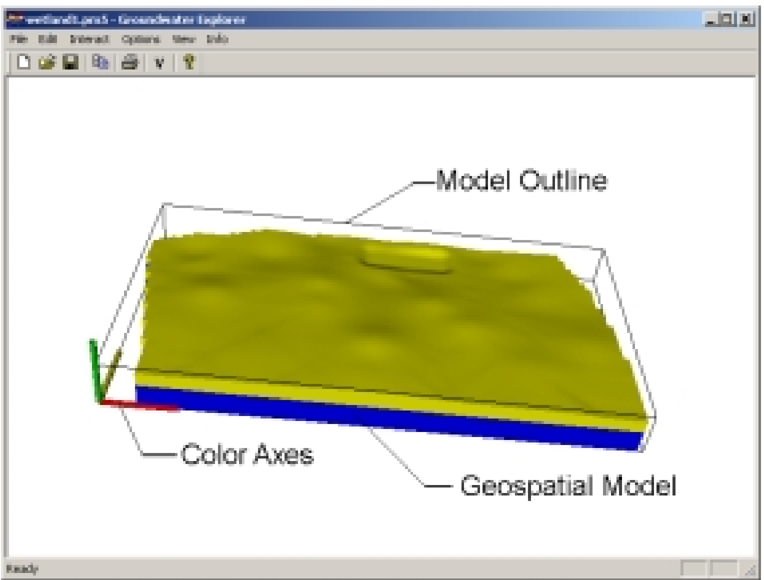


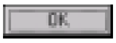
Figure 4.11: Scaled Geospatial Model visualization object.

4.2.5 Step 5: Add Potentiometric Surface

GE can be used to visualize the calculated hydraulic heads of a flow model. The way to visualize the hydraulic heads is described below.

1. Select **Visualize…** from the **Options** menu to display the **Visualize** dialog box (Figure 4.12) to which the Potentiometric Surface visualization object can be added. Note that the list of visualization objects already contains Model Outline and Geospatial Model.
2. Click the 

button to display the **Add** button items (Figure 4.13).
3. Click on **Potentiometric Surface…** to add a Potentiometric Surface visualization object. The **Potentiometric Surface** dialog box (Figure 4.14) is displayed. Keep the defaults for all boxes and click the 

button. The Potentiometric Surface is added to the list of visualization objects in the **Visualize** dialog box (Figure 4.15).
4. In the **Visualize** dialog box, click the 

button to see the result (Figure 4.16). It can be seen that the wetland is filled with water (compare to Figure 4.11, where no Potentiometric Surface was added).

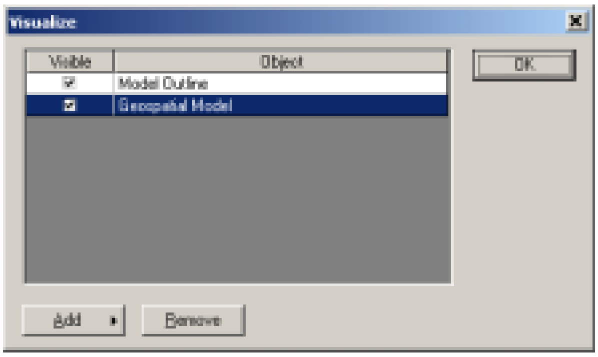


Figure 4.12: The **Visualize** dialog box.

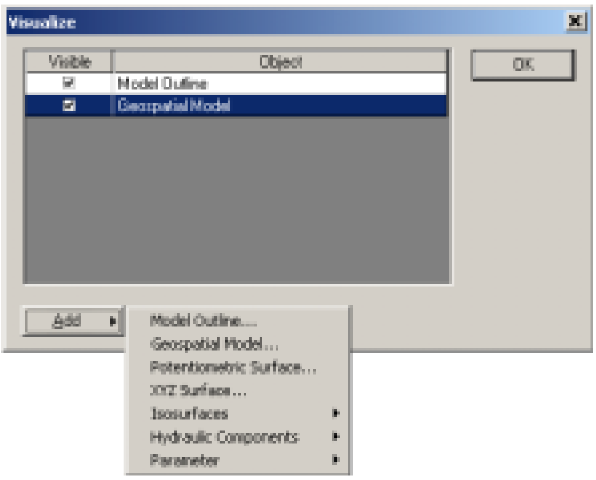


Figure 4.13: The **Add** button items.

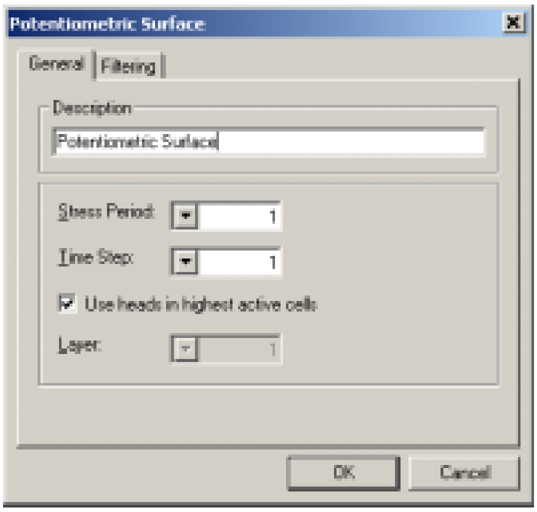


Figure 4.14: The **General** page of the **Potentiometric Surface** dialog box.

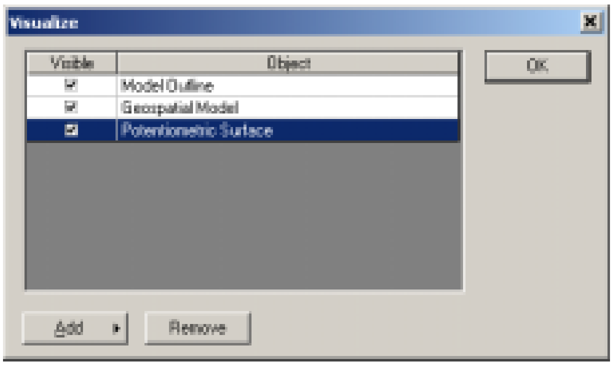


Figure 4.15: The **Visualize** dialog box with Potentiometric Surface added.

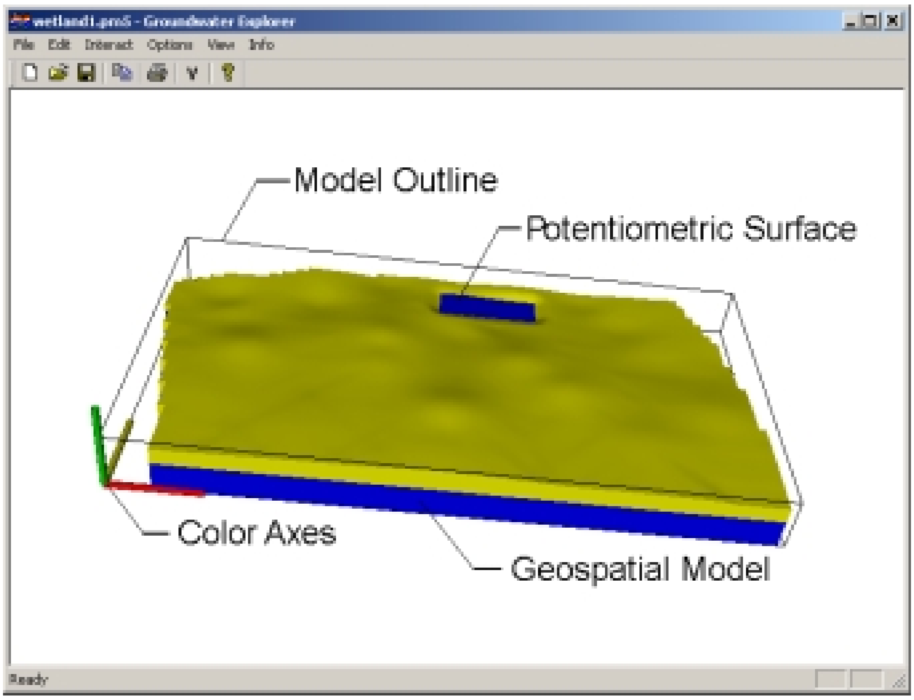


Figure 4.16: The Potentiometric Surface, Color Axes, Model Outline and Geospatial Model visualization objects.

4.3 Concentration Isosurface

Assume that the aquifer upstream of the well field is highly contaminated with Benzene and the solute concentration at the model cells [59, 39, 1], [59, 40, 1], [59, 39, 1] and [59, 39, 1] is (and remains) 10000 $\mu g/l$. According to the U.S. EPA, the maximum contaminant level (MCL) of Benzene for drinking water is 5 $\mu g/l$. If the predicted concentration at the pumping well is higher than the MCL, something must be done to protect the wells. The following sections describe the steps to display the concentration isosurface.

4.3.1 Step 1: Start GE

Select **Groundwater Explorer** from the **Programs** menu list.

4.3.2 Step 2: Open the Wetland Model

Follow the same steps as in section 4.2.2 (page 36), but open the model contained in the *...\models\wetland5* file folder on the accompanying CD, and select the *wetland5.pm5* file to open.

4.3.3 Step 3: Add Time Dependent Hydraulic Component

In order to know if if the maximum allowed concentration of Benzene reaches the well field. We need to add the discharge wells as a visualization object to the scene. Follow the steps below:

1. Select **Visualize…** from the Options menu to display the **Visualize** dialog box (Figure 4.17) from which time-dependent **Hydraulic Components** visualization object, such as wells or rivers, can be added.
2. Click the  button to display the **Add** button items (Figure 4.18). A new time-dependent **Hydraulic Components** visualization object is added by selecting **Hydraulic Components > Time Dependent**, which displays the **Hydraulic Component (Time Dependent)** dialog box (Figure 4.19). Keep the defaultx and click the  button. The time-dependent **Hydraulic Components** object is added to the **Visualize** dialog box (Figure 4.20).

3. In the **Visualize** dialog box, click the  button to display the result (Figure 4.21).

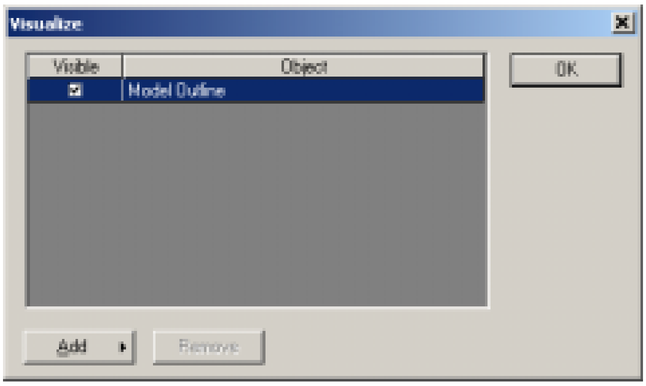


Figure 4.17: The **Visualize** dialog box.

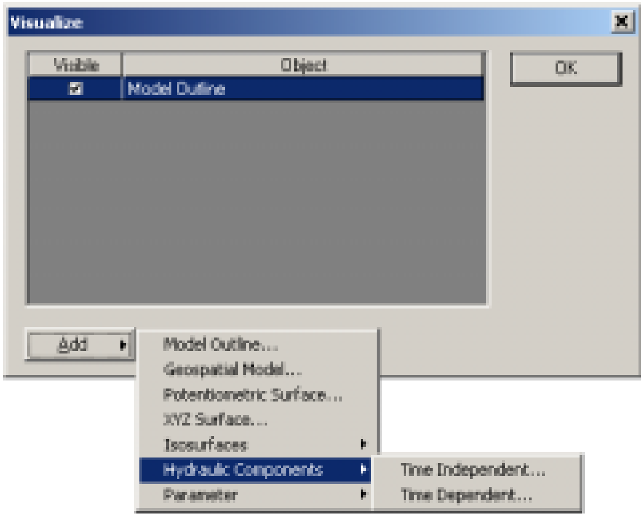


Figure 4.18: The **<u>A</u>dd** button items.

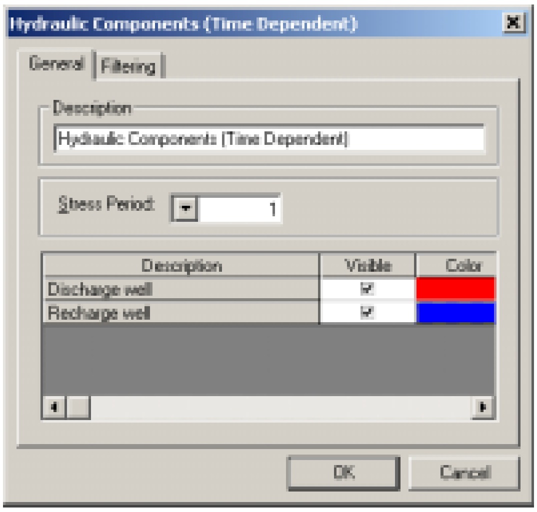


Figure 4.19: The **Hydraulic Components (Time Dependent)** dialog box.

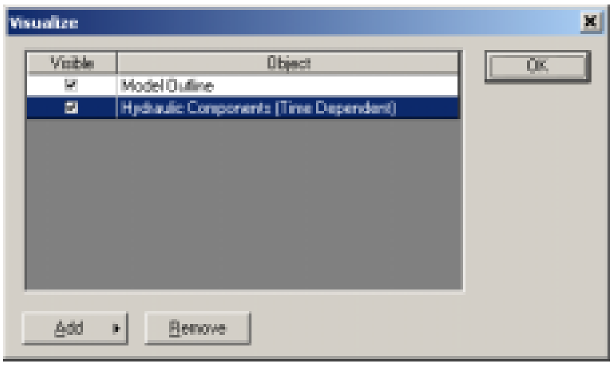


Figure 4.20: The **Visualize** dialog box with a **Hydraulic Components (Time Dependent)** visualization object added.

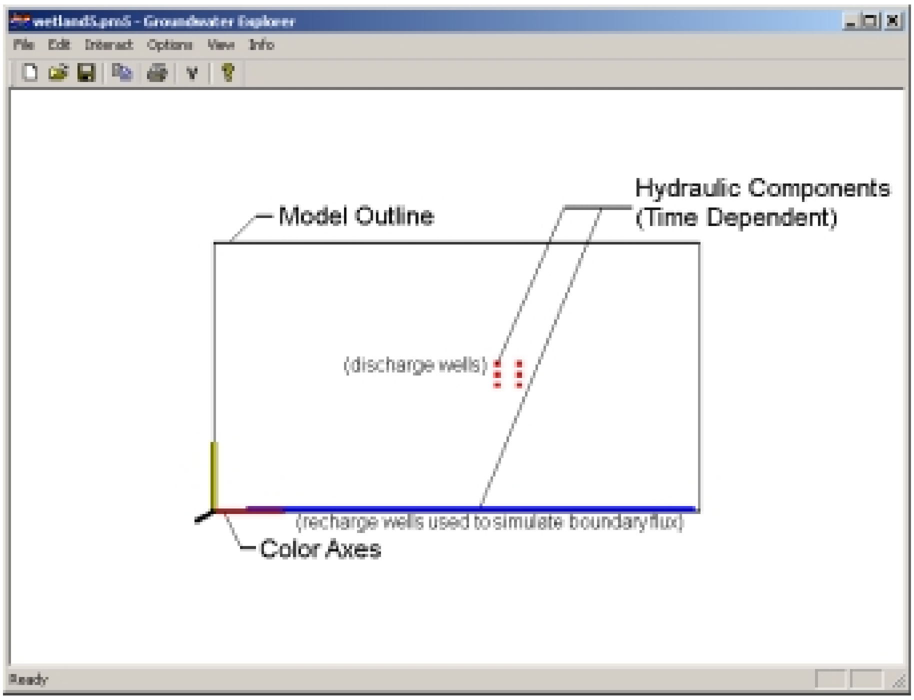
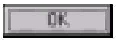


Figure 4.21: The Hydraulic Components (Time Dependent), Color Axes and Model Outline visualization objects.

4.3.4 Step 4: Add Isosurface

Follow the steps below to add a Concentration Isosurface.

1. Select **Visualize…** from the **Options** menu to display the **Visualize** dialog box (Figure 4.22), from which the Isosurface visualization object can be added.
2. Click the  button to display the **Add** button items (Figure 4.23). Click on **Isosurface>MT3DMS…** to add an isosurface object using the concentration values calculated by MT3DMS. The **Concentration Isosurface (MT3DMS)** dialog box (Figure 4.24) appears.

3. Select 473400000.000000 from the **Total Elapsed Time** dropdown list, accept the default values for all other boxes, and click the  button. The Concentration Isosurface (MT3DMS) is added to the **Visualize** dialog box (Figure 4.25).
4. In the **Visualize** dialog box, click the  button to see the result (Figure 4.26).

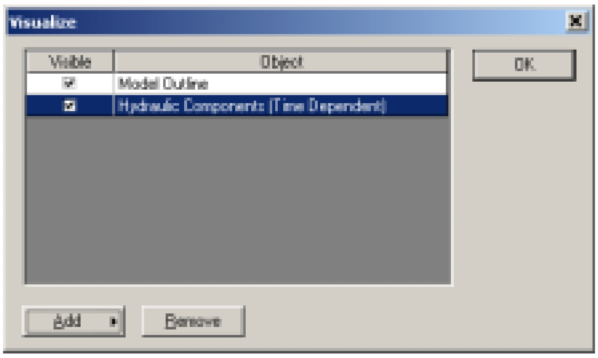


Figure 4.22: The **Visualize** dialog box.

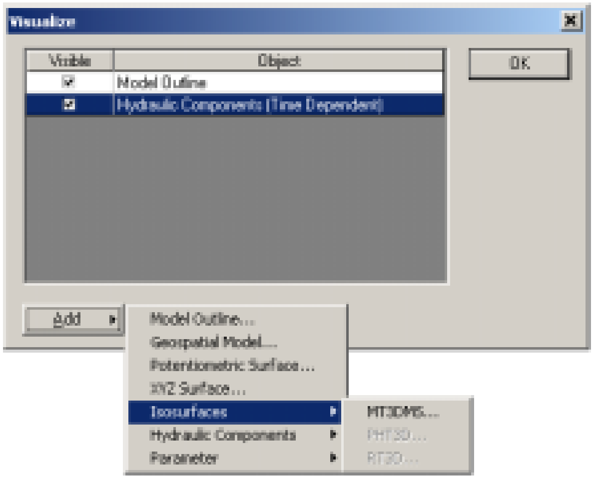


Figure 4.23: The **Add** button items.

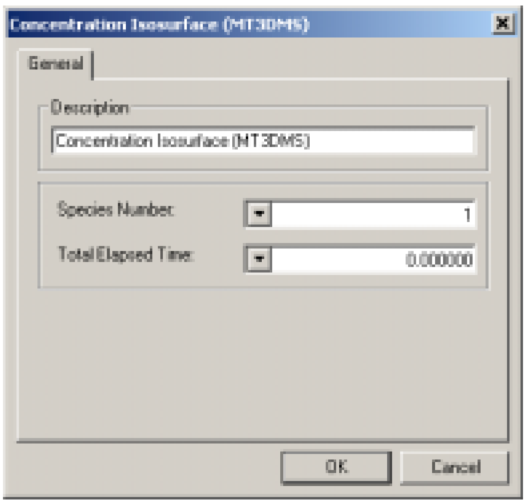


Figure 4.24: The **Concentration Isosurface (MT3DMS)** dialog box.

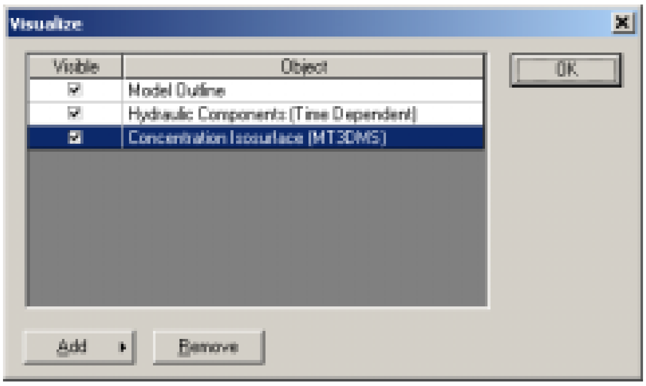


Figure 4.25: The **Visualize** dialog box with **Concentration Isosurface (MT3DMS)** visualization object added.

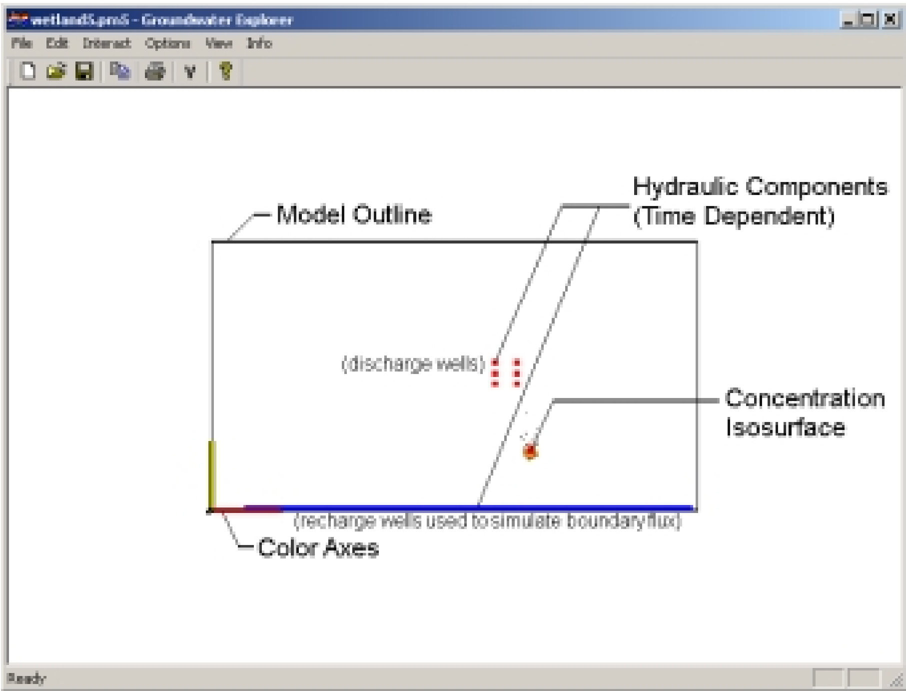


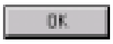
Figure 4.26: The Concentration Isosurface (MT3DMS), Color Axes, Model Outline and Hydraulic Components visualization objects.

4.3.5 Step 5: Scale

Follow the same steps as in section 4.2.4 (page 40) to set the scale factor for the z-axis to 20.

4.3.6 Step 6: Change Contours

According to USEPA, the maximum contaminant level (MCL) for Benzene in drinking water is 5 μg/l. Follow the steps below to change the contour level of the existing concentration isosurface to 5.

1. Select **Visualize…** from the **Options** menu to display the **Visualize** dialog box, from which the properties of the existing **Concentration Isosurface (MT3DMS)** object can be modified.
2. Double-click **Concentration Isosurface (MT3DMS)** to display the **Concentration Isosurface (MT3DMS)** dialog box (Figure 4.27). Note that only the **Description** on the **General** page can be changed and an additional **Contour** page appears. If other isosurfaces are to be displayed for a different **Species Number** or a different **Total Elapsed Time**, add a new Isosurface visualization object from the **Visualize** dialog box with the **Add** button.
3. Click the **Contour** tab to display the contour levels of the concentration isosurfaces (Figure 4.28).
4. Click the **Contour…** column header button to display the **Contour Level** dialog box (Figure 4.29) and change the values of the **Minimum** and **Maximum** boxes to **5** and the value of the **Interval** box to **0**. Click the  button. The contour level is set to 5 (Figure 4.30). In the **Contration Isosurface (MT3DMS)** dialog box, click the  button to accept the contour level.
5. In the Visualize dialog box, click the  button to display the result. Figure 4.31 shows a rotated and zoomed result of the scene.

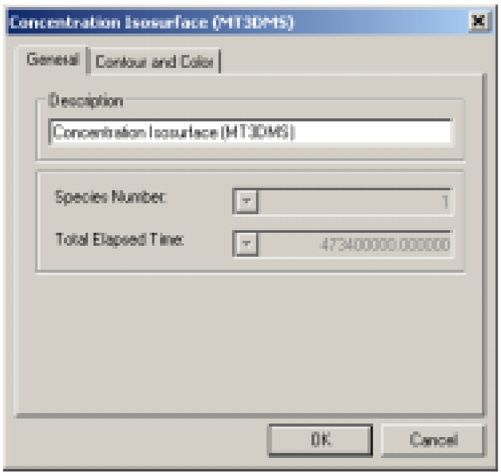


Figure 4.27: The **Concentration Isosurface (MT3DMS)** dialog box.

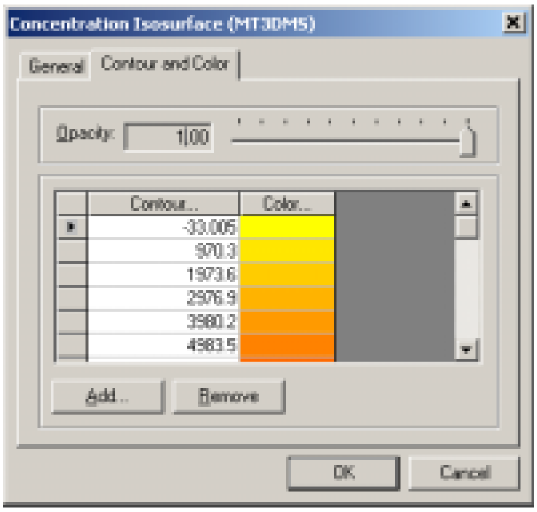


Figure 4.28: The **Contour and Color** page.

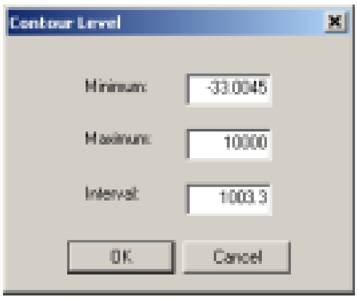


Figure 4.29: The **Color Level** dialog box.

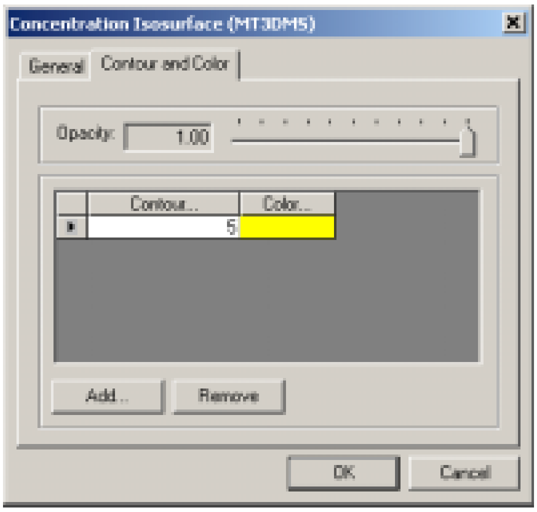


Figure 4.30: The **Contour and Color** page of the **Contration Isosurface (MT3DMS)** dialog box.

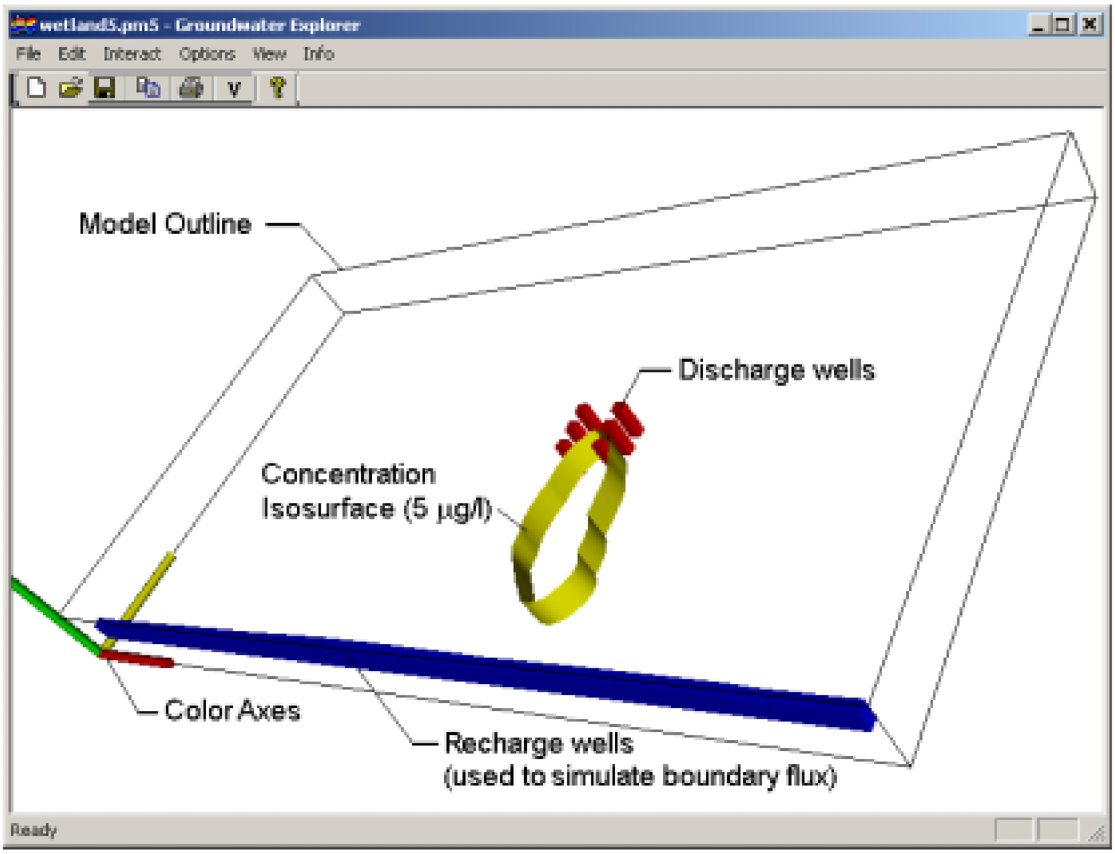


Figure 4.31: The 5 µg/l Concentration Isosurface and other visualization objects.

References

Brooks, F.P. (1996). *The computer scientist as Toolsmith II.* Communications of the ACM, March 1996, Vol. 39, No. 3.

Botha, J.F., Verwey, J.P., Van der Voort, I., Vivier, J.J.P., Buys, J., Colliston, W.P. and Loock, J.C. (1998). *Karoo Aquifers. Their Geology, Geometry and Physical Properties.* WRC Report No TT 73/95. Water Research Commision, P.O. Box 824, Pretoria 0001.

Chiang, W.H. and Kinzelbach, W. (2001). *3D Groundwater Modeling with PMWIN*. Springer, Berlin, Heidelberg, New York. ISBN 3-540-67744-5

Clement, T.P. (1997). *RT3D: A Modular Computer Code for Simulating Reactive Multi-Species Transport in 3-Dimensional Groundwater Sytems.* The U.S. Department of Energy. Contract DE-AC06-76RLO 1830. Pacific Northwest National Laboratory Richland, Washington, 99352.

Prommer, H. (2000, in preparation). *A coupled PHREEQC and MT3DMS for simulating multi-species reactive transport in groundwater.* Personal contact.

Stifter, J. (2000). *vtk Documentation 3.0.* Online vtk Documentation generated by Doxygen written by Dimitri van Heesch, © 1997-1999.

Schroeder, W.J., Avila, L.S. and Hoffman, W. (2000). *Visualizing with VTK: A Tutorial.* IEEE Computer Graphics and Applications, September/October.

Schroeder, W. J., Martin, K. M. and Lorensen, W. E. (1998a). *The Design and Implementation Of An Object-Oriented Toolkit For 3D Graphics and Visualization*. IEEE.

Schroeder, W., Martin, K. and Lorensen, B. (1998b). *The Visualization Toolkit. An Object-Oriented Approach to 3D Graphics. (2nd ed.).* Prentice Hall PTR. Upper Saddle River, NJ.

Schroeder, W.J. and Martin, K.M. (1998c). *User's Guide***.** Kitware, Inc.

Zheng, C. (1990). *MT3D. A Modular Three-Dimensional Transport Model.* S.S. Papadopulos & Associates, Inc, Rockville, Maryland.

Zheng, C. and Wang, P.P. (1998). *MT3DMS. A Modular Three-Dimensional Multispecies Transport Model.* Waterways Experiment Station, US Army Corps of Engineers, Vicksburg, Mississippi, 39187.

Zheng, C. and Wang, P.P. (1999). *MT3DMS. A modular three-dimensional multispecies model for simulation of advection, dispersion and chemical reactions of contaminants in groundwater systems; Documentation and User's Guide.* Contract Report SERDP-99-1, U.S. Army Engineer Research and Development Center, Vicksburg, MS.

Glossary

Clipping Plane. A plane that restricts the rendering or processing of data. Front and back clipping planes are commonly used to restrict the rendering of primitives to those lying between the two planes.

Color Mapping. A scalar visualization technique that maps scalar values into color. Generally used to display the variation of data on a surface or through volume.

Contouring. A scalar visualization technique that creates lines (in 2D) or surfaces (in 3D) representing a constant scalar value across a scalar field. Contour lines are called isovalue lines or isolines. Contour surfaces are called isovalue surfaces or isosurfaces.

Cutting. A visualization technique to slice through or cut data. The cutting surface is typically described with an implicit function, and data attributes are mapped onto the cut surface.

Data Visualization. The process of transforming data into sensory stimuli, usually visual images. Data visualization is general term, encompassing data from engineering and science, as well as information from business, finance, sociology, geography, information management, and other fields. Data visualization also includes elements of data analysis, such as statistical analysis.

Decimation. A type of polygon reduction technique that deletes points in a polygonal mesh that satisfies a co-planar or co-linear condition and replaces the resulting hole with new triangles.

Delaunay Triangulation. A triangulation that satisfies the Delaunay circumsphere criterion. This criterion states that a circumsphere of each simplex in the triangulation contains only the points defining the simplex.

Information Visualization. The process of transforming information into sensory stimuli, usually visual images. Information visualization is used to describe the process of visualizing data without structure, such as information on the World Wide Web; or abstract data structures, like computer file systems or documents.

Isosurface. A surface representing a constant valued scalar function.

Normal. A unit vector that indicates perpendicular direction to a surface. Normals are a common type of data attribute.

Polygon. A cell consisting of three or more co-planar points defining a polygon. The polygon can be concave but without imbedded loops.

Scene. A complete representation of the components required to generate an image or animation including lights, cameras, actors, properties, transformations, geometry, texture, and other pertinent information.

Scientific Visualization. The process of transforming data into sensory stimuli, usually visual images. Generally used to denote the application of visualization to the sciences and engineering.

Visualization. The process of transforming data to images (or other sensory stimuli). Alternatively, the end result of the visualization process.

VRML. VRML is an acronym for the Virtual Reality Modeling Language. Technically speaking, VRML is neither *virtual reality* nor a *modeling language*. At its core, VRML is simply a 3D interchange format. It defines most of the commonly used semantics found in today's 3D applications such as hierarchical transformations, light sources, viewpoints, geometry, animation, fog, material properties, and texture mapping. One of the primary goals in designing VRML was to ensure that it at least succeeded as an effective 3D file interchange format.

Zoom. A camera operation that changes the field of view of the camera.