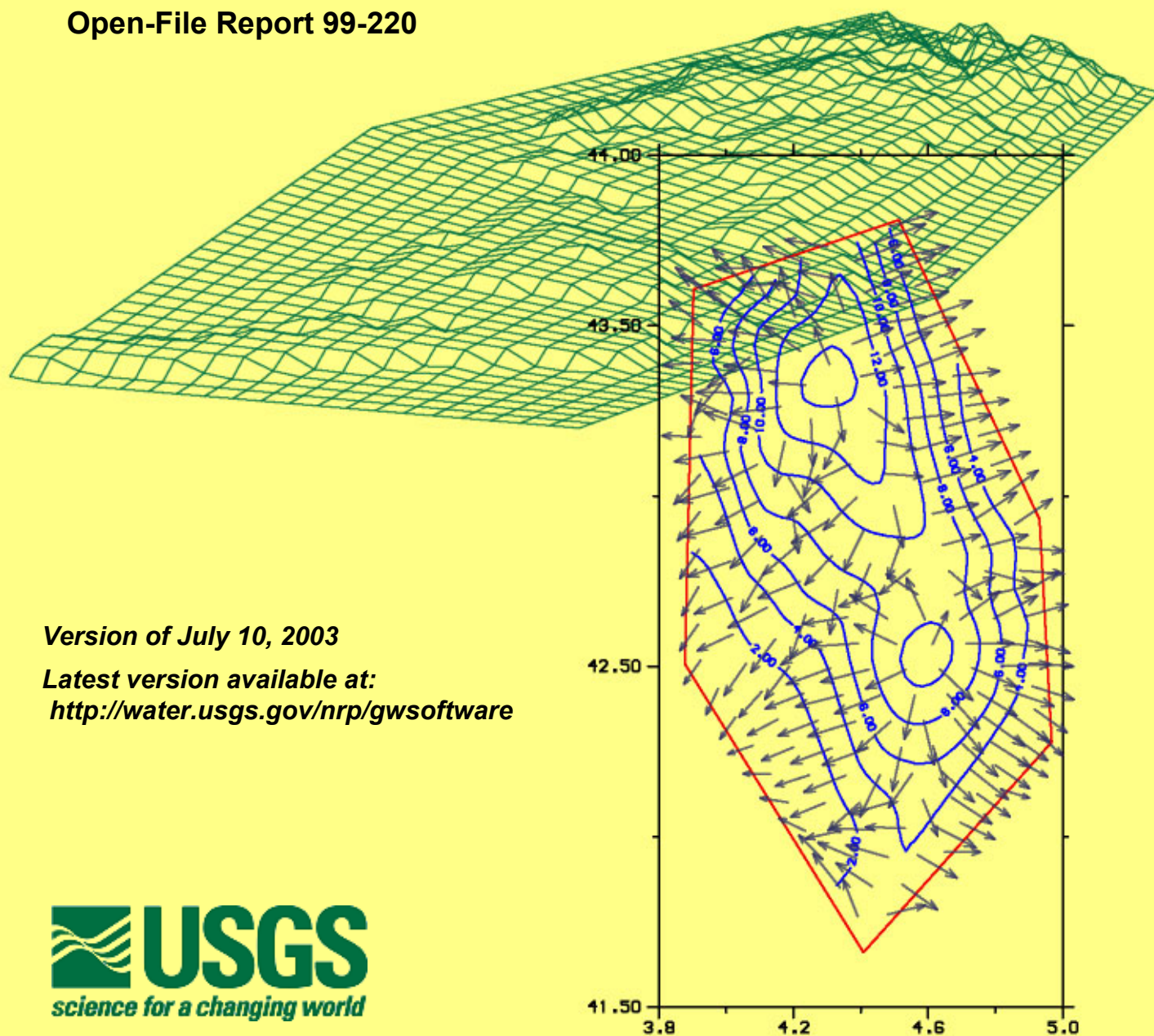


# SutraPlot

A graphical post-processor for SUTRA, a model for  
ground-water flow with solute or energy transport

Open-File Report 99-220



Version of July 10, 2003

Latest version available at:  
<http://water.usgs.gov/nrp/gwsoftware>



# **SutraPlot, a graphical post-processor for SUTRA, a model for ground-water flow with solute or energy transport**

**By William R. Souza**

---

**U. S. GEOLOGICAL SURVEY  
Open-File Report 99-220**

*This report supersedes  
U.S. Geological Survey Water-Resources Investigations Report 87-4245*

---

***Version of June 10, 2003***

*Latest version available at <http://water.usgs.gov/nrp/gwsoftware>*

**Reston, Virginia  
1999**

**U.S. Department of the Interior**  
GALE A. NORTON, Secretary

**U.S. Geological Survey**  
Charles G. Groat, Director

Although the computer program described in this report has been tested and used by the U.S. Geological Survey (USGS), no warranty, expressed or implied, is made by the USGS as to the accuracy of the functioning of the program and related material. The code and report may be changed without notice.

The use of trade, product, or firm names in this report is for descriptive purposes only and does not imply endorsement by the U.S. Government.

---

For additional information write to:

SUTRA Support  
U.S. Geological Survey  
431 National Center  
Reston, VA 20192  
USA

Copies of this report can be  
purchased from:

U.S. Geological Survey  
Branch of Information Services  
Box 25286  
Denver, CO 80225-0286  
USA

This document and the SutraPlot computer code may be downloaded without cost from a U.S. Geological Survey Internet site: <http://water.usgs.gov/nrp/gwsoftware>

# CONTENTS

ABSTRACT .....	1
INTRODUCTION .....	2
Availability of SutraPlot code .....	3
System Requirements and Installation .....	3
STEP-BY-STEP APPLICATIONS .....	4
SUTRA simulation problem .....	4
Starting SutraPlot .....	5
Application 1 - 2D section contour plot from 3D SUTRA simulation .....	5
Application 2 - Velocity vector plot .....	7
Application 3 – 3D Mesh Plot .....	9
USER’S GUIDE .....	11
Notes on Starting and Using SutraPlot .....	11
Contour Plots .....	11
Creating a new contour plot .....	11
Default settings .....	12
The Contour Plot Dialog Box .....	12
Selecting a time step .....	12
Viewing the nodewise (.nod) output file header .....	13
Selecting a plot variable .....	13
Selecting a 2D slice (3D SUTRA simulations only) .....	13
Displaying the axis labels .....	13
Displaying the finite-element mesh boundary .....	14
Using the Zoom function .....	14
Specifying Contour Levels .....	14
Using Data Scaling Factors .....	14
Specifying Contour Label Properties .....	15
Scaling the plot coordinate axes .....	15
Changing the gridding density .....	15
Using Setup files .....	16
Velocity vector plots .....	16
Creating a new vector plot .....	16
Default settings .....	17
The Vector Plot Dialog Box .....	17
Selecting a time step .....	17
Viewing the elementwise (.ele) output file header .....	17
Selecting a projection plane (3D SUTRA simulations only) .....	18
Using the Zoom function .....	18
Using Vector Thinning .....	19
Setting Vector Properties .....	19
Scaling the plot coordinate axes .....	20
Using Setup files .....	21
Finite-element mesh plot .....	21
Creating a new mesh plot .....	21
Default settings .....	22
The 2D Mesh Plot Dialog Box .....	22

Displaying the axis labels.....	22
Using the Zoom function.....	22
Scaling the plot coordinate axes.....	22
The 3D Mesh Plot Dialog Box.....	23
Selecting a viewing direction.....	23
Selecting a drawing option.....	23
Selecting a projection type.....	23
Selecting face visibility.....	23
Rotating the mesh.....	23
Scaling the plot coordinate axes.....	23
Using the Zoom function.....	23
SutraPlot Export files.....	24
Gridded data files.....	24
The Gridded Output file Dialog Box.....	24
Selecting a time step.....	24
Viewing the nodewise (.nod) output file header.....	24
Selecting a plot variable.....	25
Specifying the grid density.....	25
Selecting a grid dummy value.....	25
AutoCAD (.dxf) export files.....	25
REFERENCES.....	26
APPENDIX A – Finite-element interpolation methodology.....	27
APPENDIX B - Vector magnitude scaling and direction adjustment.....	31
APPENDIX C - Using mesh specifications to define a region of vectors.....	33

## FIGURES

1. Schematic diagram of the finite-element mesh for the sample **SUTRA** problem.
- 2-4. Plot showing:
  2. 2D concentration contour output from Application 1.
  3. 2D velocity vector output from Application 2.
  4. 3D finite-element mesh from Application 3.
5. Graphic representation of velocity vectors that have been scaled using a logarithmic function.
6. Diagram showing, (a) velocity vectors with equal x and y scaling, plotted on schematic finite-element mesh. (b) velocity vectors re-plotted in bottom row of elements with exaggerated (5:1) y-axis.

# **SutraPlot, a graphical post-processor for SUTRA, a model for ground-water flow with energy or solute transport**

By William R. Souza

## **ABSTRACT**

This report documents a graphical display post-processor (**SutraPlot**) for the U.S. Geological Survey Saturated-Unsaturated flow and solute or energy TRAnsport simulation model **SUTRA**, Version 2D3D.1. This version of **SutraPlot** is an upgrade to **SutraPlot** for the 2D-only **SUTRA** model (Souza, 1987). It has been modified to add 3D functionality, a graphical user interface (GUI), and enhanced graphic output options.

Graphical options for 2D **SUTRA** (2-dimension) simulations include: drawing the 2D finite-element mesh, mesh boundary, and velocity vectors; plots of contours for pressure, saturation, concentration, and temperature within the model region; 2D finite-element based gridding and interpolation; and 2D gridded data export files.

Graphical options for 3D **SUTRA** (3-dimension) simulations include: drawing the 3D finite-element mesh; plots of contours for pressure, saturation, concentration, and temperature in 2D sections of the 3D model region; 3D finite-element based gridding and interpolation; drawing selected regions of velocity vectors (projected on principal coordinate planes); and 3D gridded data export files.

Installation instructions and a description of all graphic options are presented. A sample **SUTRA** problem is described and three step-by-step **SutraPlot** applications are provided. In addition, the methodology and numerical algorithms for the 2D and 3D finite-element based gridding and interpolation, developed for **SutraPlot**, are described.

## **INTRODUCTION**

**SutraPlot** Version 2D3D.1 is a graphics post-processor for **SUTRA**, U. S. Geological Survey Saturated-Unsaturated flow and solute or energy TRAnsport simulation model, 2D3D.1 (Voss and Provost, 2002). This version of **SutraPlot** is an upgrade to **SutraPlot** for the 2D-only **SUTRA** model (Souza, 1987). It has been modified to add 3D functionality, a graphical user interface (GUI), and enhanced graphic output options. **SutraPlot** Version 2D3D.1 is **not** compatible with earlier versions of **SUTRA** (Voss, 1984) or with earlier **SutraPlot** setup files.

In this release, **SutraPlot** is distributed as an executable code that runs under Microsoft Windows NT/2000<sup>®</sup> or Windows 95/98/XP<sup>®</sup> operating systems. The source code is written in Fortran-90, compiled with Digital<sup>®</sup> Fortran Version 6.6, and linked to the GINO<sup>®</sup> Version 4.2 and GINOSURF<sup>®</sup> Version 3.2 graphics libraries, and GINOMENU<sup>®</sup> Version 4.0 GUI. This document describes how to use **SutraPlot** 2D3D.1 as a graphical post-processor for **SUTRA**, including descriptions of all graphical options and three step-by-step SutraPlot applications.

**SutraPlot** generates graphic output for both 2D (2-dimensional) and 3D **SUTRA** (3-dimensional) simulations. All graphic output can be saved in vector format (*.dxf* files) or in bitmap format (via the Windows<sup>®</sup> clipboard *.bmp* files). Gridded data, used for contouring, can be saved for use in other plotting applications.

SutraPlot is designed to provide quick, accurate graphical representation of output generated by the SUTRA model. It is not the intent of the program to produce publication quality graphics.

Graphics from 2D SUTRA simulations include:

- plots of contours for pressure, saturation, concentration, and temperature within the model region
- plots of velocity vectors within the model region
- drawing the finite-element mesh and mesh boundary
- 2D finite-element based gridding and interpolation
- 2D gridded data export file

Graphics from 3D SUTRA simulations include:

- plots of contours for pressure, saturation, concentration, and temperature in 2D sections of the 3D model region
- plots of selected regions of velocity vectors (projected in 2D on the principal coordinate planes)
- drawing the 3D finite-element mesh
- 3D finite-element based gridding and interpolation
- 3D gridded data export file



## Availability of SutraPlot code

The source code and for **SutraPlot** is in the public domain and may be obtained free of charge from the U.S. Geological Survey and may be redistributed freely. However, the current release of **SutraPlot** must be compiled with the commercial graphics libraries GINO<sup>®</sup>, GINOMENU<sup>®</sup>, and GINOSURF<sup>®</sup>. Under the licensing agreement with Bradley Associates (www.bradassoc.co.uk), the USGS has been granted permission to freely distribute the compiled SutraPlot code containing GINO<sup>®</sup> suite of graphics modules.

The **SutraPlot** source code, this documentation, and sample datasets, along with the **SUTRA** and **SutraPrep** codes referred to in this report can be obtained free of charge from a USGS Web site:

<http://water.usgs.gov/nrp/gwsoftware>

When this code is revised or updated in the future, new versions or releases will be made available at this web site.

## System Requirements and Installation

The executable code for **SutraPlot** 2D3D.1 provided in this release will run under Windows 95/98/XP<sup>®</sup> or Windows NT/2000<sup>®</sup> operating systems. This version utilizes dynamic array allocation and may be used for any size **SUTRA** finite-element mesh or simulation output up to the capacity of the users computing system.

Many large **SutraPlot** applications that include gridding of data for contour plots may require considerable array storage and computational effort. These applications are most effectively carried out on Pentium<sup>®</sup> III or faster computers with a minimum of 128 Mbytes of RAM. The system memory requirements are similar to those needed to run **SUTRA**. For example, the 3D sample simulation (BF3D) described in this documentation has 26,691 nodes and 24,000 elements. The system memory requirement for running **SutraPlot** with BF3D sample applications is about 17 Mbytes.

SutraPlot is distributed by the USGS as part of the **SUTRA** 2D3D.1 release package. Refer to the *ReleaseNotes.doc* file included in the SutraPlot directory for additional installation and setup instructions, suggested working directory structures, and the most current list and description of SutraPlot files.

## STEP-BY-STEP APPLICATIONS

This section introduces some of the SutraPlot features and shows the steps required to produce three plots using the **SUTRA** simulation problem described below. The rest of the sections contain more information about the graphic options not covered here. It is helpful, but not necessary, to follow the step-by-step examples in sequence.

Contour and velocity plots from 3D **SUTRA** simulations output to a graphics window are displayed on 2D sections of the 3D mesh. When using these options, the user must be able to locate the sections in 3D space and thus, must be familiar with the finite-element mesh geometry of the **SUTRA** problem. Generally, it is useful to have a diagram of the **SUTRA** mesh that shows the orientation of the mesh and the dimensions of the mesh relative to the **SUTRA** X-Y-Z coordinate system (see [Figure 1](#)). If the construction of the mesh is blockwise, it is also useful to identify the order of the node numbering directions of the finite-element mesh (See Appendix C).

### SUTRA simulation problem

The step-by-step applications use a 3D **SUTRA** simulation of a system similar to a problem described by Burnett and Frind (1987). A detailed description of the problem geometry and parameters is described in the release notes that accompany the 3D example problem distributed with **SUTRA**. The BF example uses a nearly rectangular 3D finite-element mesh with 26,691 nodes and 24,000 elements, where the top surface is an inclined plane ([Figure 1](#)). The problem simulates a hypothetical aquifer with uniform hydrologic properties. Recharge takes place over the entire top surface along with a contaminant source represented by a constant non-zero solute concentration over a small area of the surface. In the simulation, a steady-state flow field is generated. In the sample **SUTRA** output, nodewise concentration values define a steady-state 3D solute plume.

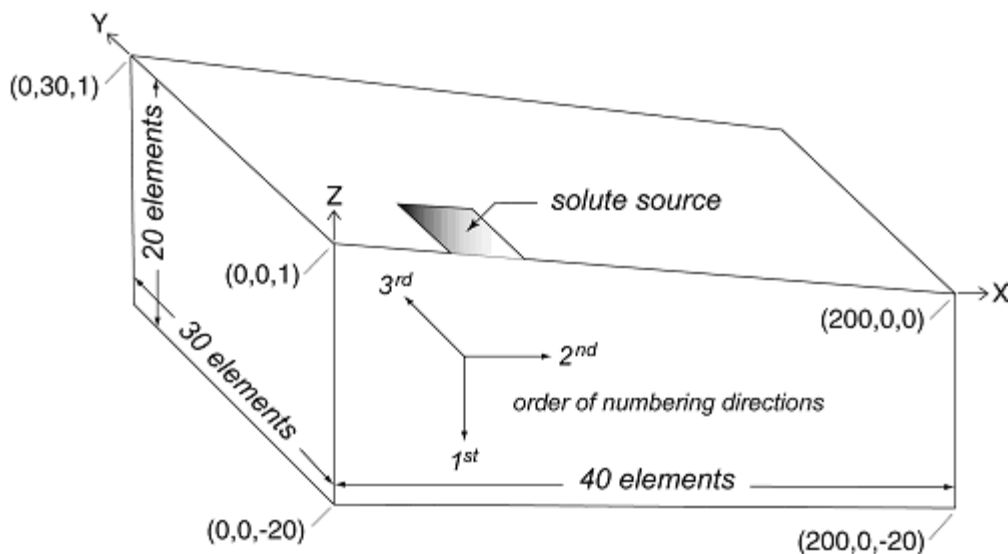


Figure 1. Schematic diagram of the finite-element mesh for the sample problem showing the orientation of the mesh relative to the **SUTRA** X-Y-Z coordinates, mesh dimensions, node numbering directions, and number of elements in each direction.

## Starting SutraPlot

- ✓ Start **SutraPlot** by double clicking on the **SutraPlot** icon (if a desktop shortcut has been created), or by double-clicking on the file *supl.exe* where it appears in any file manager. **SutraPlot** starts with an empty graphics window.
- ✓ To start a new plot session, select the **File | Load SUTRA Input File** command from the main menu. A standard Windows file browser dialog is opened. Filenames with the *.inp* extension are automatically displayed. Locate the directory containing *bf.inp*, and open the file. Files can be selected and opened by clicking on the filename in the directory listing, then clicking the Open button, or by double-clicking on the filename.
- ✓ The main input file is opened, read and **SutraPlot** displays the **Setup for 3D Simulation** dialog. This window allows the user to select an initial graphic output option. Other options using the same **SUTRA** input file may be subsequently selected during the **SutraPlot** session.

## Application 1 - 2D section contour plot from 3D SUTRA simulation

1. Select **Contours (2D section)**. As required for a contour plot, this selection opens a file browser dialog. Locate and open the **SUTRA** nodewise results file, *bf.nod*.
2. **SutraPlot** reads the *bf.inp* file to determine the number of nodes and elements, and reads in the node coordinates and incidence list, checks for compatibility with the *bf.nod* file, then opens the **Contour Plot** dialog. This is the main dialog window for a 2D contour section of a 3D **SUTRA** simulation.
3. Note that the main graphics window is now active (indicated by a change from a gray to a white background). The status bar at the bottom of the window displays the model horizontal and vertical screen coordinates in the second and third panels as the cursor moves across the window. The fourth panel is inactive at this point. The fifth panel displays the pathnames of the **SUTRA** *.inp* and *.nod* files currently in use.
4. **SutraPlot** assumes default parameters for most settings to create an acceptable contour plot. The user must select two parameters. First, select a plot variable by clicking on **Concentration** in the **Contour Plot** dialog.
5. Next, a 2D slice through the 3D mesh must be selected. Click on **Orthogonal Plane**. The **2D Slice Selection** dialog is displayed. [For more information regarding slice selection, see **2D Slice Selection** in the **User' Guide** section.]
6. In the **2D Slice Selection** dialog, select the **X-Z Coordinate Plane** from the drop-down list box. Once a plane is selected, the location of the section along the third direction needs to be input, in this case, the **Y** direction. This problem was designed to simulate half of a domain with bilateral symmetry, and thus only half the solute plume was simulated. The plane of symmetry and the middle of the plume is in the **X-Z**

**plane** along the side of the mesh, at  $Y=0$ . Enter '0' in the **Distance along the Y-axis** value box.

7. Click **Apply** in either the **2D Slice Selection** or **Contour Plot** dialog.
8. **SutraPlot** now interpolates values at all grid points in the 2D slice and concentration contours are plotted in the graphics window using default settings. The finite-element boundary around the 2D section is displayed in red. Note: Click anywhere in the graphics window and the cursor changes from an arrow to a cross. The fourth panel in the status bar now displays the spot concentration value as the cursor is moved across the plot.
9. The zoom function is always active. Starting anywhere in the graphics frame, hold down the left mouse button and drag a rectangle around a small area of the plot. Release the mouse button, then click **OK** in the **Zoom to selected area** prompt. If the selected area extends outside of the plot area (as defined by the displayed axes), the zoomed area is automatically clipped at the boundary of the plot area. Click **Zoom out** in the **Contour Plot dialog** to return to the original view.
10. Several contour plot features can be modified. First, in the **Properties** menu select **Contour Plot** dialog and select **Axis scaling**. In the newly displayed **Contour Coordinate-Axis Scaling** dialog, click the check box beneath **Scaling factor** and across from "**Z**". Enter "5" in the adjacent value box. Click **Apply** and the plot is redrawn with the Z-axis scale expanded by a factor of 5.
11. In the **Properties** menu select **Contour Level**. In the newly displayed **Contour Level Properties** dialog, enter 1.0, 0.1, and 0.1 in the **Upper Level**, **Lower Level**, and **Contour Interval** value boxes, respectively. Click **Apply** and the plot is redrawn with the new contour levels. The plot in the graphics window should be similar to [Figure 2](#).
12. Next, in the **Contour Plot** dialog, select **Oblique Plane** under **Slice selection**. In the **2D Slice selection** dialog, select **Z-axis**. Then move the upper set of sliders to **X=0** and **Y=0**, and the lower set of sliders to **Y=0** and **X=200**. This represents the same plane that was plotted using the **Orthogonal plane** option. Click **Apply**.
13. The contours are redrawn. Because a new 2D slice was selected, the axis scaling is reset to the defaults. Open the **Coordinate-axis scaling** dialog and set the vertical scaling factor to 5. Click **Apply**. The new contours are identical to those in [Figure 2](#). Note that the axis labeling has changed. The coordinates for all oblique planes are translated and recomputed to that the origin (0,0) is at the lower left corner of the plot.
14. Try additional changes to the contour plot using the **Properties** menu. To continue and start Application 2, just leave the contour plot open and skip the next step.
15. To end Application 1 and close **SutraPlot**, click on **File** in the main menu in the graphics window and select **Exit**.

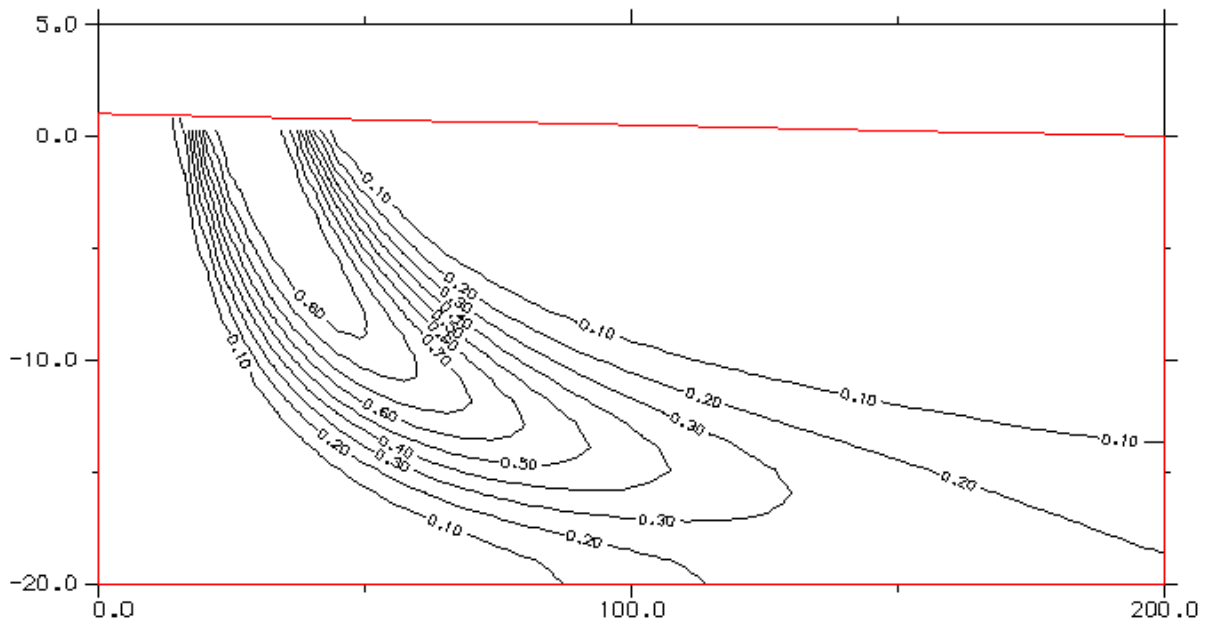


Figure 2. 2D orthogonal X-Z section, concentration contour plot produced from Step-by-step Application 1.

## Application 2 - Velocity vector plot

1. If you are continuing from Application 1, **skip this step**. If you are beginning the Applications here, start **SutraPlot** (see **Starting SutraPlot** above). Select **Velocity Vectors (2D section projected)**. As required for a vector plot, this selection opens a file browser dialog. Locate and open the **SUTRA** elementwise results file, *bf.ele* in the file browser. This opens the Vector Plot dialog, which is the main dialog window for a 2D vector plot of a region of vectors in a 3D **SUTRA** simulation. **Skip step 2**.
2. To continue from Application 1, go to the main menu, click on **Vector** and select **Open Vector Plot**. Locate and open the **SUTRA** elementwise results file, *bf.ele* in the file browser. This opens the **Vector Plot** dialog, which is the main dialog window for a 2D vector plot of a region of vectors in a 3D **SUTRA** simulation. Note: the contour plot created in Application 1 is still in memory but just hidden from view. Click on **View** in the main menu in the graphics window and select **Contour Plot**. Return to the vector plot by again clicking on **View** in the main menu and selecting **Vector Plot**.
3. **SutraPlot** assumes default parameters for most settings to create an acceptable vector plot. The user must select two parameters. First, choose a method to select a vector region by clicking on **Bounded box** in the **Vector Plot** dialog. Note: the **Vector Info** reminder to select a vector region will disappear when a region is selected. In the newly displayed **Bounded Box Constraints** dialog, enter 1.0, in the “**Y Maximum**” value box.

4. Under **Projection Plane** in the **Vector Plot** dialog, select the **X-Z Coordinate Pair** from the drop-down list box.
5. Click **Apply** in any of the displayed dialog boxes. Velocity vectors are plotted in the graphics window using default settings. A set of reference vectors is added below the plot. Click **Reference vectors** in the **Vector Properties** dialog. In the newly displayed **Set Reference Vector** dialog, enter a value of 3 e-7 in the value box in the **Maximum vector magnitude** panel. Note: the value may be entered in decimal format (.0000003) or scientific e-format (3 e-7).
6. In the **Properties** menu select **Axis scaling**. In the **Vector Coordinate-axis scaling** dialog, click on the check box beneath **Scaling factor** and across from “**Z**”. Enter “5” in the adjacent value box. Click **Apply** and the plot is redrawn with the **Z**-axis scale expanded by a factor of 5.
7. In the **Properties** menu select **Vector thinning**. In the **Vector thinning** dialog, click the first check box under **Thinning options** and enter “6” in the adjacent value box. Click **Apply** and the plot is redrawn with every 6<sup>th</sup> vector plotted.
8. In the **Properties** menu select **Vector Properties**. In the **Vector Properties** dialog, select **Logarithmic function** under **Vector scaling factors** (see Appendix B). Under **Vector size adjustment**, click the check box to the right of **Vector magnitude**. Enter “125” in the adjacent value box. The plot is redrawn with the new vector properties. The plot in the graphics window should be similar to [Figure 3](#).
9. In addition to the Bounding Box options used in Step 3, a region of vectors can be defined by a ‘logical sheet’. (For more information regarding vector regions, see **Selecting Vector Regions** and Appendix C). Select **Sheet** in the **Vector Plot** dialog.
10. In the **Sheet selection** dialog box, select **1<sup>st</sup> & 2<sup>nd</sup>** from the drop-down **Orientation** list box. In this sample problem, the node numbering directions 1, 2, and 3 correspond to and are aligned with the **SUTRA Z, X, and Y** coordinate axes. Selecting the 1<sup>st</sup> and 2<sup>nd</sup> numbering directions will display the vectors in the X-Z plane.
11. Enter “1” for the **Sheet number along 3<sup>rd</sup> direction**. This selects a set of vectors in the first sheet of elements along the side of the mesh between Y=0 and Y=1.
12. Click **Apply** and the plot is redrawn with the set of vectors. The set of vectors is the same as those selected using the **Bounding box** option, and the plot in the graphics window is again similar to [Figure 3](#).
13. Try additional changes to the vector plot using the **Properties** menu. To continue and start Application 3, just leave the vector plot open and skip the next step.
14. To end Application 2 and close **SutraPlot**, click on **File** in the main menu in the graphics window and select **Exit**.

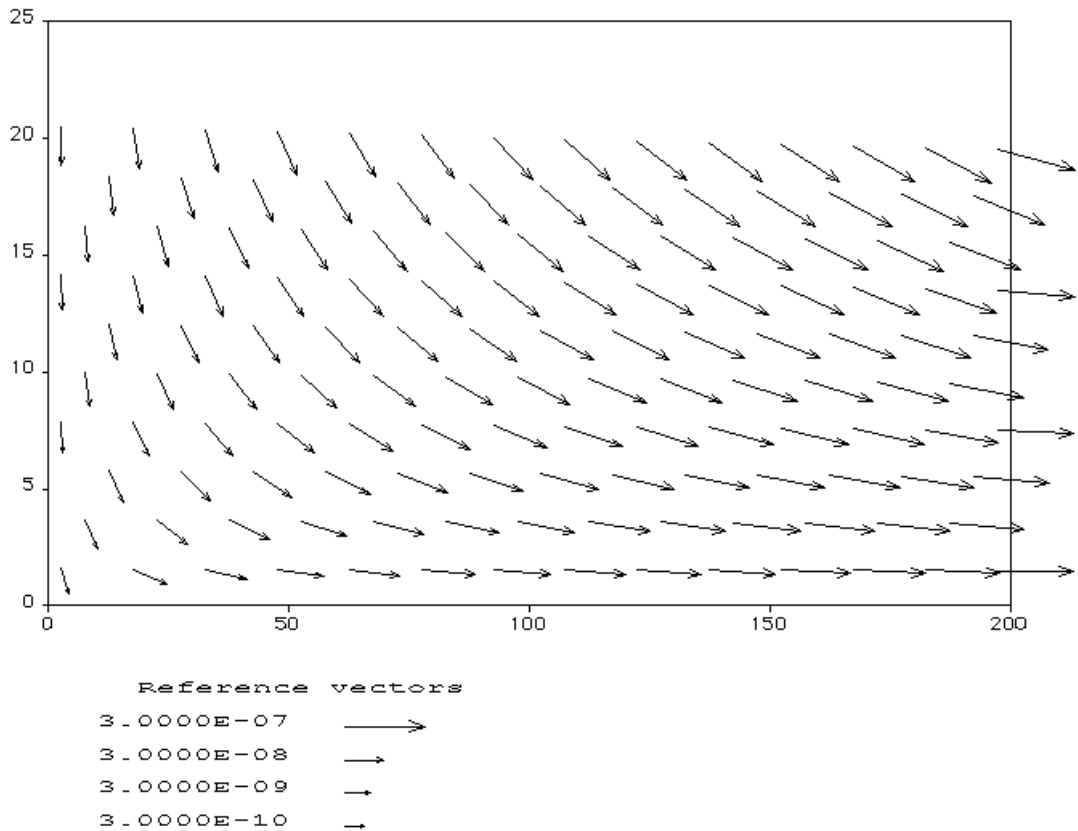


Figure 3. 2D velocity vector output from Step-by-step Application 2.

### Application 3 – 3D Mesh Plot

1. If you are continuing from Application 1 or 2, **skip this step**. If you are beginning here, start **SutraPlot** (see **Starting SutraPlot** above). Select **Finite-element mesh plot**. **SutraPlot** reads in the node coordinates and incidence list, then opens the **Mesh Plot** dialog. This is the main dialog window for a 3D mesh plot. **Skip step 2**.
2. To continue from Sample session 1 or 2, go to the main menu, click on **Mesh** and select **Open Mesh Plot**. This opens the **Mesh Plot** dialog, which is the main dialog window for a 3D mesh plot. Note: if other applications are open, they are still in memory but just hidden from view. Click on **View** in the main menu in the graphics window and select **Contour Plot** or **Vector Plot** if active. Return to the mesh plot by again clicking on **View** in the main menu and selecting **Mesh Plot**.
3. The elements on the six outside faces of the finite-element mesh are plotted in 3D. Note: The 3D mesh option has limited functionality and is supplied only as a tool for displaying general spatial relationships of the finite-element mesh.

4. In the **Properties** menu select **Axis scaling**. In the **Mesh Coordinate-axis scaling** dialog, click on the check box beneath **Scaling factor** and across from “**Y**” and “**Z**”. Enter “2” in both adjacent value boxes. Click **Apply**.
5. Click the check boxes next to **ND 1 MAX** and **ND 3 MIN** (**ND** refers to the **SutraPrep** Numbering Directions; see Appendix C). The mesh is redrawn without the faces at the end of the first numbering direction, and at the beginning of the third numbering direction. The plot in the graphics window should be similar to [Figure 4](#).
6. Click the check box next to **Show rotation guide**. Press the x, y, and z keys, to rotate the mesh around the x, y, and z-axes, respectively.
7. Click **+X** in the **Set viewing direction** panel. The plot is rotated so that the viewing direction is along the X-axis looking in the positive X direction.
8. Try additional changes to the mesh plot using the features in the Mesh plot dialog.
9. To end application 3 and close **SutraPlot**, click on **File** in the main menu in the graphics window and select **Exit**.

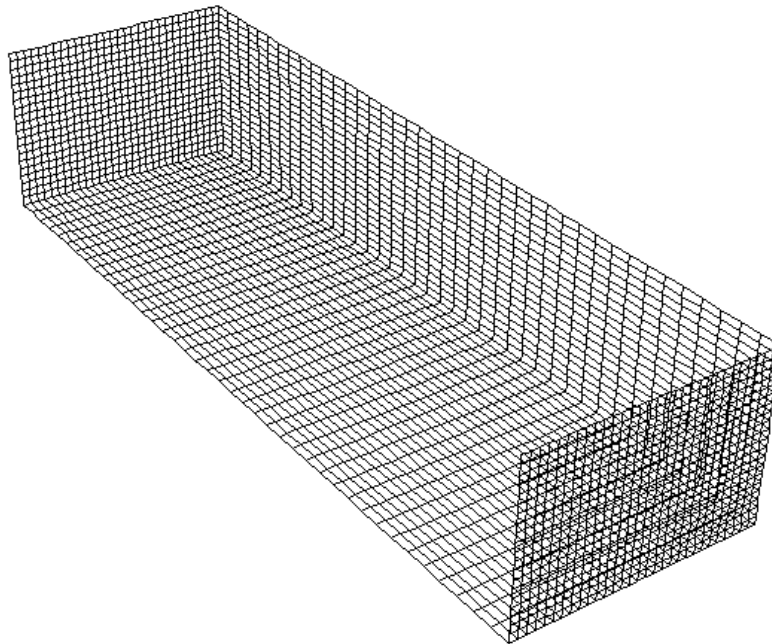


Figure 4. 3D mesh output from Step-by-step Application 3.



## USER'S GUIDE

### Notes on Starting and Using SutraPlot

- **SutraPlot** is normally started by double-clicking on the file *supl.exe* where it appears in any file manager. Refer to the *ReleaseNotes.doc* file distributed with **SutraPlot**.
- **SutraPlot** reads information and data from the **SUTRA** main input (*.inp*) file, and nodewise (*.nod*) and elementwise (*.ele*) results files. It is critical that the main input file follows the **SUTRA** 2D3D.1 formatting rules, and that the header in the results file not be modified in any way. Editing these files may result in read errors that cause incorrect plotting of data, or a program crash.
- A screen resolution of 1024x768 or higher is recommended for **SutraPlot**.
- User selected values for plotting parameters may be entered into value boxes using either decimal format (e.g. 0.001) or scientific e-format (e.g. 1 e-3).
- **SutraPlot** has online help that can be accessed from Help buttons located on various dialog boxes, or at any time through the Help command on the main menu. The online help will contain the most current information on **SutraPlot** features, and updates.

### Contour Plots

Contour plots can be generated for any **SUTRA** output variable (pressure, concentration, temperature, saturation). For all contouring options, **SutraPlot** bases contouring on a regular rectangular grid of data points that overlays the finite-element mesh. Values at grid points are evaluated by interpolating between data values at the nodes of the finite-element mesh.

**SutraPlot** uses a numerical algorithm for interpolating data values at grid points that is based on the finite-element methodology used in **SUTRA** (for more information on the gridding methodology, See Appendix A).

### Creating a new contour plot

1. *When starting a new SutraPlot session:*
  - Start **SutraPlot** and select **File | Load SUTRA input file** from the main menu.
  - Locate and open a **SUTRA** input (*.inp*) file.
  - Select **Contours (2D section)** in the **3D Simulation Setup** dialog or **Contour** in the **2D Simulation Setup** dialog.
  - Locate and open a **SUTRA** nodewise (*.nod*) results file.
  - Select a type of plot variable (concentration, temperature, pressure, or saturation).
  - For 3D **SUTRA** problems, select a 2D slice.
  - Click **Apply** in the **Contour Plot** dialog to display the plot.
2. *When starting with an active SUTRA input (.inp) file:*
  - Choose the **Contour | Create new plot** command on the main menu.
  - Locate and open a **SUTRA** nodewise (*.nod*) results file.

- Select a type of plot variable (concentration, temperature, pressure, or saturation).
- For 3D **SUTRA** problems, select a 2D slice.
- Click **Apply** in the **Contour Plot** dialog to display the plot.

**SutraPlot** draws the contours using the GINOSURF<sup>®</sup> graphics library. Default parameters are assumed to create an acceptable contour plot.

### **Default settings**

- Gridded data is generated on a 50 by 50 grid.
- Values at grid points are calculated by a finite-element interpolation method (See Appendix A).
- Eleven contour intervals are plotted, with ten evenly spaced intervals within the full range of data.
- The boundary of the finite-element mesh is drawn around the model area.
- Labeling for all plotted contour intervals is active, but labeling may be restricted by curvature of the lines and the available space between contours.
- The plot is drawn with a bounding frame, axis labels, and tick marks.

#### ***2D SUTRA simulations only:***

- Entire model region is plotted.
- Plots are displayed with the model X-coordinate shown on the horizontal screen axis and the model Y-coordinate on the vertical screen axis.

#### ***3D SUTRA simulations only:***

- The entire 2D section is plotted.
- For X-Y and X-Z sections, plots are displayed with the model X-coordinate shown on the horizontal screen axis. For Y-Z sections, plots are displayed with the Y-coordinate shown on the horizontal screen axis and the Z-coordinate on the vertical screen axis.

## **The Contour Plot Dialog Box**

The **Contour Plot** dialog is the main window for controlling the components of the contour plot. When a new contour plot is created, the **Contour Plot** dialog is displayed before the plot is drawn. The following sections describe the components of the **Contour Plot** dialog box.

### ***Selecting a time step***

To change the time step, click on one of the available time steps in the **Time step** drop-down list box. The output file is automatically read and the plotting data is updated but the plot is not re-drawn. Click **apply** to update and refresh the plot.

### *Viewing the nodewise (.nod) output file header*

Click the **Show Details** button. The header information from the **SUTRA .nod** file is displayed in a new window. Click the **Hide Details** button to hide the window.

### *Selecting a plot variable*

**SutraPlot** reads the output file header to determine which variables that have been printed. To change the plot variable, click on any of the active variables in the **Plot variable** panel. The output file is automatically read and the plotting data is updated but the plot is not re-drawn. Click **Apply** to update and refresh the plot.

### *Selecting a 2D slice (3D SUTRA simulations only)*

Contour plots of 3D **SUTRA** simulations are limited to 2D planes (slices) through the 3D finite-element mesh. There are two types of 2D slices in **SutraPlot**. The **Orthogonal slice**, which is a plane parallel to one of the principal coordinate planes; and the **Oblique slice**, which has one edge of the plane parallel to one of the principal coordinate axes. In each case, the plane is defined by slicing through a rectangular box that bounds the mesh and is aligned with the coordinate axes. Click on **Orthogonal** or **Oblique** in the **Slice selection** panel to open the respective 2D slice selection dialog.

**Orthogonal slice.** —The 2D orthogonal slice can be located at any position in the finite-element mesh, but is restricted to planes parallel to the X-Y, X-Z, or Y-Z planes. To select an orthogonal slice, click on **Orthogonal plane** in the **Slice selection panel**. A **2D slice selection** dialog is displayed. Select a **Coordinate plane** in the drop-down list box. Then enter a slice location as a distance along the remaining coordinate axis in the value box.

**Oblique slice.**— The 2D oblique slice can be located at any position in the finite-element mesh, but is restricted having one edge parallel to the X, Y, or , Z axis. There are three steps in selecting the 2D oblique plane. **First**, select a baseline. The baseline is parallel to one of the principal axes and defines one edge of a bounded plane. **Second**, select a location for the baseline. The baseline can be located at any position within the rectangular box that bounds the finite-element mesh. **Third**, select a second position within the mesh bounds to extend the baseline. A line at the second position and parallel to the baseline defines the opposite edge of the plane. The remaining edges of the plane are defined by the bounding rectangle of the finite-element mesh.

To select an oblique slice, click on **Oblique plane** in the **Slice selection panel**. A **2D slice selection** dialog is displayed. Click on **X, Y, or Z-axis** to select a baseline orientation. In the **Located at** panel, enter a value for the appropriate coordinate, slide the box along the scroll bars or press the arrow keys to select a baseline location. In the **Extending to** panel, enter a value for the appropriate coordinate, slide the box along the scroll bars or press the arrow keys to select a baseline extension location.

### *Displaying the axis labels*

A bounding box with axis labels and tick marks is drawn around all contour plots. The **Show axis labels** check box allows this feature to be turned on and off.

### *Displaying the finite-element mesh boundary*

The finite-element mesh boundary is drawn as a red line around all contour plots. The **Show mesh boundary** check box allows this feature to be turned on and off.

### *Using the Zoom function*

The zoom function allows magnification of the plot by drawing a rectangle around the area of interest. Starting anywhere in the graphics frame, hold down the left mouse button and drag a rectangle around a small area of the plot. Release the mouse button, then click **OK** in the **Zoom to selected area** prompt. Click **Zoom out** in the **Contour Plot dialog** to return to the original view. The rectangle may include areas outside the mesh bounding box, however, the zoom will be clipped at the bounding box limits.

The zoom function may be used to magnify the plot multiple times. To zoom out to the original full view, click the **Zoom out** button.

### *Specifying Contour Levels*

Choose the **Properties | Contour level** command from the **Contour Plot** dialog. The **Contour Level** dialog box is displayed. For reference, the model data limits and scaled data limits (see **Using Data Scaling Factors** below) are displayed in the **Data values** panel.

To select new contour levels, enter values for **Upper level**, **Lower level**, and **Contour interval** in the **Contour values** panel. The **Reset Default Levels** button can be used to recalculate the default contour levels.

### *Using Data Scaling Factors*

By default, the model data is plotted unscaled. There are two ways to scale the model data in **SutraPlot**. Data can be scaled by applying a linear multiplier, or by calculating  $\text{Log}_{10}$  of the data values. To change the data scaling, choose the **Properties | Contour level** command from the **Contour Plot** dialog.

**Linear scaling factors.**—To assign new scaling factors, enter values for **BASE** and/or **MULT** in the **Data Scaling factors** panel. Click the check box next to **Apply linear scaling factors**. Each data value will be divided by **BASE** and multiplied by **MULT**. The re-scaled data limits are displayed under **Scaled data**, and the values are highlighted. Click **Apply** to update and refresh the plot.

**Log<sub>10</sub> scaling factors.**—To apply the Log conversion, click the check box next to **Plot LOG of data values**. The **Contour panel** changes to display the **Upper level** contour value, and the **Number of log intervals below upper level**. By default, the **Upper level** is set to  $\text{Log}_{10}$  of the maximum data value, and the **Number of Log intervals** is set to 5. To select new contour levels, enter new values for the **Upper level** and **Number of Log intervals**. A new **Upper level** can be selected by entering a value in either **Log(value)** units or **Model** units. The re-scaled data limits are displayed under **Scaled data**, and the values are highlighted. Click **Apply** to update and refresh the plot.

### *Specifying Contour Label Properties*

Choose the **Properties | Contour Labels** command from the **Contour Plot** dialog. The **Contour Labels** dialog box is displayed. By default, labels are displayed with two decimal places, and regularly spaced with an approximate label-to-label distance of 25 mm. The actual number of labels and spacing may be restricted by the curvature of the lines and the available space between contours.

The **Show Labels** check box turns the label display on or off.

To change the number of decimal places displayed, click on one of the values (0 to 7) in the **Decimals** drop-down list box.

To increase or decrease the label spacing, slide the box along the **Contour label spacing** scroll bar or press the arrow keys. Each press of an arrow key increases or decreases the spacing by approximately 5mm.

### *Scaling the plot coordinate axes*

Choose the **Properties | Axis Scaling** command from the **Contour Plot** dialog. The **Coordinate Axis Scaling** dialog box is displayed. The minimum, maximum, and coordinate range for each direction in the plot is displayed. The information displayed depends on the type of contour plot.

**2D SUTRA simulation.--** The **X-axis** of the plot is displayed horizontally and the **Y-axis** vertically on the screen. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**” and/or “**Y**” and enter a value in the adjacent value box.

**2D Orthogonal slice, 3D SUTRA simulation.--** **X-Y** and **X-Z** section plots are displayed with the model **X**-coordinate on the horizontal screen axis. For **Y-Z** sections, plots are displayed with the **Y**-coordinate on the horizontal screen axis and the **Z**-coordinate on the vertical screen axis. The two appropriate **Scaling factor** check boxes are activated depending on the slice orientation. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**”, “**Y**”, and/or “**Z**” as appropriate for the section, and enter a value in the adjacent value box.

**2D Oblique slice, 3D SUTRA simulation.--** The coordinates for all oblique planes are translated and recomputed to that the origin (0,0) is at the lower left corner of the plot. Plots are displayed with the slice baseline on the vertical screen axis. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**Horizontal**” and/or “**Vertical**” and enter a value in the adjacent value box.

### *Changing the gridding density*

Choose the **Properties | Gridding Density** command from the **Contour Plot** dialog. The **Gridding Density** dialog box is displayed. By default, the grid is set to 50x50. To change the gridding, enter new values in the **Horizontal screen axis** and/or the **Vertical screen axis value boxes**. Click **Apply** to update and refresh the plot with the new gridding.

## *Using Setup files*

**Saving setup files.**-- After a plot has been drawn in the graphics window, all the information necessary to reproduce the plot can be saved to a **SutraPlot** setup (*.spl*) file. When the plot is saved, all the scaling, contour parameters, and options are preserved in the file.

To save a setup file, choose the **File | Save Plot Setup** command from the **Contour Plot** dialog or **File | Save Plot Setup** in the graphics window menu. The **SutraPlot setup** file dialog is displayed. An optional informational header line may be entered here and added to the setup file. Click **Continue**. The **Save Setup File** dialog is displayed. Type a file name for the setup file in the **File name** edit box. Click **Save** and the file is saved to the current directory, and the *.spl* extension is automatically added.

**Opening setup files.**--To open a setup file, choose the **File | Open Plot Setup** command from the **Contour Plot** dialog or **File | Open Plot Setup** in the graphics window menu. The **Open Setup File** dialog is displayed. Locate and open the desired **SutraPlot** setup (*.spl*) file. The **Setup File Info** dialog is opened, and the setup file header information is displayed. Click **OK** to load the setup file and display the plot. **Note:** The ten most recent setup files may be opened directly by selecting the filename from the list on the main menu under **File | Recent Setup Files**.

## **Velocity vector plots**

Velocity vector plots can be generated from both 2D and 3D **SUTRA** simulations.

For 3D simulations, the user must specify a region of the finite-element mesh from which to extract a subset of vectors (see **Selecting a vector region** below). The subset of vectors is projected and displayed in a 2D plane aligned with two of the principal coordinate axes.

### **Creating a new vector plot**

1. *When starting a new SutraPlot session:*
  - Start **SutraPlot** and select **File | Load SUTRA Input File** from the main menu.
  - Locate and open a **SUTRA** input (*.inp*) file.
  - Select **Velocity vectors (2D section projected)** in the **3D Simulation Setup** dialog or **Velocity Vectors** in the **2D Simulation Setup** dialog.
  - Locate and open a **SUTRA** elementwise (*.ele*) results file.
  - For 3D **SUTRA** problems, select a vector region and a projection plane.
  - Click **Apply** in the **Vector plot** dialog to display the plot.
2. *When starting with an active SUTRA input (.inp) file:*
  - Choose the **Vector | Create new plot** command on the main menu.
  - Locate and open a **SUTRA** elementwise (*.ele*) results file.
  - For 3D **SUTRA** problems, select a vector region and a projection plane.
  - Click **Apply** in the **Vector plot** dialog to display the plot.

**SutraPlot** draws the vectors using the GINOSURF<sup>®</sup> graphics library. Default parameters are assumed for most settings to create an acceptable contour plot.

### **Default settings**

- The magnitudes of the vectors are scaled as a linear fraction of the largest vector in the simulation.
- The vector arrowheads are scaled to the vector magnitude.
- If the plotted model region has an axis with an exaggerated scale, the vector directions are adjusted to the exaggerated scale (See Appendix B).
- The plot is drawn with a bounding frame, axis labels, and tick marks.

#### ***2D SUTRA simulations only:***

- All vectors in the model area are plotted.
- Plots are displayed with the model X-coordinate shown on the horizontal screen axis and the model Y-coordinate on the vertical screen axis.
- The boundary of the finite-element mesh is drawn around the model area.

#### ***3D SUTRA simulations only:***

- Only vectors in the selected model region are plotted.
- For X-Y and X-Z sections, plots are displayed with the model X-coordinate shown on the horizontal screen axis. For Y-Z sections, plots are displayed with the Y-coordinate shown on the horizontal screen axis and the Z-coordinate on the vertical screen axis.
- A bounding rectangle is drawn around the maximum extent of the projected area of the mesh containing the selected set of vectors.

### **The Vector Plot Dialog Box**

The **Vector Plot** dialog is the main window for controlling the components of the vector plot. When a new vector plot is created, the **Vector Plot** dialog is displayed before the plot is drawn. The following sections describe the components of the **Vector Plot** dialog box.

#### ***Selecting a time step***

To change the time step, click on one of the available time steps in the **Time step** drop-down list box. The output file is automatically read and the plotting data is updated but the plot is not re-drawn. Click **apply** to update and refresh the plot.

#### ***Viewing the elementwise (.ele) output file header***

Click the **Show Details** button. The header information, including the time step details from the **SUTRA .ele** file, is displayed in a new window. Click the **Hide Details** button to hide the window.

### ***Selecting a vector region (3D SUTRA simulations only)***

For 3D simulations, the user must specify a region of the finite-element mesh from which to extract a subset of vectors. Although it is possible to select all the vectors in a 3D simulation, this option is not recommended because the resultant plot of projected vectors will likely be too cluttered to interpret.

There are two ways to select a vector region in **SutraPlot**; designate a sheet of elements, or define a 3D rectangular box.

#### **Selecting a sheet of elements**

A ‘sheet of elements’ is a region of vectors that is contained within a contiguous sheet of elements, one element thick, and aligned with the **SUTRA** node numbering directions as defined by the structure of **SUTRA**’s regularly-connected meshes (for additional information, see Appendix C).

To select a sheet of elements, click **Sheet** in the **Vector region** panel. Select a set of 2 numbering directions from the **Orientation** drop-down list box in the **Sheet Selection** panel. This defines all the sheets of elements aligned with these two numbering directions. These sheets of elements are stacked along the remaining direction. The sheet number is obtained by counting elements along the remaining numbering direction. Enter the number of the sheet in the **Sheet number** value box.

#### **Selecting a bounding box**

A ‘bounding box’ is a region of vectors can be defined by specifying a limiting 3D rectangular box within the model domain. The box is aligned with the principal coordinate axes.

To select a bounding box, click **Bounding box** in the **Vector region** panel. A **2D slice selection** dialog is displayed. The bounding box can be constrained in any or all of the principal coordinate directions. The default settings define a bounding box that encompasses the entire model domain. To constrain the box, select new minimum and/or maximum coordinate values. New values may be selected by entering a number in the appropriate value box, or by using the sliding scroll bars.

### ***Selecting a projection plane (3D SUTRA simulations only)***

The set of vectors selected for plotting are projected onto a plane aligned with two of the principal coordinate axes. **Note:** Users should be very cautious when evaluating the projected velocities. **SUTRA** computes velocity vectors in true 3D. Depending on the general direction of 3D vectors relative to the projected 2D vectors, an extreme distortion of flow direction and flow patterns may result.

To change the projection plane, select a new **Coordinate pair** in the drop-down list box.

### ***Using the Zoom function***

The zoom function allows magnification of the plot by drawing a rectangle around the area of interest. Starting anywhere in the graphics frame, hold down the left mouse button and drag a rectangle around a small area of the plot. Release the mouse button, then click **OK** in the **Zoom to selected area** prompt. Click **Zoom out** in the **Vector Plot dialog** to return to the



original view. The rectangle may include areas outside the mesh bounding box, however, the zoom will be clipped at the bounding box limits.

The zoom function may be used to magnify the plot multiple times. To zoom out to the original full view, click the **Zoom out** button.

### *Using Vector Thinning*

Choose the **Properties | Vector thinning** command from the **Vector Plot** dialog. The **Vector thinning** dialog box is displayed. There are three ways to reduce the number of vectors displayed:

1-Plot every  $N^{\text{th}}$  vector -- A new set of vectors can be defined by specifying that every  $N^{\text{th}}$  vector be plotted. Click the **Plot every vector** check box. Enter an integer in the value box. For example, a value of 3 will plot the 1<sup>st</sup>, 4<sup>th</sup>, 7<sup>th</sup>, 10<sup>th</sup>, etc. vector starting with the first element in the elementwise data list. Note: the vectors are plotted in the same order as the elementwise data listing.

2-Skip small vectors -- A set of vectors can be defined by specifying that vectors with magnitudes **less than** a user-selected percentage of the largest vector **not** be plotted. Click the **Drop vectors < % of max magnitude** check box. Enter the lower end of the percent cutoff value in the value box. Use values between 0 and 100 percent.

3-Skip large vectors – A set of vectors can be defined by specifying that vectors with magnitudes **greater than** a user-selected percentage of the largest vector **not** be plotted. Click the **Drop vectors > % of max magnitude** check box. Enter the upper end of the percent cutoff value in the value box. Use values between 0 and 100 percent.

If the vectors thinning methods are combined, the first (Plot every  $N^{\text{th}}$  vector) method takes precedence. If an upper or lower cutoff percentage is used, the set of vectors specified by the first method is not changed, although the number of vectors displayed may change.

### *Setting Vector Properties*

The vector arrow scaling and display properties are controlled in the **Vector Properties** dialog box. To open the **Vector Properties** dialog choose the **Properties | Vector Properties** command from the **Vector Plot** dialog.

#### **Setting a reference vector**

By default, the vectors are scaled relative to the largest vector in the finite-element mesh, regardless of which vectors are displayed as a result of the user selecting a subset of vectors. (see **Using Vector thinning** and **Selecting a Vector region**). To scale the vectors to the largest vector in a selected subset, click on **Local (region) max** in the **Reference vector** panel.

#### **Selecting a vector scaling factor**

There are two ways to scale vectors in **SutraPlot**. Vectors can be scaled between the minimum and maximum data values linearly or logarithmically (See Appendix B). By default, vectors are scaled as linear fractions of the maximum velocity magnitude. To scale the vectors as a logarithmic function of the largest vector, click on **Logarithmic function** in the **Vector scaling factors** panel.

### **Setting the vector direction scaling**

By default, the vector directions are adjusted to exaggerated scales. To display the vectors without direction adjustment on a plot with an exaggerated scale, click the **Vector direction scaling** check box. As with any data that is displayed in an exaggerated section, caution should be used when evaluating velocity vectors that have been plotted in exaggerated space (see Appendix B).

### **Adjusting the vector length**

By default, the plotted length of the maximum velocity magnitude is one centimeter. Note: although the printed output will correctly scale the vectors, the screen display may vary the vector lengths depending on window size, and screen size and resolution. The vector lengths can be resized in two ways. 1- click the arrow to the left of **Vector magnitude** in the **Vector size adjustment** panel. Then click on a preset percentage in the drop-down list box to increase or decrease the size of the vectors. The default size is 100%; or 2-click the check box to the right of **Vector magnitude**, then enter a new percentage in the adjacent value box.

### **Adjusting the arrowhead size**

By default, the largest arrowhead is one-fifth the length of the maximum velocity magnitude. The vector arrowhead can be resized in two ways. 1-Click the arrow to the left of **Arrowhead size** in the **Vector size adjustment** panel. Then click on a preset percentage in the drop-down list box to increase or decrease the size of the arrowhead. The default size is 100%; or 2-Click the check box to the right of **Arrowhead size**, then enter a new percentage in the adjacent value box.

### **Selecting vector arrowhead scaling**

By default, the arrowheads are scaled to the vector magnitudes. To plot arrowheads with a fixed size, click on **Fixed size** in the **Vector Arrowhead scaling** panel.

### **Selecting a reference vector**

A set of reference vectors is displayed with the vector plot. By default, the maximum velocity magnitude is shown along with three other magnitudes. When the vectors are linearly scaled, the additional magnitudes are one-half, one-fourth, and one-eighth of the maximum. When the vectors are logarithmically scaled, the additional vectors are 1, 2, and 3 orders of magnitude smaller than the maximum. To select a new value for the displayed maximum, click on **Reference vectors**, and enter a new value in the **Maximum vector magnitude** value box. Click **Apply** to display the new set of reference vectors.

### *Scaling the plot coordinate axes*

Choose the **Properties | Axis Scaling** command from the **Vector Plot** dialog. The **Coordinate Axis Scaling** dialog box is displayed. The minimum, maximum, and coordinate range for each direction in the plot is displayed. The information displayed depends on the type of vector plot.

**2D SUTRA simulation--** The **X-axis** of the plot is displayed horizontally and the **Y-axis** vertically. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**” and/or “**Y**” and enter a value in the adjacent value box.

**3D SUTRA simulation--** **X-Y** and **X-Z** oriented vector regions are displayed with the model **X**-coordinate on the horizontal screen axis. For **Y-Z** vector regions, plots are displayed with the **Y**-coordinate on the horizontal screen axis and the **Z**-coordinate on the vertical screen axis. The two appropriate **Scaling factor** check boxes are activated depending on the region orientation. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**”, “**Y**”, and/or “**Z**” as appropriate for the region, and enter a value in the adjacent value box.

### *Using Setup files*

**Saving setup files.--**After a plot has been drawn in the graphics window, all the information necessary to reproduce the plot can be saved to a **SutraPlot** setup (*.spl*) file. When the plot is saved, all the scaling, vector parameters, and options are preserved in the file.

To save a setup file, choose the **File | Save Plot Setup** command from the **Vector Plot** dialog or **File | Save Plot Setup** in the graphics window menu. The **SutraPlot setup** file dialog is displayed. An optional informational header line may be entered here and added to the setup file. Click **Continue**. The **Save Setup File** dialog is displayed. Type a file name for the setup file in the **File name** edit box. Click **Save** and the file is saved to the current directory, and the *.spl* extension is automatically added.

**Opening setup files.--**To open a setup file, choose the **File | Open Plot Setup** command from the **Vector Plot** dialog or **File | Open Plot Setup** in the graphics window menu. The **Open Setup File** dialog is displayed. Locate and open the desired **SutraPlot** setup (*.spl*) file. The **Setup File Info** dialog is opened, and the setup file header information is displayed. Click **OK** to load the setup file and display the plot. **Note:** The ten most recent setup files may be opened directly by selecting the filename from the list on the main menu under **File | Recent Setup Files**.

## **Finite-element mesh plot**

Plots of the full finite-element mesh are available for 2D **SUTRA** simulations.

The 3D finite-mesh mesh plot has limited functionality and is supplied only as a tool for displaying general spatial relationships of the finite-element mesh. The 3D mesh display is able to fully rotate in three dimensions but is limited to drawing the elements on the six outside faces of the finite-element mesh.

### **Creating a new mesh plot**

1. *When starting a new SutraPlot session:*
  - Start **SutraPlot** and select **File | Load SUTRA input file** from the main menu.
  - Locate and open a **SUTRA** input (*.inp*) file.
  - Select **Finite-element mesh plot** in the **2D/ 3D Simulation Setup** dialog.
2. *When starting with an active SUTRA input (.inp) file:*

- Choose the **Mesh | Create new plot** command on the main menu.

## Default settings

### *2D SUTRA simulations only:*

- Plots are displayed with the model X-coordinate shown on the horizontal screen axis and the model Y-coordinate on the vertical screen axis.
- The mesh plot is drawn with axis labels and tick marks.

### *3D SUTRA simulations only:*

- The mesh plot displays the elements on all six outside faces of the mesh.
- The mesh is displayed with the Z-axis aligned with the vertical screen axis, and a viewed from a point equidistant along the three coordinate axes, toward the origin.

## The 2D Mesh Plot Dialog Box

The 2D **Mesh Plot** dialog is the main window for controlling the components of the 2D mesh plot. When a new mesh plot is created, the **Mesh Plot** dialog is displayed before the plot is drawn. The following sections describe the components of the **Mesh Plot** dialog box.

### *Displaying the axis labels*

The default mesh is drawn with axis labels and tick. The **Show axis labels** check box allows this feature to be turned off.

### *Using the Zoom function*

The zoom function allows magnification of the plot by drawing a rectangle around the area of interest. Starting anywhere in the graphics frame, hold down the left mouse button and drag a rectangle around a small area of the plot. Release the mouse button, then click **OK** in the **Zoom to selected area** prompt. Click **Zoom out** in the **Mesh Plot dialog** to return to the original view. If the rectangle includes area outside the mesh bounding box, the zoom will be clipped at the bounding box limits.

The zoom function may be used to magnify the plot multiple times. To zoom out to the original full view, click the **Zoom out** button.

### *Scaling the plot coordinate axes*

Choose the **Properties | Axis Scaling** command from the **Contour Plot** dialog. The **Coordinate Axis Scaling** dialog box is displayed. The minimum, maximum, and coordinate range for each direction in the plot is displayed. The **X-axis** of the plot is displayed horizontally and the **Y-axis** vertically on the screen. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**” and/or “**Y**” and enter a value in the adjacent value box.

## The 3D Mesh Plot Dialog Box

The 3D **Mesh Plot** dialog is the main window for controlling the components of the 3D mesh plot. When a new mesh plot is created, the **Mesh Plot** dialog is displayed before the plot is drawn. The following sections describe the components of the **Mesh Plot** dialog box.

### *Selecting a viewing direction*

The mesh may be rotated and aligned with the coordinate axes. To change the orientation, click the appropriate +/-, **X**, **Y**, or **Z** button in the **Set viewing direction** panel. The mesh will rotate so that it is viewed *along* the respective axis *from* the positive or negative direction.

### *Selecting a drawing option*

By default, the elements on all six outside faces of the mesh are drawn. Click **Outside element frame** to display just the edges of the outside faces of the mesh.

### *Selecting a projection type*

The 3D mesh as displayed is projected from three dimensions to two. There are two projection types available in **SutraPlot**, perspective and parallel. The perspective projection uses the vanishing point projection method and is the best impression of the mesh as it would actually appear from a normal viewing distance. In the parallel projection, the rays of projection are parallel, and the mesh appears as if viewed from a great distance. By default, the mesh is drawn in perspective. Click the **Parallel** button to display the mesh in a parallel projection.

### *Selecting face visibility*

To selectively display the mesh faces, click one or more of the check boxes in the **Mesh face visibility** panel. The boxes, labeled **ND 1/2/3 min/max** refer to the **SUTRA** mesh numbering directions (see Appendix C). For example, if the boxes next to **ND 1 MAX** and **ND 3 MIN** are unchecked, the mesh is redrawn without the face (and the elements) at the end of the first numbering direction, and without the face at the beginning of the third numbering direction.

### *Rotating the mesh*

To rotate the mesh clockwise (looking in the positive direction) around the X, Y, and Z-axes, press X, Y, and Z keys respectively. To rotate counterclockwise, press the alt-X, alt-Y, and alt-Z keys. Click **Show rotation guide** to display a visual rotation guide.

### *Scaling the plot coordinate axes*

Choose the **Properties | Axis Scaling** command from the **Mesh Plot** dialog. The **Coordinate Axis Scaling** dialog box is displayed. The minimum, maximum, and coordinate range for each direction in the plot is displayed. To change the respective axis scaling, click the check box beneath **Scaling factor** and across from “**X**”, “**Y**”, or “**Z**” and enter a value in the adjacent value box.

### *Using the Zoom function*

The 3D mesh zoom function is only available when using the perspective view. To zoom in, press the **I** key, and to zoom out press the **O** key. Using **Alt-I** and **Alt-O** increases the amount of zoom with each key press.

## SutraPlot Export files

### Gridded data files

A gridded data file can be exported for any **SUTRA** output variable (pressure, concentration, temperature, saturation). The gridded data can then be imported into other 2D or 3D visualization software.

**SutraPlot** interpolates data values on a regular rectangular grid of data points that overlays the finite-element mesh using the numerical algorithm described in Appendix A. For non-rectangular finite-element meshes, some of the grid points will be outside the mesh. These grid points are included in the data file and flagged with a value of 999.999. There is no maximum number of grid points. However, considerable array storage and computational effort may be required for generating large grids with large **SUTRA** finite-element meshes.

**2D SUTRA simulations.--** A dataset of model coordinates and gridded data values are computed and written to a formatted ASCII file. The 2D gridded data output filenames are automatically given the default extension, **.gr2**. Each line has values for one grid point, consisting of x and y coordinates, and a gridded data value. Values are output in three space delimited fields of 12 columns each and written in Fortran ‘G’ format. Output file size is 38 bytes per grid point.

**3D SUTRA simulations.--** A dataset of model coordinates and gridded data values are computed and written to a formatted ASCII file. The 3D gridded data output filenames are automatically given the default extension, **.gr3**. Each line has values for one grid point, consisting of x, y, and z coordinates, and a gridded data value. Values are output in four space delimited fields of 12 columns each and written in Fortran ‘G’ format. Output file size is 50 bytes per grid point.

Create gridded data files using the **File | Export | Gridded Data** command. The **Gridded data output file** dialog box is displayed.

### The Gridded Output file Dialog Box

#### *Selecting a time step*

To change the time step, click on one of the available time steps in the **Time step** drop-down list box.

#### *Viewing the nodewise (.nod) output file header*

Click the **Show Details** button. The header information, including the time step details from the **SUTRA .nod** file, is displayed in a new window. Click the **Hide Details** button to hide the window.

### *Selecting a plot variable*

**SutraPlot** reads the output file header and only activates variables that have been printed. To change the plot variable, click on any of the active variables in the **Plot variable** panel.

### *Specifying the grid density*

Grid density refers to the number of row and columns in the regular rectangular grid that is overlain on the finite-element mesh. Higher grid densities will produce a smoother plot; however, an increase in the number of grid points proportionally increases the size of the grid file, and geometrically increases the time to compute the grid points.

The number of grid points can be specified in the X and Y directions for 2D, and for the X, Y, and Z-directions for 3D. To change the grid density, enter values in the respective boxes in the **Grid density** panel.

### *Selecting a grid dummy value*

By default, grid points outside the mesh are flagged with a value of 999.999. To change this value, enter a new value in the **Grid point dummy value** box.

## **AutoCAD (.dxf) export files**

**SutraPlot** graphic output can be exported to an AutoCAD Drawing Exchange Format (**.dxf**) file. AutoCAD **.dxf** files can be imported into many graphic display and visualization programs.

To export a **.dxf** file, choose the **File | Export | DXF file** command from the graphics window menu. In the **File name** edit box, enter the name of the export file to be created. The **.dxf** file extension is automatically added to the filename. Click **Save** and the file is created.

## **REFERENCES**

- Burnett, R. D., and Frind, E. O., 1987, Simulation of contaminant transport in three dimensions, 2. Dimensionality effects: *Water Resources Research*, v. 23, no. 4, p. 695-705.
- Provost, A. M., 2002, SutraPrep, a pre-processor for SUTRA, a model for ground-water flow with solute or energy transport: U. S. Geological Survey Open File Report 02-376, U. S. Geological Survey, Reston, Virginia.
- Souza, W. R., 1987, Documentation of a graphical display program for the saturated-unsaturated transport (SUTRA) finite-element simulation model: U.S. Geological Survey Water-Resources Investigations Report 87-4245, 122 p.
- Voss, C. I., 1984, SUTRA, A finite-element simulation model for saturated-unsaturated, fluid-density dependent ground-water flow with energy transport or chemically-reactive single-species solute transport: U.S. Geological Survey Water-Resources Investigations Report 84-4369, 409 p.
- Voss, C. I. and Provost, A. M., 2002, SUTRA, a model for saturated-unsaturated, variable-density ground-water flow with solute or energy transport: U.S. Geological Survey Water-Resources Investigations Report 02-4231, 250 p.



## ***APPENDIX A – Finite-element interpolation methodology***

For all contouring options, **SutraPlot** uses a regular rectangular grid of data points that overlays the finite-element mesh. The value at each grid point is evaluated by interpolating between data values at the nodes of the finite-element mesh. The methodology and numerical algorithms for gridding are the same for 2D and 3D **SUTRA** finite elements, both of which are described below. The interpolation routine for 2D finite elements is the same as previous versions of **SutraPlot**, and was originally described by Souza (1987). For the current **SutraPlot**, Version 2D3D.1, the 2D interpolation methodology is extended to 3D finite elements.

The interpolation used in **SutraPlot** calculates the data value at a grid point using basis functions and a numerical algorithm equivalent to that used in **SUTRA** methodology. The basis functions define the value of a variable at all points within an element when the value of the variable is known at the nodes of the element. By utilizing basis functions, **SutraPlot** calculates exact values of **SUTRA** output variables at any location within an element. This results in accurate interpolated values regardless of the spacing of data in the finite-element mesh (Souza, 1987). For a more complete description of the numerical methods, refer to the **SUTRA** documentation (Voss, 1984; Voss and Provost, 2002; sections 3.2, ‘Nodewise discretization’, and 4.1, ‘Basis and Weighting Functions’).

### ***Interpolation for 2D SUTRA finite elements***

In 2D **SUTRA** finite-element simulations, a 2D piecewise continuous function is used to represent the spatial distribution of variables, rather than single values at nodes, using bi-linear basis functions. The equation that gives the value of a **SUTRA** output variable,  $U$  (i.e. concentration, temperature, pressure, saturation) at a point  $(x,y)$  may be expressed for the two-dimensional mesh as:

$$U(x, y) = \sum_{i=1}^4 U_i \phi_i(x, y), \quad (1)$$

where the subscript  $i$  refers to each of the four nodes in an element. There are a total of four values,  $U_i$ , at the coordinates  $(x_i, y_i)$  of element corner nodes  $i$ . The function  $\phi_i(x,y)$  is the basis function defined at each node  $i$ . In **SUTRA**, basis functions are described in local geometry. In a local coordinate system  $(\xi, \eta)$ , each element takes the shape of a two by two square, with the origin at the center of the element. Using local coordinates, the value at any point  $(x,y)$  in an element of **SUTRA** output variable,  $U$ , is:

$$U(x, y) = \sum_{i=1}^4 U_i \phi_i(\xi, \eta), \quad (2)$$

where  $\xi$  and  $\eta$  are the local coordinates defined by Voss (1984), and Voss and Provost (2002), and  $\phi_i$  is the local basis function (see equation 4a-d) defined at  $(\xi, \eta)$  for the  $i^{\text{th}}$  node in an element. To calculate the value of  $U(x,y)$ , the values of  $U_i$  at the corner nodes must be known; and the four basis functions corresponding to the corner nodes for the element must be

determined at the point  $(\xi, \eta)$ . The global coordinate system  $(x, y)$  can be related to the local coordinate system by:

$$x = \sum_{i=1}^4 \phi_i(\xi, \eta) x_i, \text{ and} \quad (3a)$$

$$y = \sum_{i=1}^4 \phi_i(\xi, \eta) y_i, \quad (3b)$$

where  $x$  and  $y$  are the global coordinates of the grid point located in an element;  $x_i$  and  $y_i$  are the coordinates of the corner nodes of the element, and  $\phi_i$  is the basis function defined at  $\xi$  and  $\eta$  for node  $i$ . The basis functions  $\phi_i$  ( $i=1$  to  $4$ ) are defined as:

$$\phi_1 = 0.25 (1-\xi)(1-\eta) \quad (4a)$$

$$\phi_2 = 0.25 (1+\xi)(1-\eta) \quad (4b)$$

$$\phi_3 = 0.25 (1+\xi)(1+\eta) \quad (4c)$$

$$\phi_4 = 0.25 (1-\xi)(1+\eta) \quad (4d)$$

Substituting equation 4(a-d) for  $\phi_i$  in equation 3(a-b) and expanding the result gives:

$$x = 0.25 [(1-\xi)(1-\eta)x_1 + (1+\xi)(1-\eta)x_2 + (1+\xi)(1+\eta)x_3 + (1-\xi)(1+\eta)x_4] \quad (5a)$$

$$y = 0.25 [(1-\xi)(1-\eta)y_1 + (1+\xi)(1-\eta)y_2 + (1+\xi)(1+\eta)y_3 + (1-\xi)(1+\eta)y_4] \quad (5a)$$

For each grid point,  $x$  and  $y$  are given, and equations 5a and 5b are solved simultaneously by Newton's iterative method for  $\xi$  and  $\eta$ . Equations 4(a-d) are then used to evaluate  $\phi_i$  ( $i=1$  to  $4$ ) at the grid point. Finally, the value at the grid point of the **SUTRA** output variable,  $U(x, y)$  is calculated by equation (2).

### *Interpolation for 3D SUTRA finite elements*

In 3D **SUTRA** finite-element simulations, a 3D piecewise continuous function is used to represent the spatial distribution of variables, rather than single values at nodes, using tri-linear basis functions. The equation which gives the value of a **SUTRA** output variable,  $U$  (i.e. concentration, temperature, pressure, saturation) at a point  $(x, y, z)$ , may expressed for the three-dimensional mesh as:

$$U(x, y, z) = \sum_{i=1}^8 U_i \phi_i(x, y, z), \quad (7)$$

where the subscript  $i$  refers to each of the eight nodes in an element. There are a total of eight values,  $U_i$ , at the coordinates  $(x_i, y_i, z_i)$  of element corner nodes  $i$ . The function  $\phi_i(x, y, z)$  is the basis function defined at each node  $i$ . In **SUTRA**, basis functions are described in local

geometry. In a local coordinate system  $(\xi, \eta, \zeta)$ , each element takes the shape of a two by two by two cube, with the origin at the center of the element. Using basis functions, the value at any point  $(x, y, z)$  in an element of **SUTRA** output variable,  $U$ , is:

$$U(x, y, z) = \sum_{i=1}^8 U_i \phi_i(\xi, \eta, \zeta), \quad (8)$$

where  $\xi$ ,  $\eta$ , and  $\zeta$  are the local coordinates and  $\phi_i$  is the local basis function (see equation 10a-h) defined at  $(\xi, \eta, \zeta)$  for the  $i^{\text{th}}$  node in an element. To calculate the value of  $U(x, y, z)$ , the values of  $U_i$  at the corner nodes must be known; and the eight basis functions corresponding to the corner nodes for the element must be determined at the point  $(\xi, \eta, \zeta)$ . The global coordinate system  $(x, y, z)$  can be related to the local coordinate system by:

$$x = \sum_{i=1}^8 \phi_i(\xi, \eta, \zeta) x_i, \quad (9a)$$

$$y = \sum_{i=1}^8 \phi_i(\xi, \eta, \zeta) y_i, \text{ and} \quad (9b)$$

$$z = \sum_{i=1}^8 \phi_i(\xi, \eta, \zeta) z_i, \quad (9c)$$

where  $x$ ,  $y$  and  $z$  are the global coordinates of the grid point located in an element;  $x_i$ ,  $y_i$ , and  $z_i$  are the coordinates of the corner nodes of the element, and  $\phi_i$  is the basis function defined at  $\xi$ ,  $\eta$ , and  $\zeta$  for node  $i$ .

The basis functions  $\phi_i$  ( $i=1$  to 8) are defined as:

$$\phi_1 = 0.125 (1-\xi)(1-\eta)(1-\zeta) \quad (10a)$$

$$\phi_2 = 0.125 (1+\xi)(1-\eta)(1-\zeta) \quad (10b)$$

$$\phi_3 = 0.125 (1+\xi)(1+\eta)(1-\zeta) \quad (10c)$$

$$\phi_4 = 0.125 (1-\xi)(1+\eta)(1-\zeta) \quad (10d)$$

$$\phi_5 = 0.125 (1-\xi)(1-\eta)(1+\zeta) \quad (10e)$$

$$\phi_6 = 0.125 (1+\xi)(1-\eta)(1+\zeta) \quad (10f)$$

$$\phi_7 = 0.125 (1+\xi)(1+\eta)(1+\zeta) \quad (10g)$$

$$\phi_8 = 0.125 (1-\xi)(1+\eta)(1+\zeta) \quad (10h)$$

Substituting equation 10(a-h) for  $\phi_i$  in equation 9(a-c) and expanding the result gives:

$$\begin{aligned}
 x = 0.125[ & (1-\xi)(1-\eta)(1-\zeta)x_1 + (1+\xi)(1-\eta)(1-\zeta)x_2 + \\
 & (1+\xi)(1+\eta)(1-\zeta)x_3 + (1-\xi)(1+\eta)(1-\zeta)x_4 + \\
 & (1-\xi)(1-\eta)(1+\zeta)x_5 + (1+\xi)(1-\eta)(1+\zeta)x_6 + \\
 & (1+\xi)(1+\eta)(1+\zeta)x_7 + (1-\xi)(1+\eta)(1+\zeta)x_8 ] \quad (11a)
 \end{aligned}$$

$$\begin{aligned}
 y = 0.125[ & (1-\xi)(1-\eta)(1-\zeta)y_1 + (1+\xi)(1-\eta)(1-\zeta)y_2 + \\
 & (1+\xi)(1+\eta)(1-\zeta)y_3 + (1-\xi)(1+\eta)(1-\zeta)y_4 + \\
 & (1-\xi)(1-\eta)(1+\zeta)y_5 + (1+\xi)(1-\eta)(1+\zeta)y_6 + \\
 & (1+\xi)(1+\eta)(1+\zeta)y_7 + (1-\xi)(1+\eta)(1+\zeta)y_8 ] \quad (11b)
 \end{aligned}$$

$$\begin{aligned}
 z = 0.125[ & (1-\xi)(1-\eta)(1-\zeta)z_1 + (1+\xi)(1-\eta)(1-\zeta)z_2 + \\
 & (1+\xi)(1+\eta)(1-\zeta)z_3 + (1-\xi)(1+\eta)(1-\zeta)z_4 + \\
 & (1-\xi)(1-\eta)(1+\zeta)z_5 + (1+\xi)(1-\eta)(1+\zeta)z_6 + \\
 & (1+\xi)(1+\eta)(1+\zeta)z_7 + (1-\xi)(1+\eta)(1+\zeta)z_8 ] \quad (11c)
 \end{aligned}$$

For each grid point, x, y, and z are given, and equations 11(a-c) are solved simultaneously by Newton's iterative method for  $\xi$ ,  $\eta$ , and  $\zeta$ . Equations 10(a-h) are used to evaluate  $\phi_i$  ( $i=1$  to 8) at the grid point. Finally, the value at the grid point of the **SUTRA** output variable,  $U(x,y,z)$  is calculated by equation (8).

### ***Notes on gridding algorithm***

To evaluate the value of a **SUTRA** output variable at a point (x,y) within a finite element mesh, the grid point must be associated with and located in an element. Previous versions of **SutraPlot** used various algorithms for a point in polygon search. The current (Version 2D3D.1) version uses the results of the global to local coordinate transformation described above.

To locate the specific element that contains a given grid point, **SutraPlot** does an element-by-element search. As each element is checked, the global to local coordinate transformation is computed. If the grid point is within an element, all of the local coordinates of the grid point will fall within the range -1 to 1 (as defined for the local coordinate system). If any one of the local coordinates of the grid point is out of range, then it is not in the element. Each element is checked in the same way until the criteria are met. To increase computational efficiency, the number of elements that are checked is reduced to a subset of elements using a search algorithm that compares ranges of element and grid point coordinates.

## APPENDIX B - Vector magnitude scaling and direction adjustment

### Vector magnitude scaling

Velocity vector magnitudes produced by **SUTRA** simulations are in units of meters per second (m/s). Typical values are in the range  $10^{-9}$  to  $10^{-6}$  m/s. To plot the vectors, the simulation units must be converted and scaled to plotting units. There are two ways to scale vectors in **SutraPlot**. Vectors can be scaled between the minimum and maximum data values linearly or logarithmically. By default, vectors are scaled as linear fractions of the maximum velocity magnitude.

As an option, vectors can be scaled logarithmically. Scaling vectors based on a logarithmic function to allows better visual evaluation when the range of velocities spans several orders of magnitude. Logarithmic scaling is obtained using the following relationship:

$$\text{vmagl} = \frac{1}{1 - [\log_{10}(\text{vmag} / v_{\max})]}$$

where vmagl is the scaled vector magnitude, vmag is a model vector magnitude, and  $v_{\max}$  is the maximum modeled vector magnitude. In this equation,  $v_{\max}$  is normalized to one; and for each order of magnitude  $n$ , that vmag is smaller than the maximum, the plotted length of vmag is reduced by a factor of  $1/(n+1)$ . This results in plotted vectors that decrease by factors of  $1/2$ ,  $1/3$ ,  $1/4$ , etc., for each decrease in order of magnitude. This relationship is shown graphically in [Figure 5](#).

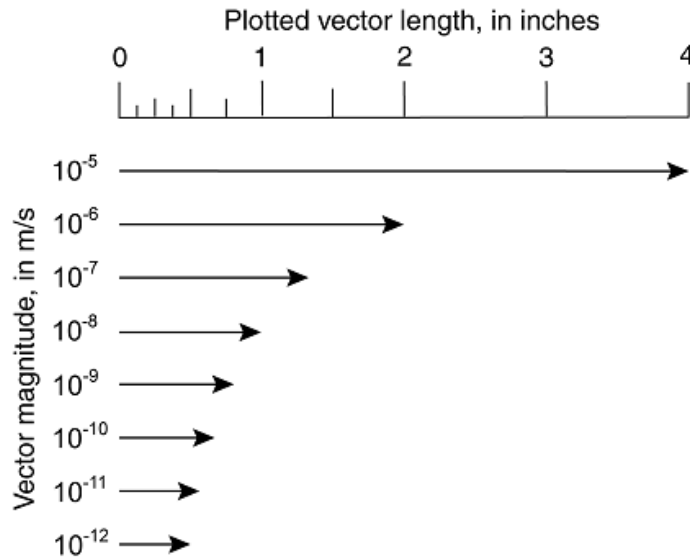


Figure 5. Graphic representation of velocity vectors that have been scaled using a logarithmic function. For each  $n$  orders of magnitude reduction in the largest magnitude, the plotted length is reduced by a factor of  $1/(n+1)$ .

### Vector direction adjustment

In plots in which one of the coordinate scales has been exaggerated, such as vertical cross-sections, velocity vectors angles are automatically re-scaled in **SutraPlot** to adjust for the exaggerated domain. Figure 6 demonstrates the need for this type of vector direction adjustment. Figure 6a shows a schematic finite-element mesh at true scale with plotted velocity vectors that indicate a direction of flow diagonally across the mesh that will intersect the upper right corner of each element. In Figure 6b, the bottom row of elements has been re-plotted with a 5X vertical exaggeration.

Two sets of vectors are shown in Figure 6b. The vectors drawn as solid arrows retain the same direction as the vectors in Figure 6a. The direction of the vectors drawn as open arrows has been adjusted to the vertical scale. To adjust the direction of the vectors, the y-component of each vector magnitude is re-scaled; that is, increased by a factor of five. The plotting angle is re-computed using the larger y-component and the original x-component. The new vector is re-plotted with the original magnitude and a new direction.

Note that in Figure 6b, the vectors copied from the true-scale plot (closed arrowheads) indicate that the flow will intersect the mesh near the center of the right side of each element. The adjusted vectors (open arrowheads) indicate that the flow will intersect the upper right corner of each element, which preserves the spatial context of the true-scale model. Thus, the adjusted vectors result in a better display of the sense of direction relative to the modeled domain.

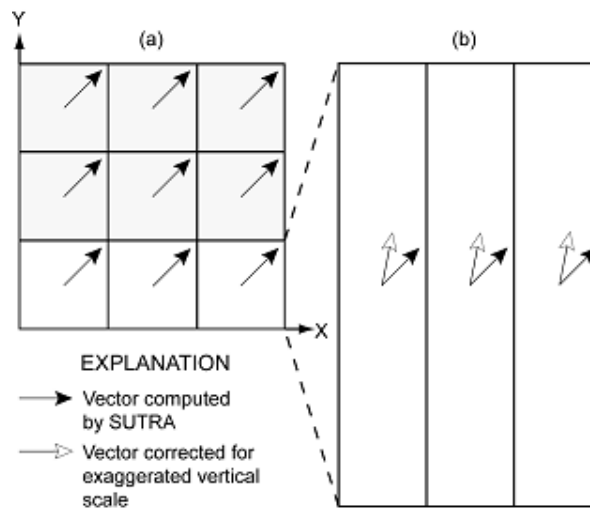


Figure 6. (a) Velocity vectors with equal x and y scaling plotted on schematic finite-element mesh. (b) Velocity vectors re-plotted in bottom row of elements with exaggerated (5:1) y-axis.

Although vectors plotted at true scale give the best sense of the true, i.e. real world, direction of flow, true scale directions when displayed on an exaggerated plot, result in a distorted sense of the direction of flow relative to the model domain, and misinterpretations of the overall flow field may result. As with any plot that is displayed with an exaggerated axis, caution should be used when evaluating velocity vectors that have been plotted in exaggerated space.

## **APPENDIX C - Using mesh specifications to define a region of vectors**

### ***SUTRA 3D meshes***

The finite elements used for **SUTRA** 3D meshes are generalized hexahedra. Each element has 6 sides, 12 straight edges, and a total of 8 nodes, one at each vertex. In the current (2D3D.1) version of **SUTRA**, the 3D-mesh structure is restricted to a nodal arrangement that is logically rectangular. The 3D connectivity for a logically-rectangular mesh is the same as a rectangular solid containing a regular 3D finite-difference grid (with nodes at the corners of the cells). Internally, each node is connected to eight and only eight elements.

Though restricted, a logically-rectangular mesh structure still allows great flexibility in mesh design. Unlike, finite-difference grids, the geometry of finite-element sides and edges need not be rectilinear. Portions of the mesh can be ‘squashed’ or ‘stretched’, and elements or element edges need not be co-planar. This is useful for incorporating geologic features in the finite-element structure such as curved layers or intersecting fractures. For additional information on the construction of logically-rectangular meshes refer to the **SutraPrep** documentation (Provost, 2002).

### ***Defining a region of vectors in a sheet of elements***

For viewing velocity vectors produced by 3D **SUTRA** simulations, users have an optional method of selecting a region of vectors. The structure of **SUTRA**'s logically-rectangular meshes allows a region of vectors to be defined that is contained within a contiguous sheet of elements, one element thick, and aligned with the node numbering directions. The mesh geometry available in **SUTRA** is flexible, and the centroids of the elements in the sheets do not have to be co-planar. Thus, this method of selecting a sheet of elements may be useful for delineating a region of vectors in meshes that incorporate geologic features in the finite-element structure that are curvilinear or not aligned with the X-Y-Z coordinates.

The user is required to identify and locate a sheet of elements and, therefore, must be familiar with the geometry of the finite-element mesh, and the node numbering directions relative to the **SUTRA** X-Y-Z coordinate system. To extract a sheet of elements: first, the user identifies the orientation of sheets of elements by specifying any two of the three node numbering directions. This defines all the sheets of elements aligned with these two numbering directions. The sheets are stacked along the remaining direction.

Second, a specific sheet of elements is identified by specifying the number of the element sheet. The sheet number is simply obtained by counting elements along the numbering direction not in the plane of the element sheet. **SutraPlot** automatically computes and displays the total number of element sheets in this direction.

By using this process, the set of elements extracted is contained within a contiguous sheet of elements aligned with the node numbering directions [Note: these directions may or may not be aligned with the X-Y-Z coordinates of the mesh.] For output in a 2D graphic window, the user must specify a projection plane on which to project the velocity vectors. **SutraPlot** limits projections to one of the three principal X-Y-Z coordinate planes. Caution must be used in evaluating projected data from meshes where the element sheets are not closely aligned with the **SUTRA** X-Y-Z coordinate system.