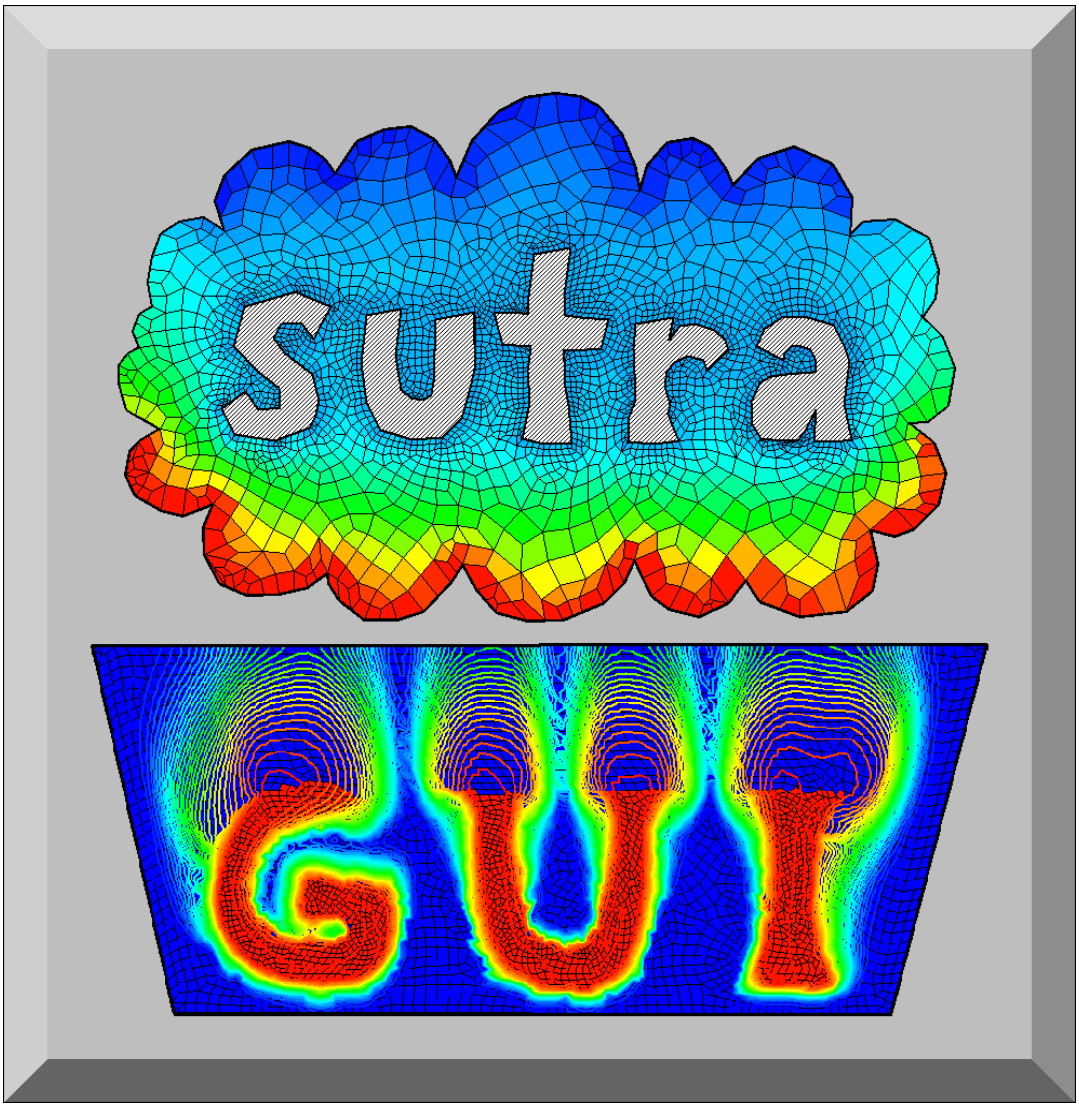
U.S. Department of the Interior
U.S. Geological Survey

A Graphical-User Interface for the U.S. Geological Survey's SUTRA Code using Argus ONE

(for Simulation of Variable-Density Saturated-Unsaturated Ground-Water Flow with Solute or Energy Transport)

Open-File Report 97-421
Version of: January 3, 2001



USGS
science for a changing world

About the cover:

The image consists of two projects created with the Argus-SUTRA GUI.

The upper project, "SUTRA", is an irregular SUTRA finite-element mesh in which the spatially-varying values of hydraulic conductivity are represented elementwise by color. The interpolation method is Inverse Distance Squared. The letters were created in the ***Domain Outline layer***, as open islands within the external domain boundary, resulting in holes in the mesh. Mesh density has a higher value (i.e. finer mesh) along the letters than at the external boundary.

The lower project, "GUI", is a constant-density areal transient transport SUTRA simulation, in which the solute concentrations are represented by the colored contours. The mesh is irregular and has a constant density. The letters are regions of very high hydraulic conductivity-1000 times greater than the rest of the domain. Each letter contains one or two points of specified hydraulic head set at a value of 100. Inflowing fluid at these points has a concentration of 100, while the initial background concentration is zero. The upper horizontal boundary of the domain is set to a specified hydraulic head of zero. Thus, the fluid flows from the points of specified head in each letter, through the highly permeable letter, towards the upper model boundary. The high-head points were chosen such that the solute would trace out the shape of each letter in its path towards the outflow boundary. The mesh is colored by the hydraulic conductivity value of each finite element, and the contours are drawn using the postprocessing module of the Argus-SUTRA GUI.

A Graphical-User Interface for the U.S. Geological Survey's SUTRA Code using Argus ONE

(for Simulation of Variable-Density Saturated-Unsaturated Ground-Water Flow with Solute or Energy Transport)

by Clifford I. Voss, David Boldt, and Allen M. Shapiro
U.S. Geological Survey, Reston, VA

U.S. GEOLOGICAL SURVEY
Open-File Report 97-421

Reston, Virginia
1997

U.S. DEPARTMENT OF THE INTERIOR

Bruce Babbitt, Secretary

U.S. GEOLOGICAL SURVEY

Mark Schaefer, Acting 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.

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

For additional information write to:

Chief, Office of Ground Water
U.S. Geological Survey
411 National Center
Reston, VA 20192

Copies of this report can
be purchased from:

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

Table of Contents

Abstract 1

Overview

Introduction 3
- Computing Platforms and Required Files 4
- Availability of Plug-in Extension (PIE) Code for SUTRA 4
- The Argus ONE Paradigm 4
- Acknowledgments 5

System Requirements and Installation 5

Simple Description of Software Use 7

Simple Step-By-Step Applications of The Argus-SUTRA GUI 8
- Areal Ground-Water Flow Model 8
- Areal Solute Transport Model 13
- Areal Solute Transport Model With Barrier 15
- Areal Model With Transient Solute Transport 18
- Henry Seawater Intrusion Problem with Variable-Density Flow 19

User's Guide

Starting Argus ONE and the SUTRA Interface 25

SUTRA Project Information Dialog 25
- Title Window 26
- SUTRA Interface-Configuration Window 27
- SUTRA Non-Spatial Information dialog 29
  - Headings 32
  - Modes, Numerical Controls 33
  - Temporal Controls 34
  - Output 35
  - Iteration 36
  - Fluid Properties 37
  - Solid Matrix, Adsorption 38
  - Production, Gravity 39

The Argus ONE Window and Argus ONE Layers 40
- Saving and Retrieving Argus ONE-SUTRA Projects 41
- Layer List Window 41
- Visibility of Layers 42
- Active and Inactive Layers 43
- Layer Types 43
- Layers and Their Parameters 45
- Interpretation of Contours and Point Data 45
- Visualizing and Inspecting Values in Layers 52
- Adding or Changing Layers 52
  - A Warning about Layer and Parameter Names 53

SUTRA Model Layers 53
- Domain Outline 53
- Mesh Density 56
- Fishnet Mesh Layout 56
- SUTRA Mesh 60
  - Visualizing Geospatial Information Linked to SUTRA Mesh Elements 65

Visualizing Geospatial Information Linked to SUTRA Mesh Nodes 66
Observation Areas 67
Geospatial Information Layers 67
Hydrogeology 72
Thickness 72
Porosity 72
Permeability / Hydraulic Conductivity 72
Dispersivity 72
Unsaturated Properties 73
Hydrologic Sources 74
Sources of Fluid 75
Sources of Solute / Energy 77
Hydrologic Boundaries 78
Specified Pressure / Hydraulic Head 78
Specified Concentration / Temperature 79
Hydrologic Initial Conditions 80
Initial Pressure/Hydraulic Head 80
Initial Concentration/Temperature 80
Map or Point Data Layers 80
Map Layer 80
Point Data Layer 81
Creating SUTRA Input-Data Files and Running SUTRA 81
Creating the Mesh and Renumbering 81
Exporting Data and Running SUTRA 82
Files Created by SUTRA Simulations 83
Running SUTRA Outside of Argus ONE 84
Post-Processing: Visualizing SUTRA Simulation Results 84
Starting the Post-Processor 85
Post-Processing Plot Manipulation 87
Exporting Post-Processing Plot Graphics 88
References Cited 89

Appendixes

Appendix A 91
SUTRA Output files for Post-Processing, d3 and d4 91
Appendix B 93
Troubleshooting Failed SUTRA Simulations 93
Appendix C 94
Plotting a Non-Converged Solution 94
Appendix D 95
Setting the Level of Error Checking Done for SUTRA Input Data 95
Appendix E 96
Adding and Linking New Layers 96
Adding a Precipitation Data Layer and Linking It to Fluid Sources Layer 96
Calculating a Thickness From Two New Layers or Two New Parameters 97
Appendix F 100
Current Limitations 100

Figures

1. Lakes in Areal Ground-Water Flow Step-by-Step Example 9
2. Model Boundary in Areal Ground-Water Flow Step-by-Step Example 10
3. SUTRA Mesh in Areal Ground-Water Flow Step-by-Step Example 11
4. Head and Vector Plot in Areal Ground-Water Flow Step-by-Step Example 12
5. Concentrations and Velocities in Areal Solute Transport Step-by-Step Example 14
6. Low-Conductivity Barrier in Areal Solute Transport Step-by-Step Example 16
7. Mesh Representation of Low-Conductivity Barrier, areal solute transport step-by-step example 17
8. USGS-SUTRA PIE Title Window 26
9. SUTRA Interface-Configuration Window 27
10. SUTRA Non-Spatial Information dialog; Headings 32
11. SUTRA Non-Spatial Information dialog; Modes, Numerical Controls 33
12. SUTRA Non-Spatial Information dialog; Temporal Controls 34
13. SUTRA Non-Spatial Information dialog; Output 35
14. SUTRA Non-Spatial Information dialog; Iteration 36
15. SUTRA Non-Spatial Information dialog; Fluid Properties 37
16. SUTRA Non-Spatial Information dialog; Solid Matrix, Adsorption 38
17. SUTRA Non-Spatial Information dialog; Production, Gravity 39
18. Argus ONE Window 40
19. Layer List Window, expanded 42
20. Exact Contour Method 48
21. Nearest Contour Method 49
22. Interpolation Method 50
23. NN2D Contour Method 51
24. InvDistSq Contour Method 51
25. Domain Outline, example showing vertices on a closed contour 54
26. Contour Information Dialog 55
27. Contour Tool 55
28. Fishnet Mesh Outline, example 58
29. Completed Fishnet Mesh, for layout shown in Figure 28 59
30. Irregular Mesh, with inactive area, for domain shown in Figure 25 on page 54 61
31. Elementwise Display of Parameter Values with a Color Map, example 65
32. Post-Processing Tool, drawing contours 66
33. Layers Dialog 68
34. Plotting Dialog Window 85
35. Post-Processing Graphics, Concentration and Velocity, example 86
36. Contour Diagram Window 87
37. Concentration Contours with Small Delta, example 88
38. Constructing an Equation in the Expression Dialog, example 98

Tables

1. Default Values for SUTRA Non-Spatial Information dialog for General Case 30
2. Default Values for SUTRA Non-Spatial Information dialog for Special Cases 31
3. Argus-SUTRA GUI Layers and Parameters 46
4. Parameters of the SUTRA Mesh Layer 63
5. Unit Abbreviations 69
6. Default Background Values for Layer Parameters 70

A Graphical-User Interface for the U.S. Geological Survey's **SUTRA** Code using **Argus ONE**

(for Simulation of Variable-Density Saturated-Unsaturated Flow with Solute or Energy Transport)

Clifford I. Voss, David Boldt and Allen M. Shapiro

ABSTRACT

This report describes a Graphical-User Interface (GUI) for SUTRA, the U.S. Geological Survey (USGS) model for saturated-unsaturated variable-fluid-density ground-water flow with solute or energy transport, which combines a USGS-developed code that interfaces SUTRA with Argus ONE, a commercial software product developed by Argus Interware. This product, known as Argus Open Numerical Environments (Argus ONE™), is a programmable system with geographic-information-system-like (GIS-like) functionality that includes automated gridding and meshing capabilities for linking geospatial information with finite-difference and finite-element numerical model discretizations. The GUI for SUTRA is based on a public-domain Plug-In Extension (PIE) to Argus ONE that automates the use of Argus ONE to: automatically create the appropriate geospatial information coverages (information layers) for SUTRA, provide menus and dialogs for inputting geospatial information and simulation control parameters for SUTRA, and allow visualization of SUTRA simulation results. Following simulation control data and geospatial data input by the user through the GUI, Argus ONE creates text files in a format required for normal input to SUTRA, and SUTRA can be executed within the Argus ONE environment. Then, hydraulic head, pressure, solute concentration, temperature, saturation and velocity results from the SUTRA simulation may be visualized. Although the GUI for SUTRA discussed in this report provides all of the graphical pre- and post-processor functions required for running SUTRA, it is also possible for advanced users to apply programmable features within Argus ONE to modify the GUI to meet the unique demands of particular ground-water modeling projects.

OVERVIEW

INTRODUCTION

The numerical model is a modern tool of the hydrologist involved in practical or theoretical evaluation of subsurface flow and transport processes. Numerical models cannot be considered as exact representations of the real world because the complexity of natural hydrogeologic systems is not easily captured in a discrete model representation. The strongest application of numerical modeling is to test hypotheses concerning the major hydrogeologic controls on subsurface system behavior. The most popular use of numerical models is to predict the response of subsurface systems to applied stresses. Both of these uses often are time consuming because numerical models have complex data requirements, and it is rarely practical for hydrologists to carry out thorough analyses. The 'drudgery' of numerical model application can be alleviated and the modeling process facilitated by the development and use of graphical-user interfaces (GUIs) which can manage and synthesize geospatial information and simulation control parameters, to create meshes and other formatted input data for the numerical models, and to visualize the simulation results.

This report describes a GUI developed for SUTRA, the U.S. Geological Survey (USGS) model for two-dimensional, finite-element saturated-unsaturated variable-density ground-water flow and solute or energy transport (Voss, 1984). This GUI for SUTRA was developed using commercially available software developed by Argus Interware. The Argus Interware product, known as Argus Open Numerical Environments (Argus ONE™), is a model-independent, programmable system with geographic-information-system-like (GIS-like) functionality that includes automated gridding and meshing capabilities for synthesizing geospatial information and linking it with finite-difference and finite-element discretizations. The programmable nature of Argus ONE allows geospatial information and simulation control parameters to be exported to ASCII files that can be read by the numerical model, in this case, SUTRA. No changes are required in SUTRA in order for Argus ONE to create its input data sets. A modified version of SUTRA has been released by USGS to provide additional output files that are used by Argus ONE (or other applications) for post-processing.

A few attributes of Argus ONE make it particularly flexible for model developers who wish to easily create a model GUI and for users of the models. Argus ONE allows externally-developed Plug-In Extensions (PIEs) (executable codes loaded into the memory of Argus ONE at run time), which appear as an integral part of the Argus ONE user environment. Such PIEs may be developed independently by users (Argus Interware, 1997b) and are similar in use to user-programmed macros for common spreadsheet and word processing software. For example, PIEs can be developed for purposes such as spatial interpolation, grid and mesh generation, geostatistics, and control of model pre- and post-processing. In addition, Argus ONE allows users to import geospatial information from other GIS-type applications to provide compatibility with existing user projects.

The GUI for SUTRA consists of a public-domain, freely-available PIE, developed by USGS, that must be used in conjunction with the Argus ONE commercial package. Together, these codes provide a fully functioning graphical pre- and post-processor, that significantly reduces the time and effort required for use of SUTRA as a hydrologic tool. In addition, the advanced user may apply programmable features within Argus ONE to extend or modify the Argus-SUTRA GUI to meet the unique demands of any modeling project. This report, however, is a description of the use of the USGS PIE together with Argus ONE as a graphical pre- and post-processor for SUTRA, and is not intended as a description of the programming that has been used to develop the PIE.

It is assumed herein that the reader has a knowledge of the basic data requirements and underlying principles for using the USGS SUTRA code. Voss (1984) provides thorough discussions of the principles underlying SUTRA and its application to hydrologic problems. Moreover, it is assumed that the reader has a knowledge of the general attributes and use of Argus ONE. Information about Argus ONE can be obtained from the Argus ONE User's Guide (Argus Interware, 1997a) and the World-Wide Web site for Argus Interware (**http://www.argusint.com**), from which demonstration versions and documentation may be downloaded.

COMPUTING PLATFORMS AND REQUIRED FILES

Although Argus ONE operates on various computing platforms, the PIE for SUTRA has been compiled only for computers operating under the Windows 95™ and Windows NT™ operating systems. Future releases may support UNIX™ and Macintosh™ operating systems as well. The PIE appears as the file SUTRA.DLL. The file extension ".DLL" indicates that the file is a dynamically linked library, which is the name of the Microsoft™ technology that allows external codes to be loaded as integral parts of software applications when the application is run. The file SUTRA.DLL combines both the pre-processor for SUTRA and the post-processor for visualizing pressure, hydraulic head, concentration, temperature, saturation and velocity vectors from solutions generated by SUTRA simulations. Additionally, the PIE allows the user to choose to generate "fishnet-type" meshes for SUTRA rather than the irregular mesh typically generated by Argus ONE's MeshMaker™ quadrilateral meshing engine. This PIE requires an executable version of the commercial Argus ONE package (version 4 or higher) to operate. Furthermore, the PIE described in this report is referred to as version 1, because enhancements are anticipated as users' needs are evaluated. Four other files, provided by the USGS, are required as described in the section, "System Requirements and Installation" (page 5).

AVAILABILITY OF PLUG-IN EXTENSION (PIE) CODE FOR SUTRA

The source code for the PIE (developed with Microsoft Visual C++ version 5.0) is available for those who wish to modify or augment the graphical pre- and post-processing capability developed in this version. This code is in the public domain and may thus be obtained free of charge from the U.S. Geological Survey and may be redistributed (and modified) freely. Both the USGS SUTRA code and the SUTRA PIE for Argus ONE (with this documentation and examples) may be obtained from USGS on the World Wide Web at URL **http://water.usgs.gov/software/ground_water.html**.

THE ARGUS ONE PARADIGM

Argus ONE provides a user environment in which geospatial (map-type) information (or coverages) may be easily manipulated in preparation for use as input to numerical ground-water (or other types of) models. As in GIS systems, the various types of geospatial information are saved in coverages or "layers" which the user may view and interact with on the computer screen. New layers may be created by the user as mathematical or logical functions of other layers, from data imported into Argus ONE, or from spatial contours drawn by the user directly on the computer screen.

Much of the power of this environment comes from the ability to automatically assign values derived from the geospatial information layers to cells, nodes and elements of numerical-model meshes. Three types of meshing are available in Argus ONE: finite-difference grids, and triangular and quadrilateral finite-element meshes. Meshes of various types and designs are easily generated, and when an existing mesh is modified, the geospatial information is automatically reassigned to the new mesh. The resulting mesh parameters may then be exported to input files for the desired numerical model.

The geospatial coverages may be used interchangeably for any simulation model coupled with Argus ONE. For example, coverages prepared for SUTRA may used with MODFLOW96 (Harbaugh, A.W. and McDonald, M.G., 1996), another USGS ground-water flow model coupled to Argus ONE by Shapiro and others (1997), and with MOC3D (Konikow and others, 1996), a solute transport code linked to MODFLOW96 (L.F. Konikow, U.S. Geological Survey, oral communication, 1997).

The Argus ONE approach offers the advantage for numerical model developers that the ground-water numerical model is completely separate from the commercial Argus ONE software, allowing developers to use the Argus ONE functionality to its fullest extent while separately maintaining and modifying the numerical models. Argus Interware's "open environment" concept provides model developers with the capability of configuring many fundamental aspects of Argus ONE by creating files and programs external to the commercial portion of the software. These external files are much like macros that may be written for many commercial spreadsheet and word-processing programs. In the Argus ONE case, the files configure a seamless connection between a particular numerical code and the Argus ONE geospatial information and meshing capability. These are relatively straightforward to create by beginning programmers of Argus ONE for a new numerical model. For example, to connect the USGS-SUTRA code with Argus ONE, it is only absolutely necessary to create two such files (a ".met" export template file, and a ".lay" layer structure file). These files have already been created by USGS for the SUTRA model.

Argus's open environment also provides the capability for user-written programs to appear within Argus ONE menus, and makes it possible for these programs to exchange information with Argus ONE. Using this feature of the Argus open environment requires more complex programming, for example, in C++, Java or Python languages, but allows Argus ONE to be customized in the most-convenient manner for model users. For SUTRA, this has been done by creating three other external files (a C++ program generated ".dll" file that creates a user dialog which appears within Argus ONE, and a ".val" and ".ini" file containing various default data for the model input and the GUI interface, respectively). The model developer may decide on the type of distribution that will be allowed for the associated files; the five USGS SUTRA configuration files for Argus ONE are freely available to the public.

ACKNOWLEDGMENTS

The authors acknowledge the significant contributions made by Argus Interware personnel, J. Margolin, S. Dolev and Y. Ben-Israel, in the development of the Argus-SUTRA GUI. Argus Interware provided the initial C++ code for the tabbed-dialog window, the fishnet meshing, and the postprocessing, according to the specifications of the authors. Argus Interware also provided reliable expert support to all of our questions and further development and aided in modifications to the codes. The software described herein was developed using Argus ONE™ and Argus PIE technology.

SYSTEM REQUIREMENTS AND INSTALLATION

The PIE developed for SUTRA and described in this report uses Argus Open Numerical Environments (Argus ONE™) version 4. More recent versions of Argus ONE also are compatible. Argus ONE operates on various computing platforms, however, the PIE discussed in this report has been developed only for computers operating under Windows 95 and Windows NT.

Operation of Argus ONE requires at least an Intel 486™ 100 Mhz processor with 16 MB of RAM, 7 MB of free disk space and an SVGA video card capable of displaying 256 colors; however, a faster computer (e.g. Intel Pentium™ 150 MHz or better), more RAM (24 MB or greater) and capability to display 65,000 colors at higher resolutions is recommended.

Instructions for installing Argus ONE are given by Argus Interware when the software is purchased or downloaded from the Argus ftp site. Additional information about Argus ONE can be found on Argus' World-Wide Web site **http://www.argusint.com/**.

The PIE for SUTRA has been developed using Microsoft Visual C++™ (version 5.0). The source codes for the PIEs are available at the USGS World-Wide Web site (**http://water.usgs.gov/software/**) along with the files discussed below; this information appears under the category "Ground Water". PIEs can be written in other languages, and Java or Python, for example, may be used in future programming.

To use the post-processing features of the Argus-SUTRA GUI, it is necessary to run the **V09972D** version of the SUTRA code, which is also found on this site. The GUI for SUTRA consists of five files, **SUTRA.DLL**, **SUTRA.VAL**, **SUTRA.INI**, **SUTRA.MET**, and **SUTRA.LAY**. The latter two are always required for Argus ONE applications and are in standard Argus ONE formats, while the first three are associated with the PIE written as an external code for the Argus-SUTRA GUI application. All files but the DLL are ordinary text files that can be modified with any editor.

SUTRA.DLL is the PIE (written in C++) associated with pre-processing of simulation controls and geospatial information for SUTRA, and the post-processing visualization of pressure, hydraulic head, concentration, temperature, saturation and velocity vectors from SUTRA simulations.

SUTRA.VAL consists of default values for simulation control parameters used with **SUTRA.DLL**.

SUTRA.INI contains information used by **SUTRA.DLL** for the top-level pre-processing dialog which allows the user to select the type of SUTRA simulation problem to be constructed.

SUTRA.MET is the Argus ONE export template for exporting simulation control and geospatial information into the usual ASCII files that are read by SUTRA.

SUTRA.LAY contains the description of the GIS layers to be used in the Argus ONE environment for the Argus-SUTRA GUI and is modified by **SUTRA.DLL** to create the layers required for the type of simulation problem the user chooses. The information in this file is dynamically modified by the PIE, **SUTRA.DLL**, according to information in **SUTRA.INI** when new SUTRA projects are created in Argus ONE.

To use the SUTRA-Argus ONE GUI, these files should be placed in the directory **\Program Files\Argus Interware\ArgusPIE\SUTRA**. Argus ONE searches the ArgusPIE directory and sub-directories, then loads the PIEs that it encounters into the Argus environment.

The executable version of SUTRA (**SUTRA.EXE**) should be located in a directory with the pathname **C:\SUTRA**; the full pathname of the executable for SUTRA is assumed to be **C:\SUTRA\SUTRA.EXE**. If the user prefers to locate the executable version of SUTRA in another directory, the **SUTRA.MET** file in the **\Program Files\Argus Interware\ArgusPIE\SUTRA** directory must be edited using a text editor that can save files in an ASCII format. The pathname of the executable version of SUTRA is given near the top of the file. This line currently reads:

```
Set Variable: SutraPath:= "C:\SUTRA\sutra.exe"
```

The pathname in quotation marks should be replaced with the full pathname of the executable version of SUTRA.

Note that **SUTRA.EXE**, version **V09972D**, is a revised version of SUTRA that creates two new output files which are used in the post-processing step of the GUI. See "SUTRA Output files for Post-Processing, d3 and d4" (page 91) for examples of these files. Simulation of unsaturated flow or time-dependent boundary conditions in SUTRA requires use of a FORTRAN compiler.

SIMPLE DESCRIPTION OF SOFTWARE USE

This section provides a brief description of how the Argus ONE environment with the USGS SUTRA PIE is used to create and evaluate numerical simulations with SUTRA. More detail on each step may be found in subsequent sections of this documentation.

1. After starting Argus ONE, the *Argus* ONE *Window* appears and the user begins pre-processing for SUTRA by selecting **New SUTRA Project**.

2. Choices then appear for the type of simulation problem desired, e.g.: areal solute transport, cross-sectional variable-density flow with solute transport, saturated-unsaturated flow with energy transport, or other available types. This choice causes the PIE to structure the kinds of geospatial coverages (information layers) required for such a SUTRA simulation and automatically makes them available to the user for data entry and manipulation.

3. Next, the user may enter simulation control parameters (those that are not spatially-dependent, such as time step size) into an interactive *tabbed-dialog* box that appears on the computer screen. Upon completing data entry or editing of these values, the user closes the dialog and returns to the *Argus* ONE *Window*.

4. The user should then modify the default information in any geospatial information layer, by manually drawing closed or open contours, or points to represent the desired spatial distributions of hydrogeologic and hydrologic parameters, sources, sinks and boundary conditions, as well as a desired distribution of finite-element mesh density. As an alternative to drawing, any of these spatial distributions may be directly imported from other applications that can generate either simple **text** files, **DXF** (AutoCAD format) files, or **Shape** (ArcView™ format) files.

5. Next, the user requests that Argus ONE create the finite-element mesh. Two mesh types are available, a general irregularly-connected mesh of quadrilaterals (normally provided in Argus ONE), and a "fishnet mesh" of regularly-connected quadrilaterals (provided by the SUTRA PIE). Before finally proceeding to run SUTRA, the user may modify any of the spatial or non-spatial information already input.

6. The user then selects the ***SUTRA Mesh*** layer, and from the menu **PIEs**, proceeds to “export” the geospatial and non-spatial information, at which time Argus ONE writes out the standard input-data files for SUTRA to the selected directory, and runs the SUTRA simulation. When the simulation is complete, the user may choose to plot any of the simulation results within Argus ONE in a post-processing step provided by the SUTRA PIE.

The power of the GUI approach in hydrogeologic hypothesis testing and practical modeling should be apparent to any experienced modeler at this point. It is simple to return to any type of spatial information already input to the SUTRA-Argus ONE environment, easily make major modifications to it or the finite-element mesh, re-export from Argus ONE, run SUTRA, and graphically evaluate results of new simulations. Each cycle of changing information, running SUTRA, and inspecting results, can take as little as a few minutes.

SIMPLE STEP-BY-STEP APPLICATIONS OF THE ARGUS-SUTRA GUI

This section provides the new user the opportunity to begin using the Argus-SUTRA GUI quickly and easily for a few applications without reading any other parts of the documentation. In-depth details and discussions of the various steps, windows and dialogs encountered below may be found in other parts of this report. The following step-by-step examples are intended to be followed in sequence.

AREAL GROUND-WATER FLOW MODEL

This walk-though creates a simple areal steady state ground-water flow model (ignoring solute transport), runs SUTRA, visualizes simulation results, and saves the project for use in subsequent step-by-step examples.

1. Double-click on the Argus ONE icon. This opens Argus ONE.

2. From the **PIEs** menu found along the top of the window, select, **New SUTRA Project...**. This brings up the USGS-SUTRA PIE *Title window*.

3. Click **Continue.** This brings up the *SUTRA Interface-Configuration window*.

4. Here, the type of problem to be simulated is chosen. To select the default type of **AREAL** click on **OK**. This sets up a problem of areal flow with constant-density solute transport using an irregular finite-element mesh, and brings up the *SUTRA Non-Spatial Information dialog*.

5. This window allows the user to specify values for the SUTRA simulation that are not spatially variable, i.e. those that may have only a single value, such as "time step". These may be inspected on the various panes by clicking the tabs found at the top of the window. Rather than making changes here now, accept the default values by clicking **OK**. This brings up a new *Argus* ONE *window,* called "untitled1".

   Default values are given in "Default Values for SUTRA Non-Spatial Information dialog for General Case" (page 30) and in "Default Values for SUTRA Non-Spatial Information dialog for Special Cases" (page 31). The default values set up simulation of steady-state ground-water flow (and solute transport of a non-reactive solute) in a sandy-type aquifer.

6. This is the window in which the model will be designed, run, and evaluated. It contains many layers in a stack; each layer will hold either model or mesh information. Additionally, another window, the *Layer List window* (also called "Layer Floater") appears, in which the user can see which layers are available. This window may be resized to display the full layer names.

   The *Layer List window* shows which information layers are available for the particular problem type (AREAL) chosen in step 5. The window allows the user to control which of the layers will be visible (those with the open eye) and which layer is on top of the stack and thus available for input from the screen. Clicking on an 'eye' toggles the layer visibility, and clicking to the left of an 'eye' makes the layer 'active' (i.e. brings the layer to the top of the stack) and puts a 'check mark' next to the active layer.

7. One way to begin a modeling project is to input hydrogeologic information for the area into the appropriate geospatial information layers provided in the Argus-SUTRA GUI. The layers listed in the *Layer List window*, e.g. ***Porosity***, and ***Hydraulic Conductivity***, have already been assigned

default values by the interface (these values are shown in Table 6). Rather than modify any of these values in this step-by-step example, for simplicity, the defaults will be accepted. A situation will be considered, in which only boundary conditions need be assigned to make a simple run.

Assume that there are four lakes, one at each corner of closed alluvial valley. The lakes control the hydraulic head at each corner, and no flow crosses the bedrock sides of the valley. A hydraulic head value will be specified where each lake exists. Assume that three of the lakes are at the same elevation somewhat higher than the fourth lake.

To enter the lakes into the model, activate the layer ***Specified Hydraulic Head***, by clicking on its 'eye' in the *Layer List window*.

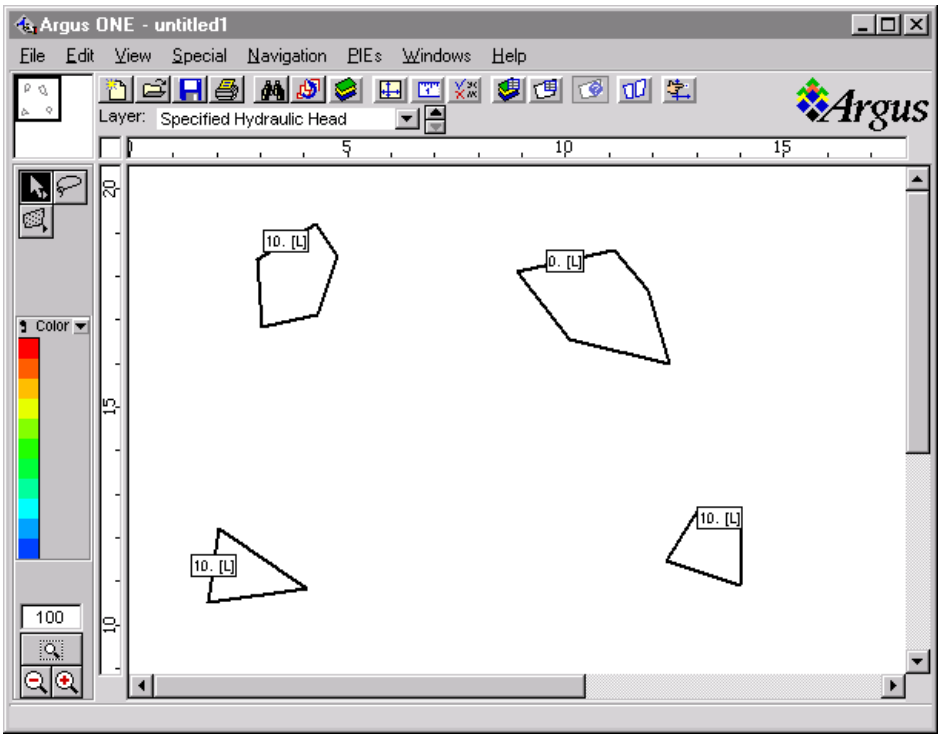


Figure 1. Lakes in Areal Ground-Water Flow Step-by-Step Example

8. Draw a lake by first activating the contour-drawing tool. To do this, click on the small quadrilateral just below the arrow along the left side of the *Argus* ONE *window*. Then move the cursor into the white workspace and click three times creating the first vertices of a closed contour. Try to create a lake that looks like one of the four shown in Figure 1. Then, double-click on the location desired for the last vertex. The *Contour Information dialog* appears.

9. The specified head value and concentration of water potentially entering the aquifer from the lake must be specified in the dialog. This is done by clicking next to **specified_hydraulic_head** (below **Value**) and entering the value, then clicking next to **concentration** and entering a value, and finally clicking **OK**. For the first lake, specify 10.0 for the head, and enter 0.0 for the concentration. For this step-by-step example, all the lakes contain no solute.

10. Draw the other three lakes by clicking and dragging to create small closed regions similar to the ones shown in Figure 1. For each lake, double-click on the last vertex to close the shoreline. This brings up the *Contour Information dialog* for each lake. For two of the remaining lakes, enter 10.0 for the head, and for the last lake, enter 0.0 for the head. For all lakes, enter 0.0 for the concentration.

The entry of hydrogeologic data for simulating ground-water flow in this system is now completed.

11. The model boundary will now be drawn. Make the ***Domain Outline*** layer active by clicking to the left of its 'eye'. This brings the layer to the top of the stack, while the ***Specified Hydraulic Head*** layer remains visible below. The purpose of this layer is to contain model boundary locations. The model boundary will follow the valley walls and the corners will be placed at the center of the lakes.

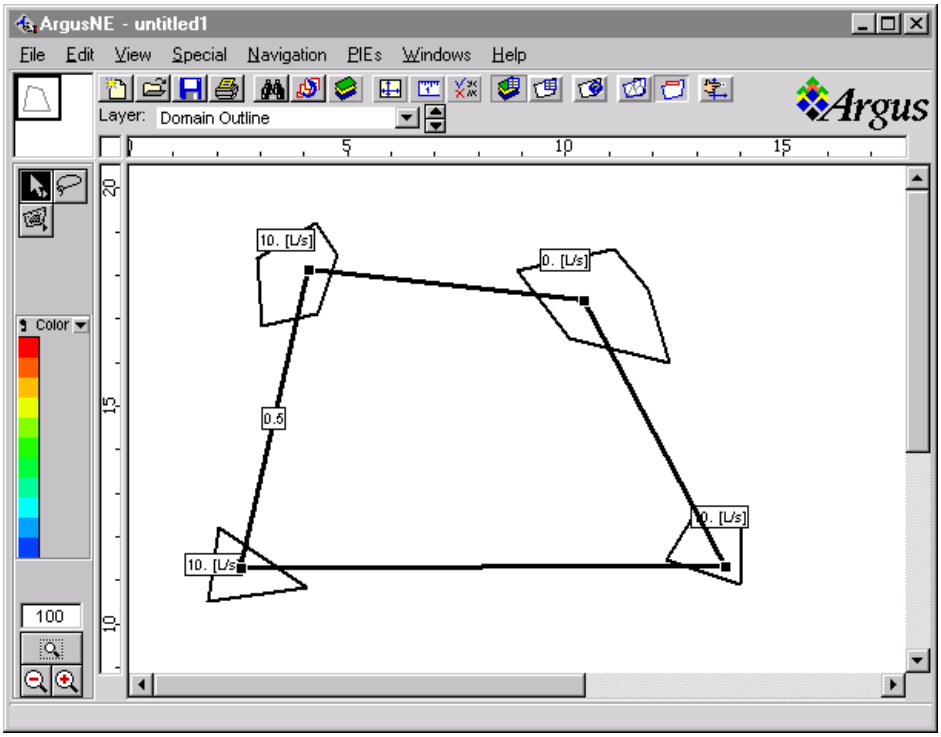


Figure 2. Model Boundary in Areal Ground-Water Flow Step-by-Step Example

12. Draw a model boundary by first activating the contour-drawing tool. Then move the cursor into the white workspace and click in the center of one of the lakes. Proceeding around the workspace, click at the center of each lake. Try to create a domain outline that looks like the one shown in Figure 2. Then, double-click at the center of the last lake to close the model domain contour.

13. The *Contour Information dialog* appears. Here the desired typical size of finite elements to be created by the mesh generator is specified. Type 0.5 in the space below the label, **Value**. This sets the desired width of an element to 0.5 in the units shown in the rulers around the periphery of the workspace. The result should look similar to the model boundary shown in Figure 2. Click **OK** to exit the window.

14. Next, the finite-element mesh will be created. Activate the ***SUTRA Mesh layer*** by clicking to the left of its "eye" in the *Layer List window.*

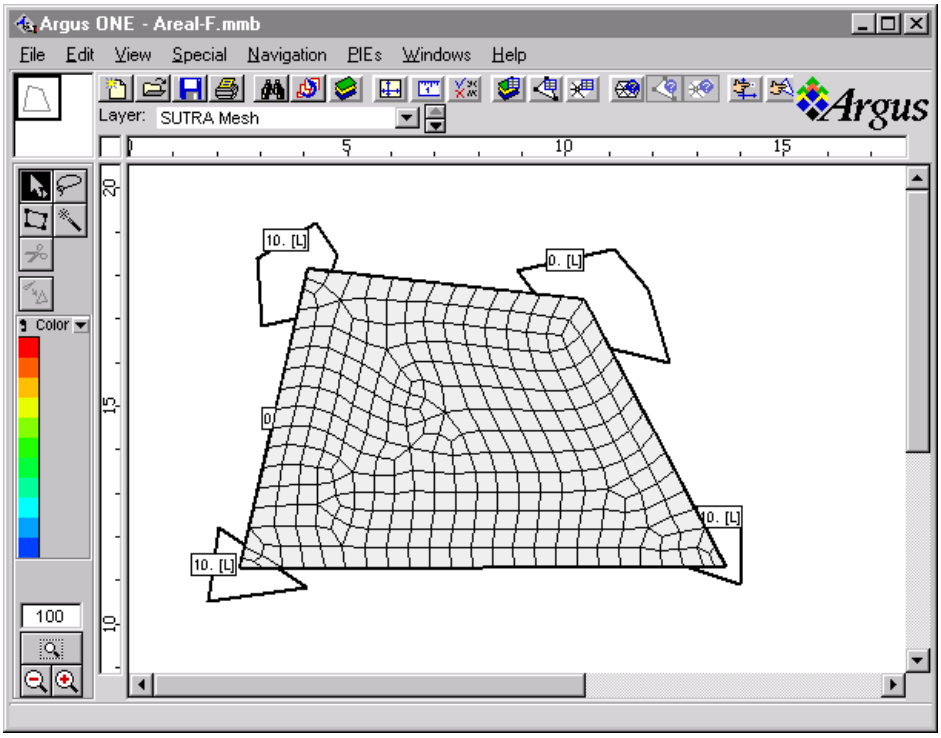


Figure 3. SUTRA Mesh in Areal Ground-Water Flow Step-by-Step Example

15. Click on the 'magic wand' found along the left side of the *Argus* ONE *window* near the arrow, and then click the magic-wand cursor inside the model boundary just drawn. An irregular finite-element mesh containing elements of about 0.5 size is generated and displayed. Figure 3 shows the type of mesh that may be expected.

16. The band-width of a newly-generated mesh always needs to be reduced. Select the **Special** menu along the top of the *Argus* ONE *window*, then select **Renumber...**. This brings up the **Renumber window**. In this window click on **Optimize Bandwidth**. and then **OK**. The mesh numbering is then optimized for the matrix solver currently used by SUTRA.

The entry of data and preparation of the mesh for simulating ground-water flow in this system is now completed.

17. Save the project so far by clicking **File**, and then **Save As...**. Select the desired directory and type in the desired name (e.g. "Areal-F" for areal flow model) and then click on **Save**. A project file called Areal-F.mmb is created in the directory you chose, and the window name becomes the same, as shown in Figure 3.

18. The model information entered now needs to be exported from Argus ONE creating input files that SUTRA requires, and the simulation can then be run. (Note that the ***SUTRA Mesh*** layer must be active in order to export.) In the **PIEs** menu, select **Run SUTRA**. The *Run SUTRA window* appears.

19. This window allows the user to choose only creation of SUTRA input files, or both creation of files and running of SUTRA (which is already selected). Click **OK** to proceed. An *Enter export file name window* appears.

20. Select the directory into which the (two) SUTRA input files will be placed by Argus ONE. Then select the name of the files by typing in the space next to **File Name** (e.g. "Areal-F"). The files created will all begin with the name entered here. and the suffixes will be appended (e.g. Areal-F.d5, Areal-F.d55). Note, ignore the **Save as type** box.

21. Click on **Save** and the export takes place while the barber pole is visible, and then the SUTRA simulation is run while the DOS window is visible. If requested, hit any key to exit the DOS window. Now, the completed SUTRA simulation has created (four) output files, Areal-F.d6, Areal-F.d66, Areal-F.d3, and Areal-F.d4. in the directory that was selected.

22. To visualize the results, from menu **PIEs** select **Sutra Post Proc...**. The *Please select d5 file: dialog* appears.

23. Find the directory selected above for the SUTRA files and double-click the appropriate d5 file (Areal-F.d5). This brings up the *Select SUTRA results to visualize window*.

24. This window contains a list of all results available from the SUTRA simulation for visualization (see Figure 34 on page 85 for an example). Because the simulation was for steady-state conditions, only one time step appears, Click on the **no** below **Hydraulic Head** (or **Pressure**) to change it to **yes** (this selects a contour map of head). Also select a velocity vector map by changing the **no** below **Velocity** to **yes**. Then click on **OK** and the plots are created.

25. Because the ***SUTRA Mesh*** layer was active, the plots appear below the mesh. Bring the plots to the top of the stack by activating the ***SUTRA Post Processing Charts*** layer (click left of the 'eye' in the *Layer List window*).

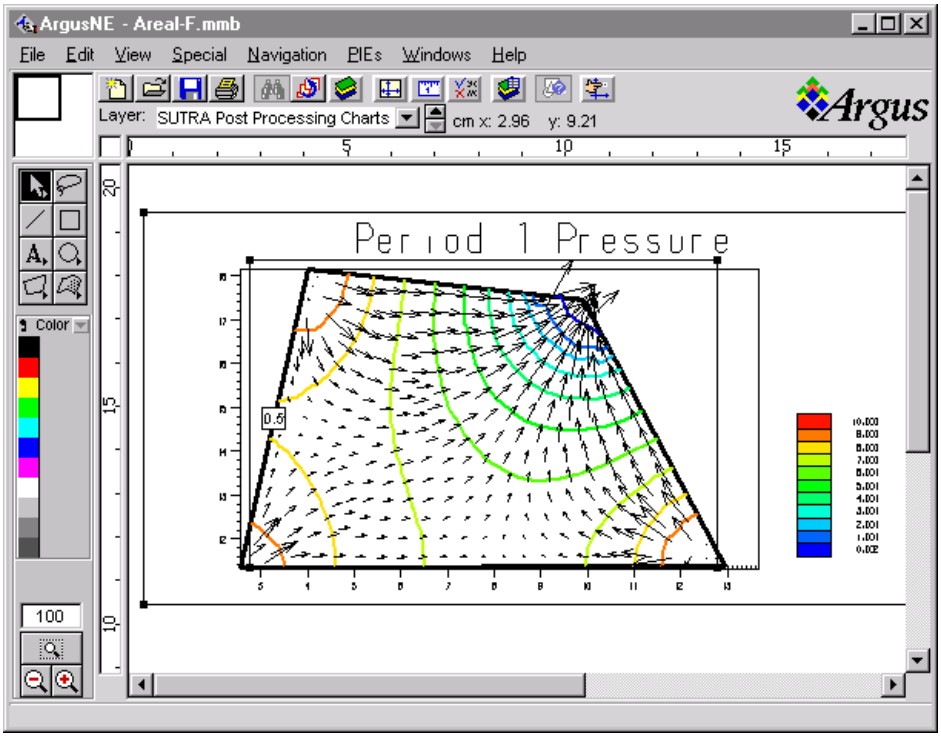


Figure 4. Head and Vector Plot in Areal Ground-Water Flow Step-by-Step Example

26. The plot appears, but is too cluttered because the mesh is also visible. Make the mesh invisible by clicking on the 'eye' next to ***SUTRA Mesh*** in the *Layer List window.* A plot of velocity vectors and head contours, similar to that in Figure 4 is visible.

Ground water flows into the aquifer from three of the lakes and exits at the fourth lake. Hydraulic head in the aquifer decreases near the outflow lake.

27. Save the current state of the project by selecting **File**, and then **Save**.

28. The Argus ONE application may now be closed by selecting **File**, and then **Quit**. The same state that the project was left in will be reproduced when the project is reopened in Argus ONE at any later time.

AREAL SOLUTE TRANSPORT MODEL

This step-by-step example continues the simple areal steady state ground-water flow model and considers steady-state solute transport that occurs in the steady-state flow field. Then SUTRA is run, simulation results are visualized, the visualization is modified, and the project is saved for use in subsequent examples.

1. Double-click on the Argus ONE icon to open Argus ONE.

2. Select **File**, then **Open...,** to bring back the project that was saved in the above step-by-step example. In the *Choose file to open: window* that appears, move to the appropriate directory and double click on the ".mmb" project file that was saved in the above example (e.g. Areal-F.mmb). This returns the user environment to the same state as when the project was previously saved (Figure 4).

Assume that the lake in the corner diagonally opposite the lake with the lowest water level contains a solute with concentration 100 units. The other lakes have no solute, as was previously entered. This information will be treated as the concentration of fluid that enters the aquifer from the lake, and is associated with the specified head boundary condition at the lake.

3. To input this information, activate the layer ***Specified Hydraulic Head***. This brings the layer in which the lakes were entered to the top.

4. Then select the lake diagonally across from the low-head lake by double clicking on it. This brings up the *Contour Information dialog*. Double-click next to **concentration** (below **Value**) and type in 100 to enter the concentration of fluid from this lake that may flow into the aquifer. Note that the **specified_hydraulic_head** is still set to 10. Click **OK** to exit the window. This is the only modification required before exporting the data from Argus ONE and rerunning the simulation.

5. To export and run, activate the ***SUTRA Mesh*** layer again bringing the mesh to the top of the stack. Select **PIEs**, and then **Run SUTRA**.

6. Click **OK** in the *Run SUTRA window*, and Select the directory into which the SUTRA input files will be placed and select the name of the new files that will run a solute transport simulation (e.g. "Areal-T"). Note, ignore the **Save as type** box. Click on **Save** to export and run.

7. If needed, press any key to exit the DOS window in which the SUTRA code was run.

8. This time, results for concentration and velocity will be plotted. First, the old plots of heads and velocities will be removed. Activate the ***SUTRA Post Processing*** layer and choose **Select All** from the **Edit** menu along the top of the window. Then press the delete key to remove both plots.

9. To visualize the new results, from the **PIEs** menu select **Sutra Post Proc...** Then, *Please select d5 file: dialog* appears.

10. Find the directory selected above for the SUTRA files and double-click the appropriate d5 file (e.g. Areal-T.d5). This brings up the *Select SUTRA results to visualize window*.

11. Click on the **no** below **Concentration** to change it to **YES** (this selects a contour map of concentration). Also select a velocity vector map again by changing the **no** below **Velocity** to **YES**. Then click on **OK** and the plots are created.

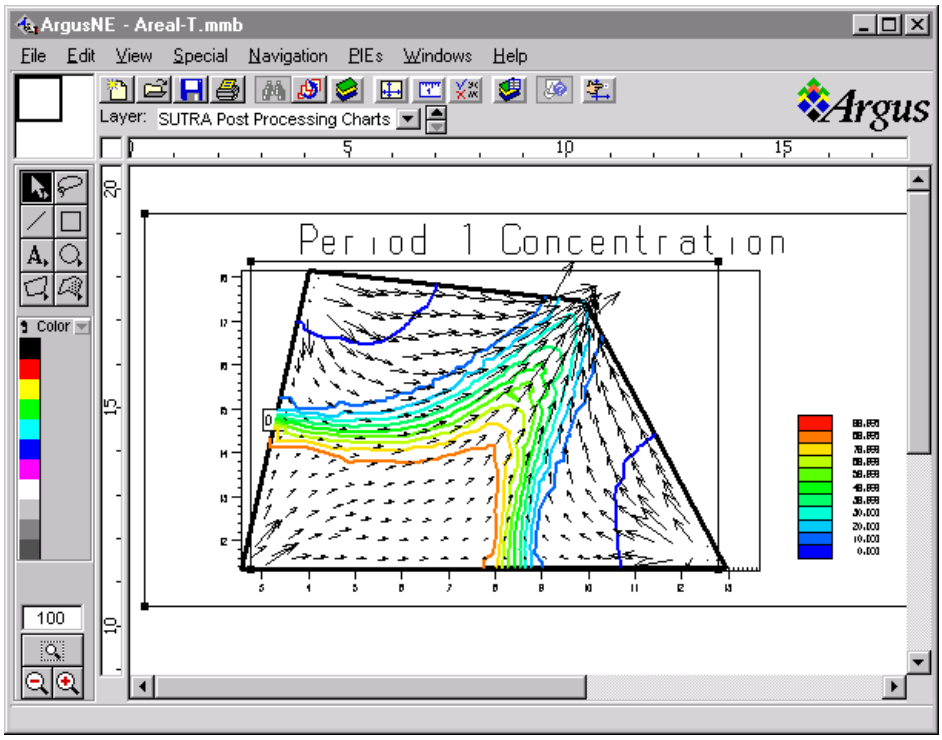


Figure 5. Concentrations and Velocities in Areal Solute Transport Step-by-Step Example

12. The plot is again too cluttered because the mesh is also visible. Make the mesh invisible by clicking on its 'eye'. A plot of velocity vectors and concentration contours (similar to that shown in Figure 5) is visible.

The steady-state solute plume from the lake containing solute fills the central portion of the aquifer, but due to dispersion, is somewhat diluted before it discharges to the low-head lake. The distribution of no-solute waters from the side lakes also is apparent, and these waters increase in solute concentration due to dispersion from the plume before discharging.

13. Each plot may be manipulated somewhat. For example, the contours may be made more dense as follows. Double-click on the contour plot to bring up the *Contour Diagram dialog*.

    Note that there are two plots visible and each has separate controls. The wrong dialog may appear depending on the position on the screen that was double-clicked. The plot frames are visible on the screen and the user should click on a spot bounded by only *one* frame, the one desired. This may require some experimentation. If the wrong dialog appears (i.e. the *Vector Diagram dialog*), simply click on **Cancel** in the dialog and try again.

14. To make contours thicker, select **3 Points** near the bottom of the dialog. To plot more contours, change the numbers found in the box. For example, set Minimum to 1.0, set Maximum to 100.0, and set Delta to 1.0. Click on **OK** and the plot is drawn. The plume is displayed in nearly solid color. (The smoothness of the color scale shown depends on the type of graphics available on the computer, and on the number of colors and resolution selected in the operating system.)

15. Save the current state of the project by selecting **File**, and then **Save As...**, and choose a new project name (e.g. Areal-T) to distinguish this from the previous model of only ground-water flow.

16. The Argus ONE application may now be closed or continue with step 2 of the following step-by-step example.

AREAL SOLUTE TRANSPORT MODEL WITH BARRIER

This walk-though continues the areal steady state transport model and considers the change that occurs in steady-state solute transport observed above if there is a low-conductivity barrier within the aquifer. The barrier is added, its discretization is evaluated visually, the information is exported, and the simulation is run and results visualized. Finally, the barrier's properties are changed and the simulation is run again.

1. Start Argus ONE, and open the ".mmb" project file that was saved in the above example (e.g. Areal-T.mmb). Note that this may also be done by double-clicking the ".mmb" project file name in any 'file manager' window.

2. Activate the ***Hydraulic Conductivity*** layer.

A low-conductivity region will be created that extends part-way across the aquifer.

3. Make the SUTRA Post Processing layer invisible in order to unobscure the workspace.

4. With the closed-contour drawing tool, create a closed contour that extends across one side and part-way across the model domain (see Figure 6) partly blocking the outflow lake. After closing the contour, enter the value 0.0001 for both the **maximum** and **minimum** values of hydraulic conductivity, and click **OK** in the *Contour Information dialog*. (Any contour object or region in the ***Hydraulic Conductivity*** layer has three parameters that contain the spatial distributions of maximum and minimum conductivity components and the angle of the maximum direction to the X-direction in the workspace. Thus, conductivity may be set to be anisotropic.) This decreases the conductivity of the region by ten times relative to the background value (0.001) used for conductivity.

5. Because the conductivity distribution includes a specific 'object' of low value, the interpretation method that Argus ONE uses by default for assigning values throughout the workspace must be changed. (The default method is intended for interpreting contour maps of conductivity.) To change the method, click on the **Layers** button in the *Layer List window*. This brings up the *Layers dialog*.

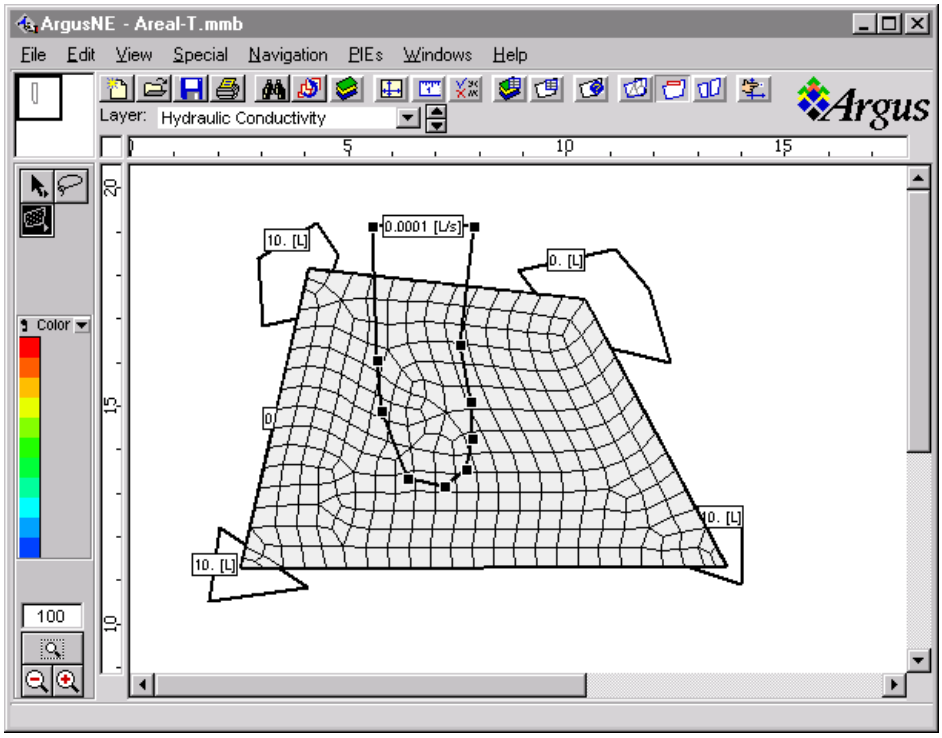


Figure 6. Low-Conductivity Barrier in Areal Solute Transport Step-by-Step Example

6. In the *Layers dialog*, many attributes of available layers may be changed and new ones added or existing ones deleted. Select the ***Hydraulic Conductivity layer*** found in the top half of the dialog by clicking on the line in which it is found. If it is already highlighted then it is not necessary to click on it. (Note that the default values of the layer's parameters are shown in the bottom half of the dialog.)

7. At the bottom of the dialog, **Nearest Contour method** appears as the interpretation method for this layer. Hold the mouse button down over this box and select **Exact Contour method** instead. Then click **Done**.

8. To visually determine whether the barrier shape is discretized sufficiently by the current mesh, make the ***SUTRA Mesh*** layer active, bringing it to the top of the stack of layers.

9. Hold down the mouse button over the triangle next to the word **Color** found along the tool palette on the left edge of the window. A list of parameters of the SUTRA Mesh appears that vary by element (rather than by node). Select **PMAX** (the maximum component of hydraulic conductivity; see Table 4, "Parameters of the SUTRA Mesh Layer" (page 63) and section 7.1 of the SUTRA documentation, Voss(1984).

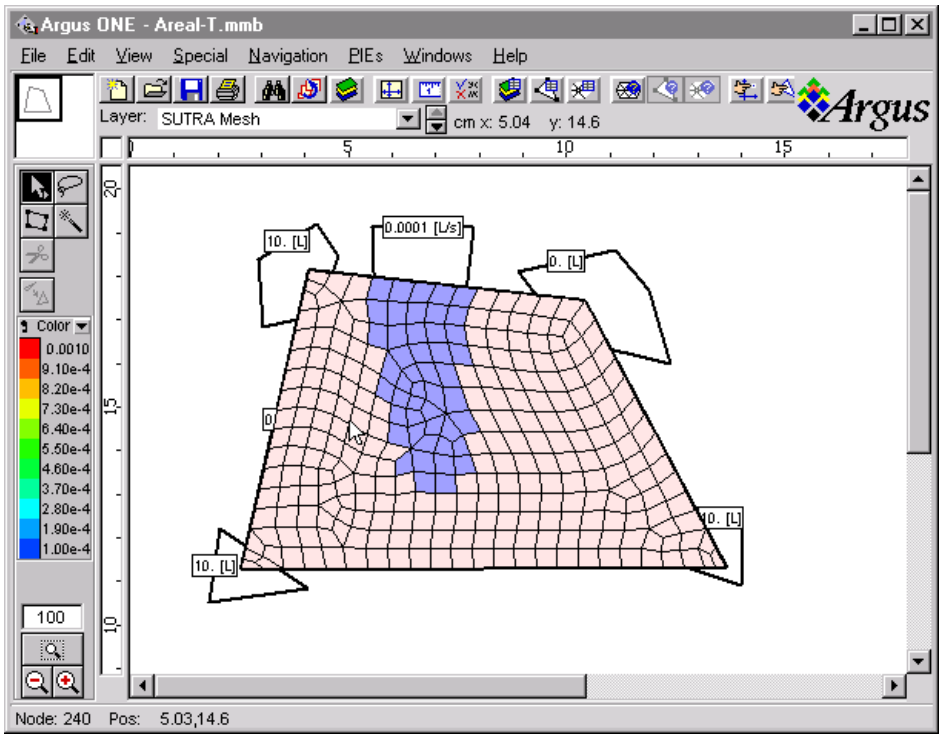


Figure 7. Mesh Representation of Low-Conductivity Barrier, areal solute transport step-by-step example

10. The elements are colored according to maximum conductivity value (see Figure 7 for an example). The barrier may now be examined to determine whether it was sufficiently discretized, for example, to avoid having high-conductivity 'holes' through it because elements are too large. If adjustments are needed, it is suggested that, for simplicity, the barrier is widened rather than refining the mesh at this point. Try adjusting the shape of the barrier by activating the ***Hydraulic Conductivity*** layer, clicking on one of its vertices and dragging it to a new position. Note that the discretization shown in the mesh layer adjusts automatically to the new barrier shape.

11. When the barrier discretization is satisfactory, the project may be exported (use a new name for SUTRA files, e.g. Areal-B) and run. Activate the ***SUTRA Mesh*** layer, export and run as before.

12. Visualize the new steady-state concentration contours and velocity field to evaluate the effect of the barrier. The old plot may be cleared automatically by responding to the request that appears, and the new plot may be viewed by activating the ***SUTRA Post Processing*** layer. Again, if it is too cluttered, make other layers invisible.

13. Evaluate the effect of further reducing the conductivity of the barrier by ten times by activating the ***Hydraulic Conductivity*** layer, double-clicking on it, and adjusting the **maximum** and **minimum** values in the *Contour Information dialog* that appears. Then re-export, re-run and visualize the results.

14. Save the revised project as a new project file (e.g. Areal-B).

AREAL MODEL WITH TRANSIENT SOLUTE TRANSPORT

Rather than evaluating only the steady state of the solute plume, as in the previous step-by-step examples, its evolution over time is simulated and visualized in two ways.

1. Using **File**, then **Open...**, restore one of the projects modeling transport. Alternatively, if Argus ONE was not closed after the last example, continue to the next step.

2. To make the most recent steady-state transport simulation into a transient simulation, the SUTRA Non-Spatial Information must be modified. Bring up this dialog by using **PIEs**, then **Edit Project Info...**.

3. In the *SUTRA Non-Spatial Information dialog*, parameters that are not spatially dependent in the SUTRA code may be modified. They are all assigned default values initially. Click on the tab along the top of the dialog that reads, *Modes, Numerical Controls*.

4. Hold the mouse button down over the entry that says **Steady-state solute transport**, and select **Transient solute transport** instead. Also set the **Hydraulic Head Boundary Condition factor**, **GNUP**=1.E-3, to provide a good match of specified and calculated heads.

5. Click on the T*emporal Controls* tab to adjust the time steps for the simulation. Set the number of time steps to run to 50, **ITMAX**=50, and the time step size in seconds to 100 s, **DELT**=100. In order that the simulation not stop because of the maximum allowed simulation time, set **TMAX**=1.E99.

6. Click on the *Output* tab to adjust the frequency of output to every 5 time steps by setting **NPRINT**=5.

7. Click **OK** to exit the dialog. Save the project (e.g. to Areal-tr.mmb).

8. Set the active layer to ***SUTRA Mesh***, export the project (e.g. to filename prefix Areal-tr) and run the simulation.

9. Inspect transient transport results by using **SUTRA Post Proc...** under menu **PIEs**, and select the d5 file (e.g. Areal-tr.d5) just created. Results for time-steps 5,10, 20 and 40 will be viewed.

10. The *Select SUTRA results to visualize window* appears.

    Because the simulation assumed steady-state flow, head may only be plotted for the first time step (a **no** is available to toggle to **YES** only for **Time Step** 1 under **Head**). Although it appears that **Velocity** is also available to plot for all time steps, the velocities are all the same as the flow is at steady state. **Concentration** for any time step may be viewed.

    To view results for **Time Step** 5, toggle the **no** to **YES** for **Velocity** and **Concentration** in the appropriate line of the dialog, and click on **OK**.

11. Select **OK** when prompted to clear the old plot, then activate the ***SUTRA Post Processing Charts layer***, and make the ***SUTRA Mesh*** layer invisible. This produces a viewable plot of concentrations after time step 5, and steady-state velocities.

12. Because results for a few time steps will be viewed sequentially, it is advisable to use the same concentration contour levels each time. To pick these levels and adjust the contour plot, double click on it to bring up the *Contour Diagram dialog*.

13. Set the **Minimum** value to 1.0, the **Maximum** to 100.0, and the **Delta** to 1.0. Then click **OK**, and the plot redraws. Note the position of the leading edge and bulk of the plume for comparison with a later simulation time.

14. Now view results for **Time Step** 10, by again using **SUTRA Post Proc...** under menu **PIEs**, and re-select the d5 file created for this simulation. In the *Select SUTRA results to visualize window* toggle the **no** to **YES** for **Velocity** and **Concentration** in the line for **Time Step** 10, and click on **OK**.

15. The new plot appears. Reset the contour levels, as was done for time step 5, and note that the leading edge and bulk of the plume have progressed downstream.

16. The plot may be printed to a printer or file at this point, by selecting **File**, then **Print...**, and modifying the *Print dialog* that appears.

17. Repeat steps 14 through 16 to view results for time step 20, and then once again for time step 40. Save the project.

18. Finally, the progression of the plume may instead be viewed for all time steps at once. First, make all layers invisible except the ***SUTRA Post Processing Charts layer***, by clicking on the **Show: None** button at the top of the *Layer List window*.

19. Then, from the **PIEs** menu, select **SUTRA Post Proc...** and then reselect the d5 file (e.g. Areal-tr.d5).

20. In the *Select SUTRA results to visualize window*, click on the word **Concentration** found along the top of the window; this makes each **no** in the column into a **YES** and selects plots of all the available concentration results. Before leaving this window, click on the **Overlay Charts** box in the lower left corner to deselect this option, so that all of the concentration plots will appear separately. Then click **OK**.

21. Click **OK** again to allow the layer to be cleared when so prompted, and the new plots appear. If these appear small, then they may be zoomed into view by clicking on the zoom tool at the lower left corner of the window (the magnifying glass without '+' or '-' sign). Then click and drag a box around the group of small plots to increase their size. This type of plot can provide a good preview of the concentration results for all of the time steps selected for output in the SUTRA simulation.

HENRY SEAWATER INTRUSION PROBLEM WITH VARIABLE-DENSITY FLOW

This step-by-step example reproduces a classic simulation example for testing variable-density transport codes, that is presented as an example simulation in the SUTRA documentation (Voss(1984), section 6.5, page 196). It involves creating a perfectly rectangular cross-sectional model domain containing a Fishnet Mesh, and applying boundary conditions exactly along the two vertical edges of the domain. Along one edge, a total fluid inflow is specified and the Argus-SUTRA GUI is allowed to distribute the value of fluid sources along the edge. Along the other edge, the user creates a specified pressure condition in a single object that linearly increases with depth representing hydrostatic seawater. It is necessary that the user read the above-mentioned section of the SUTRA documentation for background on this problem before beginning.

1. Start Argus ONE. In the *SUTRA Interface Configuration window*, select a cross-sectional orientation with saturated flow, solute transport with variable-density fluid, using pressure, user-specified model thickness, and a fishnet mesh. Then click OK.

2. In the *SUTRA Non-Spatial Information dialog*, a number of changes to the initial default values are required to match the Henry problem.

On the **Headings** pane,set	**TITLE1** to Henry Problem from SUTRA Documentation	
On the **Modes** pane, select	**Transient groundwater flow**	
and	**Transient solute transport**	
On the **Temporal Controls** pane, set	**ITMAX**=100	**DELT**=60.
On the **Output** pane, set	**NPRINT**=50	**TMAX**=1.e9
On the **Fluid Properties** pane, set	**SIGMAW**=18.8571e-6	**VISC0**=1.e-3
On the **Gravity** pane, set	**GRAVY**=-9.8	

Then click **OK**.

3. In the *Argus* ONE *window*, select **Drawing Size...** found under **Special** along the top of the window and set size as follows:

Horizontal Extent: = 4
Vertical Extent: = 3
Horizontal Origin: = -1
Vertical Origin: = -1

4. The workspace shrinks because of the reduced drawing size. Resize it using the zoom tool and resize the window if necessary.

The next steps create the 2 by 1 model domain by first approximating the shape, then exporting it, editing to make it exact, and re-importing the result.

5. Activate the ***Fishnet Mesh Layout layer***.

6. Click on the closed contour drawing tool.

7. While holding down the shift key on the keyboard, place four vertices at about (0,0), (0,1), (2,1) and double click on the last vertex at (2,0). The shift key causes lines to be drawn horizontally and vertically.

8. This brings up the *Contour Information dialog*. Set **elements_in_x**=20 and **elements_in_y**=10. Click **OK**.

9. Export the superblock just created to a text file by selecting **Export Fishnet Mesh Layout...** under **File**, **Export**. Select the desired directory for saving and then **Save**.

10. Leaving Argus ONE operating in the background, start any available text editor and open the exported file, **Fishnet_Mesh_Layout.exp**. Change the five pairs of approximate (x, y) coordinates representing the four vertices of the closed contour to:

```
0  0
2  0
2  1
0  1
0  0
```

and then save the file.

11. Returning to the *Argus* ONE *window*, click the arrow tool and press the delete key to remove the approximate superblock.

12. Import the edited file by selecting **File**, **Import Fishnet Mesh Layout...**, **Text File**, and double click on the file **Fishnet_Mesh_Layout.exp**. This inserts a perfectly rectangular superblock into the layer. Check that it is indeed perfect by double clicking on it and inspect the values in the *Contour Information dialog* that appears.

13. Adjust the zoom if needed so that the block nearly fills the window. Save the project.

The next steps create a line source exactly along the left vertical edge of the domain representing inflow of fresh water to the cross section. The total inflow across this boundary (0.066) and the solute concentration of inflowing fresh-water (0.0) are specified.

14. Click on the superblock once or twice to cause the four vertices to be indicated by filled squares. Then click on the Lasso tool.

15. Holding down the mouse button, circle the left vertical side of the superblock. This highlights the two left vertices.

16. Copy this edge of the superblock by pressing **Ctrl+C** (control button and at the same time, letter C) or select **Edit**, **Copy**.

17. Activate the ***Sources of Fluid layer*** and then click in the workspace.

18. Paste in the copied object by pressing **Ctrl+V** (or select **Edit**, **Paste**). To see only what was just pasted, make the ***Fishnet Mesh Layout layer*** invisible for a moment.

19. Double click on the open contour just pasted to bring up its *Contour Information dialog*.

20. The parameter values from the superblock were also copied here and must be removed and replaced by appropriate values for this layer. The user may specify either a total source value for the open contour object, or a source per length of the object. Only one of these may have a value, and the other **must** be set to $N/A, meaning 'undefined'. Set:

    total_source = 0.066
    specific_source = $N/A
    concentration_of_source = 0.0

    and click **OK**. Save the project.

The following steps create a background value for the ***Specified Pressure layer*** that represents the pressure of seawater under hydrostatic conditions. This background value provides a pressure everywhere in the workspace. Sea level is set at elevation y=1 and pressure increases below this point. The density of sea water is given by the linear fluid density expression used by SUTRA, (1000. + 700. * 0.0357), gravity is 9.8, and depth is given by (1. - y).

21. Open the *Layers dialog* either by clicking the appropriate tool button along the top of the window or the **Layers...** button in the *Layer List window*.

22. In the upper window, click on the ***Specified Pressure layer*** to select it.

23. In the lower window, hold the mouse button down over the **Value** column of **specified_pressure** and select **Expression...**.

24. In the *Expression dialog* that appears, type the following equation:

    (1000. + 700. * 0.0357) * 9.8 * (1. - Y())

    (The Y() is an Argus ONE function and may be automatically inserted by clicking on **Mathematical** (under **Functions**) in the lower left list, and then clicking on **Y** in the lower right list.)

    Then click **OK**.

25. Click on **concentration** in the **Layer Parameters** (lower window), Then click in the **Value** column and select **Expression...**.

26. In the *Expression dialog*, set the value to 0.0357, and click **OK**.
    Click **Done** in the Layers dialog. Save the project.

27. Activate the ***Specified Pressure layer*** and move the cursor up and down in the workspace to observe the varying background pressure value just specified. (The value is shown in the lower left margin of the window.) Note that zero occurs along the top edge of the model domain and pressure increases with depth.

The next steps create a specified pressure boundary condition exactly along the right vertical edge of the superblock by copying the edge from the ***Fishnet Mesh Layout layer*** and modifying the object's values.

28. Activate the ***Fishnet Mesh Layout layer*** and click on the superblock to select it (making the vertices appear).

29. Using the lasso tool, circle the right vertical edge of the superblock (highlighting the two vertices on this edge). Copy this edge by typing **Ctrl-C**.

30. Again activate the ***Specified Pressure layer*** and click in the workspace. Then paste the object (**Ctrl+V**).

31. Double-click on the pasted object to bring up its *Contour Information dialog*.

32. The over-ridden default values (copied from the superblock and indicated by the rectangles around values) must be cleared so that this object can derive values from the background. Click on the **specified_pressure** line in the **Value** column to highlight the "20", and then delete this value.

33. Hit the **arrow down** key on the keyboard (or click on the next value in this column in the **concentration** line). A new value appears (which is the default value at the first vertex in the object of the background expression set in step 24). The actual value is not important, rather what is necessary is to clear the over-ridden values so that the object can receive its background default values.

34. Delete the second over-ridden value and hit **arrow up** to see the effect. The default parameter value entered in step 26 appears. Click **OK**. Save the project.

When this domain is meshed, nodes falling exactly along this object will be automatically assigned both specified pressure values varying with depth and seawater concentration to create the desired boundary condition. In the next steps, the initial default parameter values for various layers will be set to the values required for the Henry problem.

35. Bring up the *Layers dialog*.

36. Highlight the ***Porosity layer*** in the upper window. In the lower window, reset the initial default value of 0.1 to 0.35 by bringing up the *Expression dialog* for the ***porosity*** parameter.

37. Highlight the ***Permeability layer*** in the upper window. In the lower window, select ***maximum*** and set its default value to 1.020408E-9.

38. Select the ***minimum*** parameter and set its value to that of ***maximum***. To do this, in the *Expression dialog*, select **Permeability** from the lower left list, then double click **maximum** in the lower right list. the parameter **maximum** then appears in the upper expression window. Click OK to exit it.

39. Highlight the ***Dispersivity layer*** in the upper window and set all four dispersivity parameters to zero (0.0).

40. Click **Done** to exit the *Layers dialog*. Save the project.

This completes entry of information for the Henry problem. Next the mesh is created and numbering optimized.

41. Activate the ***SUTRA Mesh layer***.

42. Select **File**, **Import SUTRA Mesh...**, **Fishnet Mesh...**. This creates the Fishnet mesh which appears on the screen. (Compare this with the mesh shown on page 197 of the SUTRA documentation.)

43. Select **Special**, **Renumber...**, and then click on **Optimize BandWidth** and **OK** in the *Renumber dialog*.

44. Inspect the values assigned to nodes and elements in the mesh by double clicking the node or element of interest. Note the distribution of inflow (QIN) along the left-hand boundary nodes, where the upper and lower node are assigned half the inflow of the others along the boundary to correctly create a uniform inflow. Note also the distribution of specified pressure (PBC) along the right-hand boundary nodes is equivalent to the pressure a hydrostatic column of seawater.

45. Save the project.

Finally, the simulation is run, and results plotted.

46. Export and run SUTRA.

47. Using **SUTRA Post Proc...** in the **PIEs** menu, plot concentration and velocity for time step 100. To plot each 10% of seawater concentration, set **Minimum:** to 0.000357, **Maximum:** to 0.0357, and **Delta:** to 0.00357 in the *Contour Diagram dialog*. Then, in order to compare with the results found in the SUTRA documentation on page 201, plot each 25% by setting both **Minimum:** and **Delta:** to 0.008925. Save the project.

The user may wish to continue by experimenting with various aspects of this problem. For example, the mesh size may be halved in each direction, a low conductivity barrier or high conductivity channel may be added, or parameters such as dispersivity and diffusion coefficient may be changed and the simulation rerun and results visualized.

USER'S GUIDE

STARTING ARGUS ONE AND THE SUTRA INTERFACE

The pre-processing function of the PIE for SUTRA enhances the use of Argus ONE to automatically create the appropriate geospatial coverages or layers and install menus and dialogs specific to SUTRA. With the five files described previously (see "System Requirements and Installation" (page 5)) contained in the SUTRA directory located within the ArgusPIE directory, Argus ONE will automatically locate and load the PIE needed to run the interface for SUTRA at run time. It is recommended that the screen resolution be set to 800x600 or higher before starting Argus ONE.

Argus ONE may be started by double-clicking the Argus ONE icon or by double-clicking a ".mmb" project file (where it appears in any file manager window) saved previously in an Argus ONE session.

After starting Argus ONE, the SUTRA pre-processor is invoked by selecting the **New SUTRA Project** menu item from the **PIEs** menu.

SUTRA PROJECT INFORMATION DIALOG

The project information dialog allows the user to choose the desired type of SUTRA application (e.g. areal solute-transport model, or cross-sectional saturated-unsaturated energy-transport model, etc.) and to input required information for the SUTRA simulation that is not spatially-dependent (e.g. time step size, SUTRA output controls, etc.). This dialog also allows the user to edit this project information at any time from within the Argus ONE environment. Input of spatially-dependent information is done in geospatial information layers accessible from the standard Argus ONE window after exiting from the SUTRA project information dialog.

This dialog and the tabbed dialog that follows are generated by the USGS SUTRA PIE program, **SUTRA.DLL**.

TITLE WINDOW

The *Title Window* provides information about the current version of the SUTRA PIE and information about SUTRA and Argus ONE documentations. It is not possible to return to the *Title Window* after leaving it.

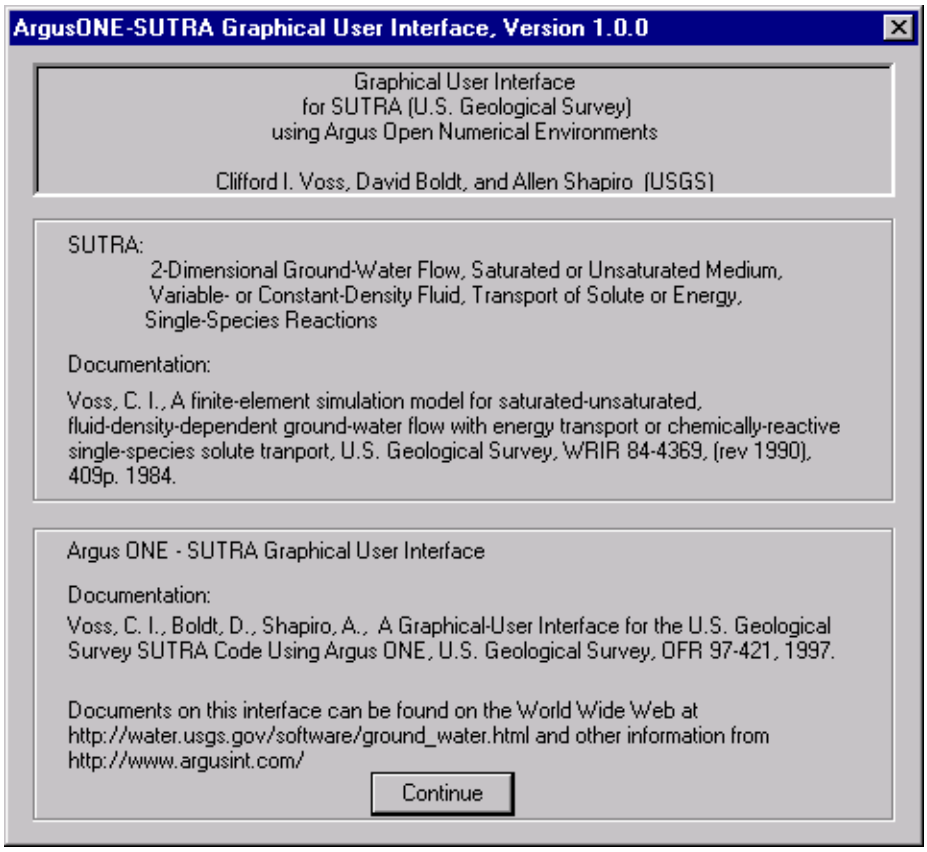


Figure 8. USGS-SUTRA PIE Title Window

The USGS-SUTRA PIE *Title Window* shown in Figure 8 appears on the screen, and when dismissed initiates the *SUTRA Interface-Configuration Window* described below.

Click **Continue** to proceed.

SUTRA INTERFACE-CONFIGURATION WINDOW

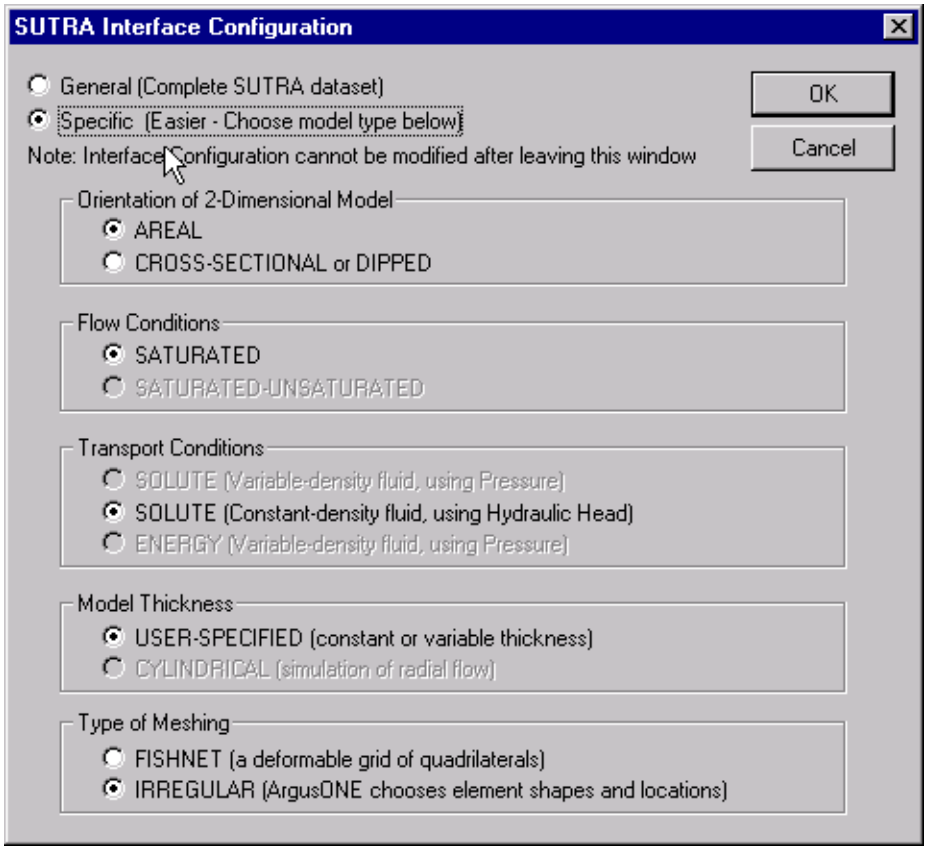


Figure 9. SUTRA Interface-Configuration Window

The *SUTRA Interface-Configuration window* (Figure 9) allows the user to choose the type of simulation problem to be solved with SUTRA, and sets up the GUI to include the appropriate geospatial information layers, and sets names of layers and parameters to be appropriate for the chosen problem type. For example, an areal problem will include an information layer called "hydraulic conductivity" and refer to "hydraulic head" whereas a cross-sectional problem with variable-density fluid will have a layer called "permeability" and refer to "pressure"; an unsaturated problem will include an information layer called "unsaturated properties" while a saturated-only problem will not have such an information layer.

The SUTRA code solves very general equations for flow and transport allowing much flexibility in problem type. This is reflected in the choices available in the *SUTRA Interface-Configuration window*. Previously (before distribution of the present interface) the user was required to input data in terms of pressure and permeability; now, input is simpler, particularly for problems with constant-density fluid. Internally, the SUTRA code still works with its original set of variables, but the Argus-SUTRA GUI translates these into more practical quantities for ease of use.

- Should the user prefer to work with the original SUTRA variables, and have access to all possible information layers for SUTRA at once, then the option **General** should be selected.

- Should the user prefer to configure the Argus-SUTRA GUI for a certain type of problem, then the option **Specific** should be selected. This makes available a set of choices to the user in the lower portion of the window, allowing the user to specify the problem type, as well as the type of meshing (see Figure 9).

Explanations of the choices follow:

Orientation of 2-Dimensional Model

AREAL
Perfectly horizontal model for constant-density solute transport under saturated conditions (e.g. used for typical contaminant plume problems or tracking of natural tracers).

CROSS-SECTIONAL OR DIPPED
Perfectly vertical cross-sectional model or non-horizontal model potentially allowing unsaturated conditions with variable-density solute or energy transport. The gravity vector components in this model are potentially non-zero (used for a variety of problem types depending on choices below).

Flow Conditions

SATURATED
Only saturated conditions allowed.

SATURATED-UNSATURATED
Unsaturated conditions may potentially occur in the model (used specifically for problems of unsaturated flow and transport).

Transport Conditions

SOLUTE (Variable-density fluid, using Pressure)
For transport problems where solute concentrations affect fluid density. (e.g. used for typical saline fluid intrusion problems or flow near salt domes).

SOLUTE (Constant-density fluid, using Hydraulic Head)
For transport problems where solute concentrations do not affect fluid density; (e.g. used for typical contaminant plume problems).

ENERGY (Variable-density fluid, using Pressure)
For transport problems where temperature changes are to be tracked in the subsurface and where temperature potentially affects fluid density; (e.g. used for hydrothermal convection problems, tracking seasonal surface water recharge, aquifer energy storage).

Model Thickness

USER-SPECIFIED
The user will specify the model thickness in a geospatial information layer. The thickness may be constant or variable.

CYLINDRICAL
Model thickness is automatically set such that a radial cross-sectional flow field is obtained. Thickness at each ***SUTRA Mesh*** node is automatically set to $2\pi X$, where X is the radius (X-coordinate, *must be positive*) of the node. Cylindrical thickness is used for vertical cross sectional simulation only.

Type of Meshing

FISHNET

The mesh consists of superblocks (large contiguous quadrilaterals) each subdivided into a specified number of quadrilateral finite elements connected four to each internal node. The mesh may be considered similar to a deformed finite-different grid. The mesh is created by the SUTRA PIE and not by Argus ONE. Detailed instructions may be found in the section describing the SUTRA Model layer, "Fishnet Mesh Layout" (page 56).

IRREGULAR
The mesh consists of irregularly-connected quadrilateral elements created by Argus ONE's quadrilateral meshing engine. For an example of an irregular mesh, see Figure 30 on page 61.

Once the user exits this window, the SUTRA-Argus ONE interface will have been configured, and the problem type cannot be changed without starting over with a **New SUTRA Project**.

Click **OK** to proceed to the *SUTRA Non-Spatial Information dialog*.

SUTRA NON-SPATIAL INFORMATION DIALOG

After exiting from the *SUTRA Interface-Configuration window*, or when **Edit Project Info** is selected from the Argus ONE **PIEs** menu, the *SUTRA Non-Spatial Information dialog* (Figure 10) appears. This dialog allows the user to enter and change values of parameters that have no spatial dependence and are required for simulation with SUTRA. Each tab is labeled according to the information that appears on the pane to which the tab is attached.

Clicking a tab brings the appropriate pane to the foreground for data entry. Tabs that are cut off by the right or left edge of the window may be scrolled into view by clicking the left or right arrow near the top-right of the window.

Most of the information to be entered in the *SUTRA Non-Spatial Information dialog* is required in SUTRA input data set "K1" (often referred to as the "d5" data set), and with a few exceptions, appears in the same order as normally found in any SUTRA input data set, as described in the SUTRA documentation (Voss, 1984). The 'initial time' from SUTRA input data set "K2" ("d55") also appears here. The SUTRA input dataset number and the SUTRA parameter name are given on each tabbed pane to allow the user to easily find detailed instructions for each particular data type in the documentation. Some data-entry boxes may be greyed out and inaccessible; these parameter values are fixed by the user's choice of problem type (on the *SUTRA Interface-Configuration window*, discussed just above) and need not be altered.

When all entry or modification of data in the *SUTRA Non-Spatial Information dialog* is complete click on **OK** to accept changes, or on **Cancel** to ignore any changes made; in either case, the user proceeds to the Argus ONE environment. If **Cancel** is selected for the creation of a new SUTRA project, then the creation of that new project is canceled.

The **OK** or **Cancel** button can be clicked at any time while editing project information for the SUTRA simulation; it is not necessary to access all of the panes or data-entry boxes before using the **OK** or **Cancel** buttons as these may be returned to at any later time.

The default values for all parameters set in the *SUTRA Non-Spatial Information dialog* are given in Table 1 and Table 2. The information required on each pane is described below.

Table 1: Default Values for SUTRA Non-Spatial Information dialog for General Case[1]

Variable	Default	Variable	Default	Variable	Default
SIMULA	SUTRA SOLUTE TRANSPORT	NUCYC	1	URHOW0	0
TITLE1	SUTRA Model created using Argus ONE	TSTART	0	DRWDU	0
TITLE2	USGS sample Interface	NPRINT	9999	VISC0	1
IUNSAT	0	KNODAL	0	COMPMA	0
ISSFLO	1	KELMNT	0	CS	0
ISSTRA	1	KINCID	0	SIGMAS	0
IREAD	1	KVEL	1	RHOS	2600.0
UP	0	KBUDG	1	ADSMOD	NONE
GNUP	0.1	ISTORE	9999	CHI1	0
GNUU	1.0	NOBCYC	0	CHI2	0
ITMAX	1	ITRMAX	1	PRODF0	0
DELT	1	RPMAX	0	PRODS0	0
TMAX	1.0	RUMAX	0	PRODF1	0
ITCYC	9999	COMPFL	0	PRODS1	0
DTMULT	1.0	CW	1	GRAVX	0
DTMAX	1	SIGMAW	0	GRAVY	0
NPCYC	1	RHOW0	1	NTOBS	calculated[2]

1. Values in MKS units (also known as SI units).
2. This value is not entered but is calculated by the GUI, which runs a pre-simulation of the time step logic.

NOTE: Default values are only initial suggestions, and must be checked and reset, if needed, by the user to appropriate values for the user's project.

Table 2: Default Values[1] for SUTRA Non-Spatial Information dialog for Special Cases[2]

Solute transport with variable density		Solute transport with constant density		Thermal Energy transport		Saturated-unsaturated flow	
Variable	**Default**	**Variable**	**Default**	**Variable**	**Default**	**Variable**	**Default**
SIGMAW	1.0E-9	SIGMA	1.0E-9	CW	4182.0	IUNSAT	1
RHOW0	1000.0	RHOW0	1	SIGMAW	0	ISSFLO	0
DRWDU	700.0			RHOW0	1000.0	ISSTRA	0
VISC0	0.001			URHOW0	20	ITRMAX	2
GRAVY	-9.81			DRWDU	-0.375	**Saturated-only flow**	
				CS	840.0		
				SIGMAS	3.5	**Variable**	**Default**
				GRAVY	-9.81	IUNSAT	0

1. Values in MKS units (also known as SI units).
2. Only values that differ from the defaults in Table 1 are listed.

NOTE: Default values are only initial suggestions, and must be checked and reset, if needed, by the user to appropriate values for the user's project.

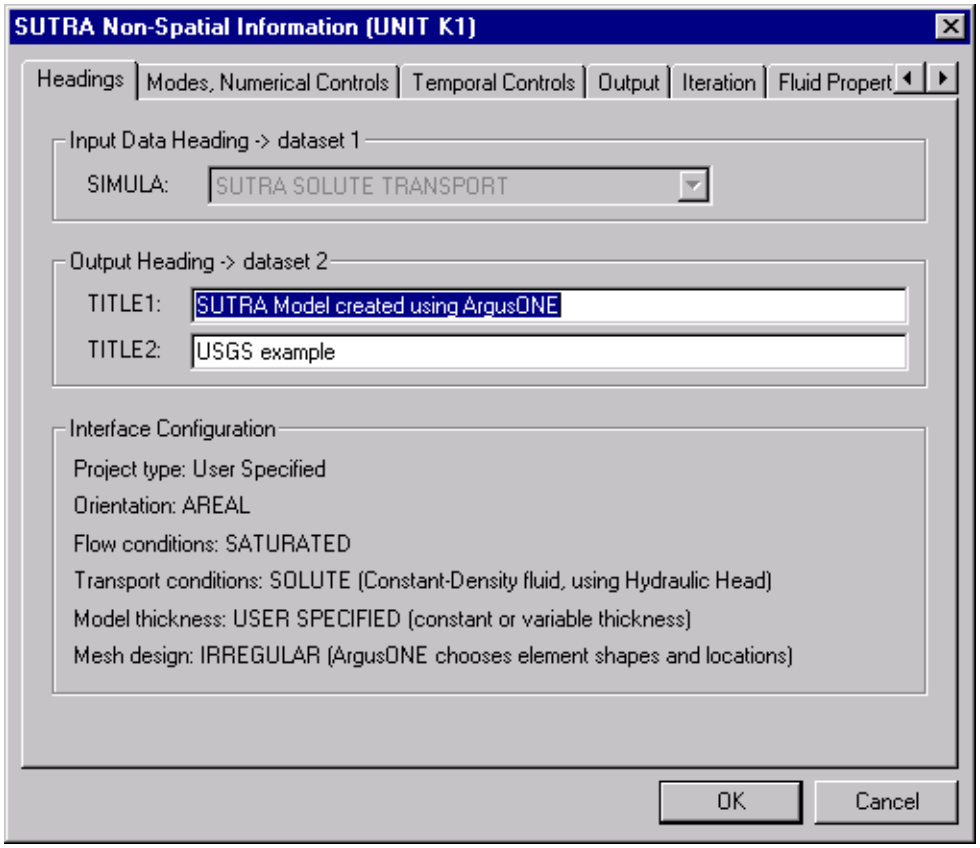


Figure 10. SUTRA Non-Spatial Information dialog; Headings

Headings

Figure 10 shows the contents of the dialog window containing heading information.

Input Data Heading
: Clicking and holding down the arrow provides a choice of solute or energy transport simulation.

Output Heading
: Two lines of title may be specified here that will appear on the SUTRA "K3" output file (often referred to as the "d6" file because of the Fortran file unit number usually used for this file).

Interface Configuration
: The particulars of the choices made by the user in the *SUTRA Interface-Configuration window* are shown here, but they cannot be altered.

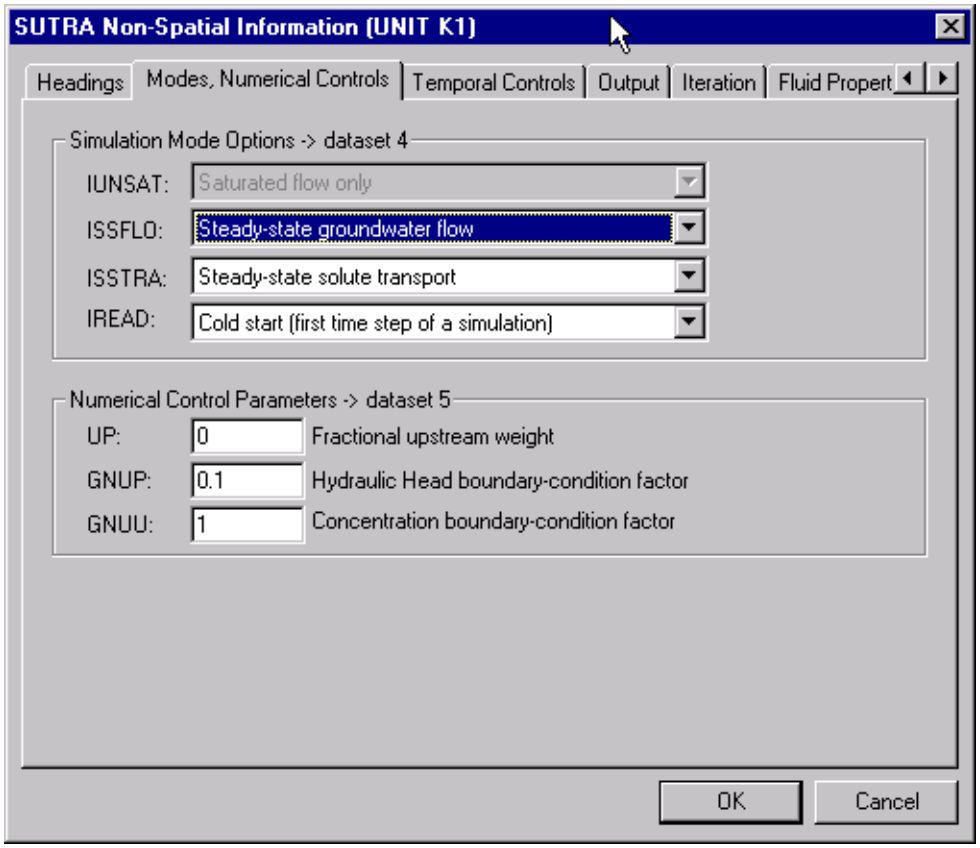


Figure 11. SUTRA Non-Spatial Information dialog; Modes, Numerical Controls

Modes, Numerical Controls

Figure 11 shows the contents of the dialog window containing simulation modes and numerical controls.

Simulation Mode Options
: Clicking and holding down the arrow allows the choice of saturated or saturated-unsaturated flow, steady-state or transient fluid flow, steady-state or transient transport, and cold or warm start.

Numerical Control Parameters
: Numerical entry of upstream weight, and boundary-condition factors.

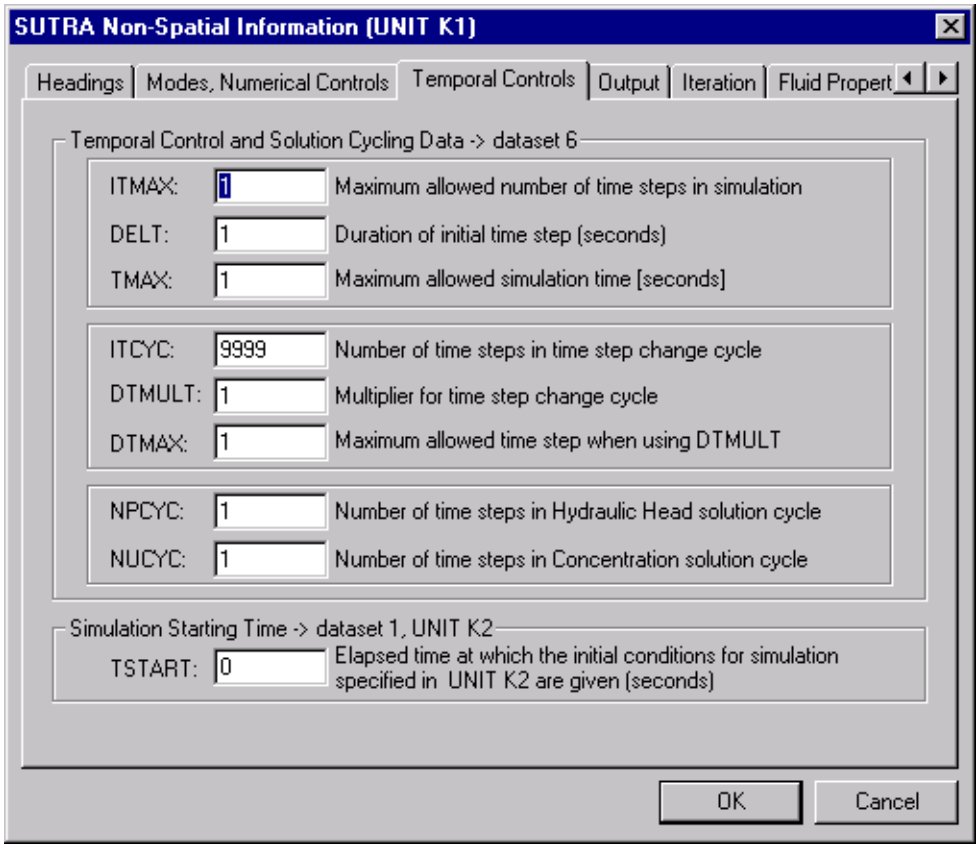


Figure 12. SUTRA Non-Spatial Information dialog; Temporal Controls

Temporal Controls

Figure 12 shows the contents of the dialog window containing time information.

Temporal Control and Solution Cycling Data
: Numerical entry of time steps in simulation, time-step size, maximum time-step size, time-step change parameters, and solution cycling parameters.

Simulation Starting Time
: Numerical entry of simulation clock time at start of simulation. This value is written by Argus ONE to the SUTRA input data set "K2" (often referred to as the "d55" data set).

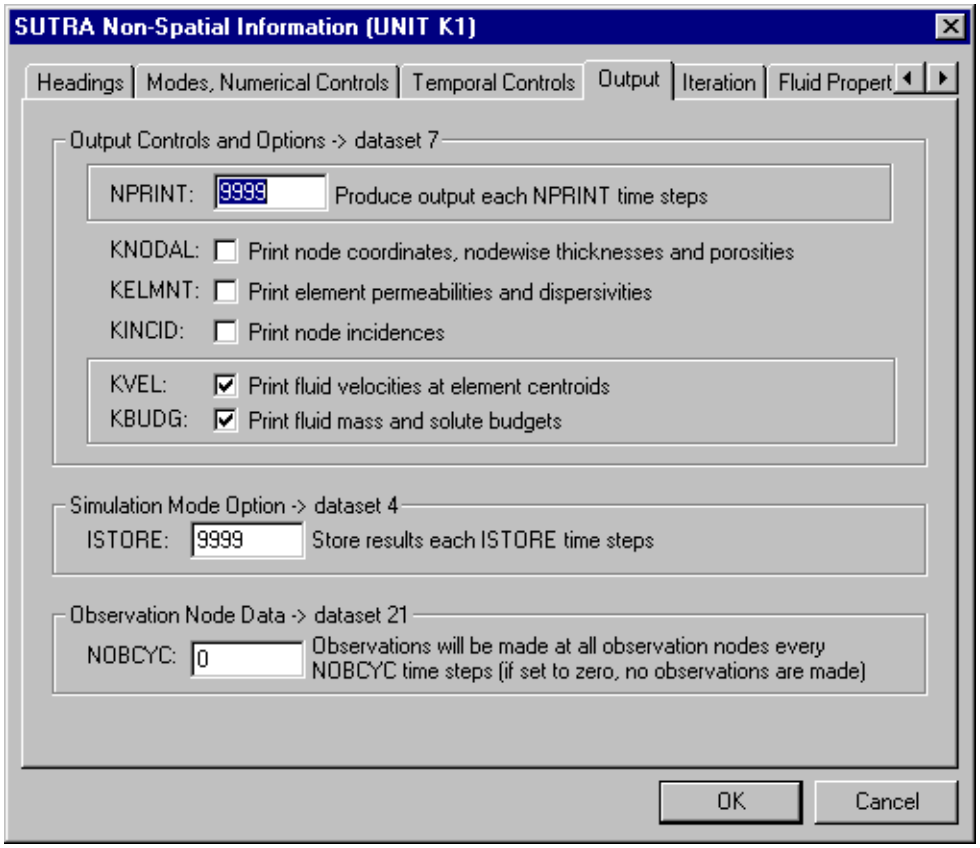


Figure 13. SUTRA Non-Spatial Information dialog; Output

Output

Figure 13 shows the contents of the dialog window containing output controls.

Output Controls and Options
: Numerical entry of output cycle for simulation results. Also, check entry (click on box) to toggle various types of output from SUTRA.

Simulation Mode Option
: Numerical entry of storage cycle for SUTRA output to be used as initial conditions in simulation restart.

Observation Node Data
: Numerical entry of observation cycle for observation nodes.

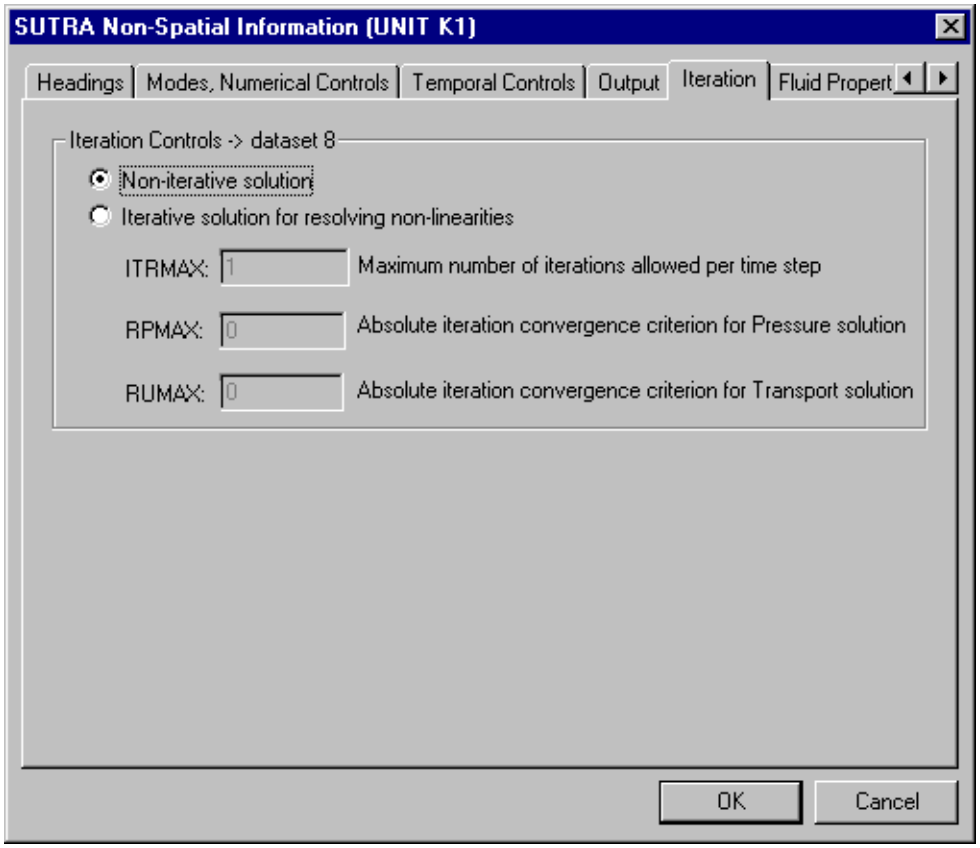


Figure 14. SUTRA Non-Spatial Information dialog; Iteration

Iteration

Figure 14 shows the contents of the dialog window containing iteration controls.

Iteration Controls

Click on the radio button (circle) next to desired mode: Non-iterative or Iterative. If Iterative is selected, then numerical entry of iteration parameters is required just below.

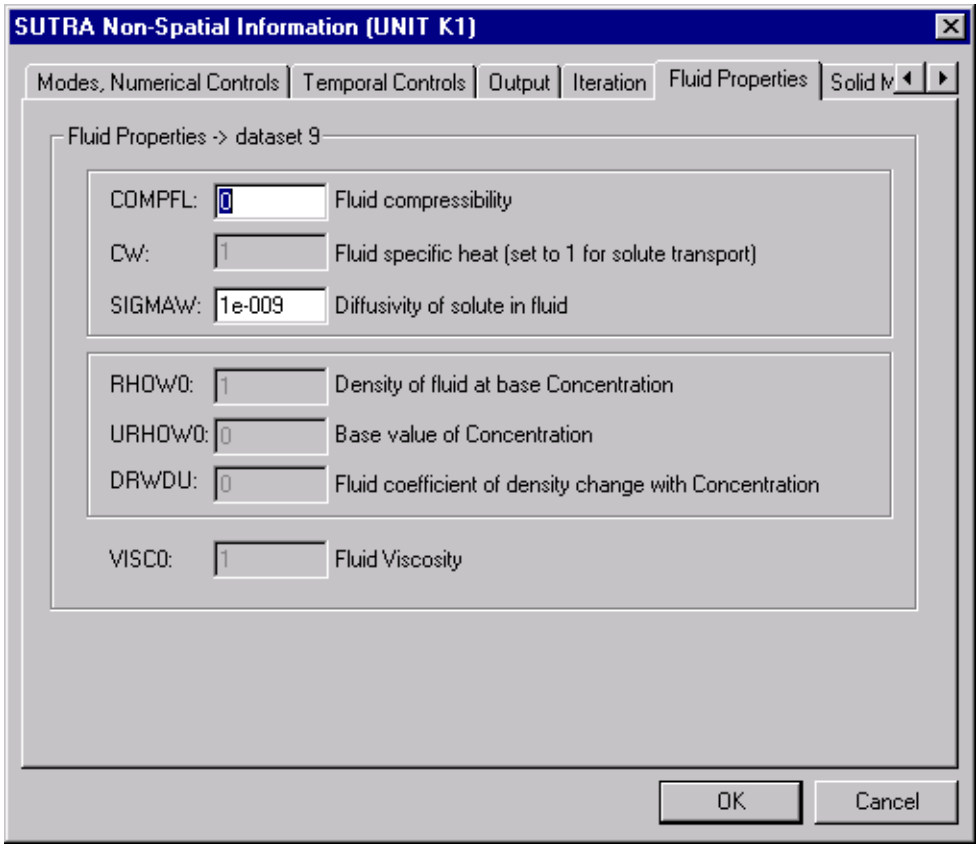


Figure 15. SUTRA Non-Spatial Information dialog; Fluid Properties

Fluid Properties

Figure 15 shows the contents of the dialog window containing fluid information.

Fluid Properties
: Numerical entry of fluid properties and parameters describing linear density-dependence of fluid on concentration or temperature.

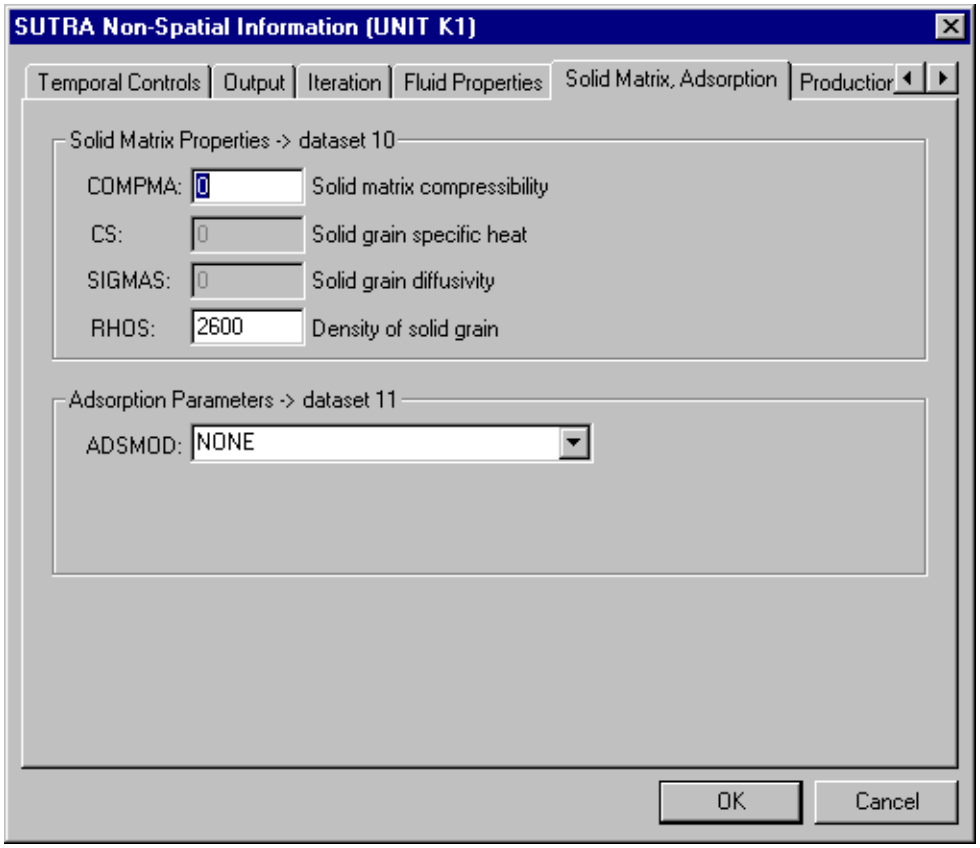


Figure 16. SUTRA Non-Spatial Information dialog; Solid Matrix, Adsorption

Solid Matrix, Adsorption

Figure 16 shows the contents of the dialog window containing aquifer material properties.

Solid Matrix Properties
: Numerical entry of solid matrix properties.

Adsorption Parameters
: Clicking and holding down the arrow allows the choice of NONE (no sorption) or available sorption isotherms. For LINEAR, one numerical data entry is required, for LANGMUIR and FREUNDLICH, two numerical data entries are required.

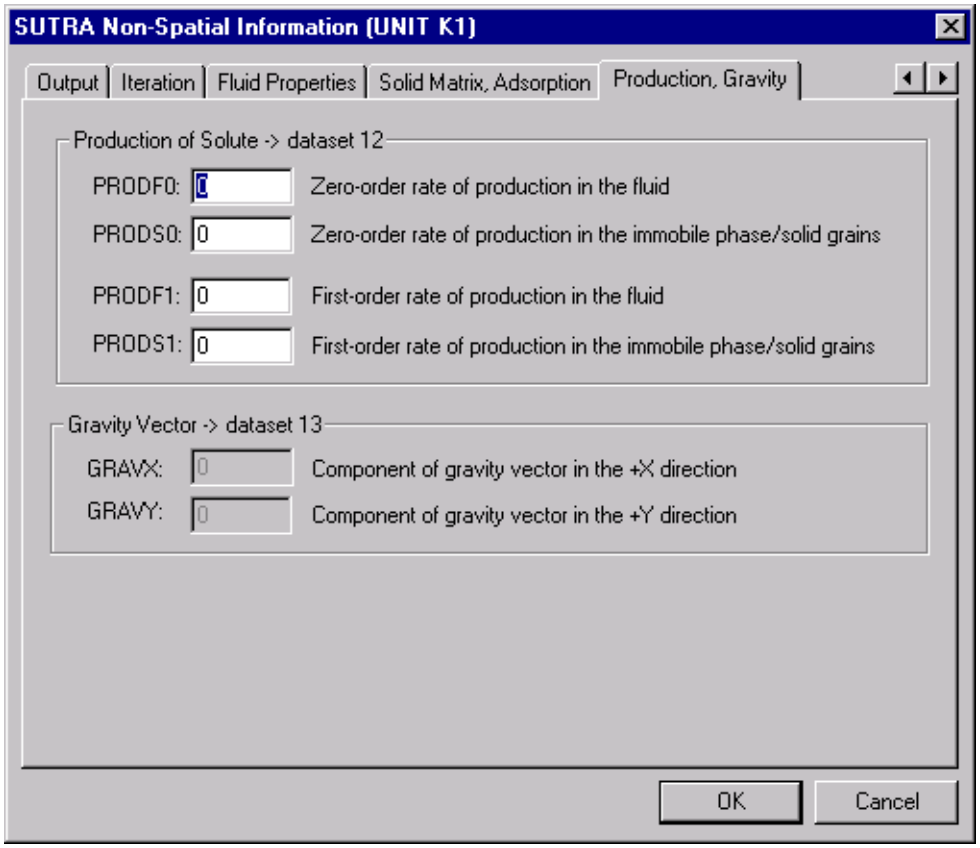


Figure 17. SUTRA Non-Spatial Information dialog; Production, Gravity

Production, Gravity

Figure 17 shows the contents of the dialog window containing solute production and gravity vector information.

Production of Energy or Solute
: Numerical entry of zero-order and first-order production rates.

Gravity Vector
: Numerical entry of X and Y components of the gravity vector.

THE ARGUS ONE WINDOW AND ARGUS ONE LAYERS

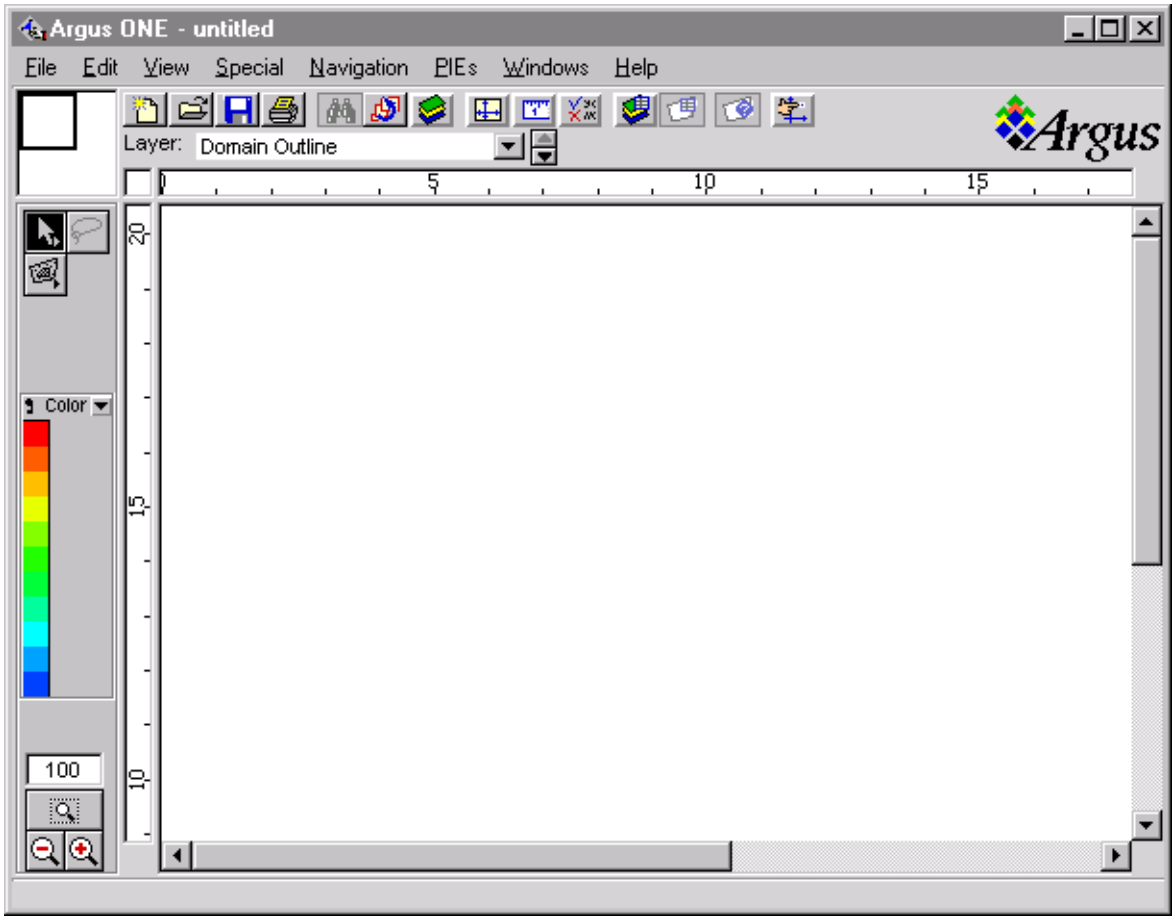


Figure 18. Argus ONE Window

Following exit from the *SUTRA Non-Spatial Information dialog*, described previously, the standard *Argus ONE window* appears (Figure 18). This window contains many controls for the user environment, and these are all described in the Argus ONE documentation. Only controls that are specifically related to use of the interface for SUTRA simulations are described in this document.

At this point there will actually be two overlaid windows, the original "Untitled" window, and a new "Untitled1" window in which you will want to work. The user may wish to close or iconify the original "Untitled" window so as not to confuse the two.

For convenience, the window may be resized to full screen by clicking the button containing an upward pointing triangle in the upper-right corner of the window.

Prior to creating or importing information in geospatial information layers, it is recommended that the user decide on and possibly specify the drawing size of the work space, as well as the scale and units of the modeling project. These project parameters can also be changed automatically when importing information that extends beyond the specified ranges.

This information can be entered from the **Drawing Size...** and **Scale and Units...** commands in the **Special** pull-down menu found along the top of the *Argus ONE window*. Note that the view of the workspace may be exaggerated or reduced in either direction by selecting **Non-uniform:** in the *Scale and Units dialog*. This is often required for cross-sectional models. Note that this also affects the shape of elements generated in an irregular mesh.

The Argus-SUTRA GUI assumes that all data are referenced to the same reference elevation or datum. For areal problems, the datum is arbitrary. For cross-sectional problems, the datum is given by the coordinate system selected by the user in the **Scale and Units...** commands in the **Special** pull-down menu found along the top of the *Argus ONE window*.

The Argus User's Guide provides information on setting attributes in the dialog boxes that appear when these commands are used.

SAVING AND RETRIEVING ARGUS ONE-SUTRA PROJECTS

The non-spatial information for the SUTRA simulations as well as all information entered into the geospatial information layers in Argus ONE is saved by using the **Save** or **Save As...** command in the **File** menu found along the top of the *Argus ONE window*.

The default extension for the Argus ONE saved project file names is ".mmb".

Starting Argus ONE and opening an existing project file containing SUTRA simulation information (by selecting **Open** in the **File** menu) will retrieve all of the non-spatial project information into the *SUTRA Non-Spatial Information dialog* and all of the geospatial information into Argus ONE layers that was entered previously, returning the user to the same state as when the project was saved.

Saving the project after exiting the *SUTRA Non-Spatial Information dialog* and saving regularly during input or edit of geospatial information is a good practice that will protect the user from the need to re-enter data in case of user errors or system problems.

LAYER LIST WINDOW

The *Layer List window*, which displays a list of available GIS coverages (layers) for the chosen type of SUTRA simulation, may be displayed by clicking on the **Layers...** button along the top of the window. This window is referred to in the Argus documentation as the "Layer Floater".

It may be resized to show all of the available layers by dragging and stretching the window from one of the lower corners with the mouse. The window may be moved to cover the unneeded grey area that may appear (depending on the size of Argus work area and on whether screen resolution is high enough) on a side of the *Argus ONE window* by holding down the mouse button on the top bar and dragging it to this location.

The *Layer List window* organizes the available coverages into groups or sub-lists including: SUTRA Model layers, various Geospatial Information layers, and Map or Point Data layers. These are described in detail in the sub-sections of "SUTRA Model Layers" (page 53), "Geospatial Information Layers" (page 67) and "Map or Point Data Layers" (page 80). The *Layer List window* also shows the interpretation type (described in "Interpretation of Contours and Point Data" (page 45)).

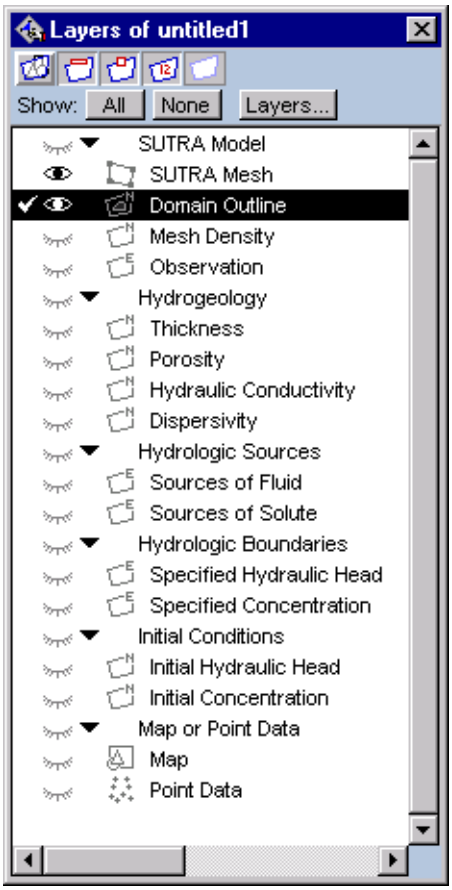


Figure 19. Layer List Window, expanded

Opening the list allows the user to see all available layer names within any group. To open or close groups, click on the triangle to the left of the group name. To close an expanded list, click again on the same triangle.

A view of the *Layer List window* (for the user-selected case of "cross-sectional saturated solute transport using pressure" with an "irregular mesh") with all groups expanded is shown in Figure 19.

VISIBILITY OF LAYERS

The Argus ONE layers or coverages are all superimposed on the *Argus ONE window* and may be viewed by the user all at once, singly or in selected groups. Note that nothing will appear on the screen when a layer is made visible unless there is an object (point or contour) in that layer.

The visibility of each layer may be controlled in any of three ways.

- (Easiest Method) Clicking on the open or shut eye in the *Layer List window* next to a particular layer toggles its visibility on (open eye) and off (closed eye). Toggling the eye next to a category of layers controls the visibility of all layers in that category at once.

- **Show&Hide** or **Hide Others** in the **View Menu** found along the top of the *Argus ONE window* (see Figure 18) may be used to control visibility.

- Visibility may also be controlled from the *Layers dialog* (which may be displayed by clicking the **Layers** button in the *Layer List window*, and closed by clicking **Done** in this dialog) by toggling the appropriate eyes open or closed

Figure 19 shows a *Layer List window* with ***SUTRA Mesh***, ***Domain Outline***, and ***Hydraulic Conductivity*** visible, and the other layers invisible.

ACTIVE AND INACTIVE LAYERS

Only the active layer is available for input and editing of graphical information; this layer will appear to be on top of any other layers in the working area of the *Argus ONE window*. All other layers will appear in the order shown in the *Layer List window*, from top to bottom. Any layer can be made active in one of three ways.

- (Easiest method) Clicking to the left of the eye next to a layer name in the *Layer List window* puts a check mark there indicating that this is now the active layer. Clicking next to an eye for another layer, makes this layer active instead. Clicking next to a closed eye (invisible layer) makes the layer visible and active.

- A layer may be made active by selecting it from the list of visible layers contained in the pull-down list below the buttons along the top of the *Argus ONE window*. This may be done by clicking the up or down arrow to the right of this pull-down list, or by selecting from the pull-down list itself by holding down the mouse button on the list and sliding to the layer which the user wishes to be active. Note that only visible layers are available here for selection.

- The active layer may be selected from the *Layers dialog* by clicking to the left of the eye next to the desired layer name to move the check mark to this location. The *Layers dialog* may be displayed by clicking the **Layers** button in the *Layer List window*, and closed by clicking **Done** in this dialog.

Figure 19 shows a *Layer List window* with the ***Domain Outline*** layer active.

LAYER TYPES

Argus ONE offers a GIS environment where two-dimensional coverages or layers of spatial information are superimposed. Note that the term "layer" in the context of Argus ONE does not have the same meaning as a geologic layer or stratum. Rather, layer refers to a two-dimensional spatial distribution of a particular kind of information which the user may interact with and view by itself or together with other layers as described in section "Visibility of Layers" (page 42). The information contained in each layer of Argus ONE can be generated using drawing tools that are part of Argus ONE, or the spatial information can be imported from a variety of sources including **TEXT** files, **DXF** files and ARC/Info™ or ArcView™ **Shape** files. Additional information about importing geospatial information into Argus ONE layers can be found in Argus ONE User's Guide and the World-Wide Web site for Argus Interware **http://www.argusint.com/**.

The Argus ONE layers contain spatially-distributed information and the information contained in any one layer may be dependent on or a function of information in one or more other layers. For example, the value of a spatial distribution in one layer may be set equal to the sum of the values of distributions contained in two other layers. Much more complex relationships are possible. The possibility of user specification of

such interdependence is the distinguishing characteristic of GIS technology. In the Argus ONE user environment, dependence of the information in one layer on another is called "linking". See "Adding and Linking New Layers" (page 96).

Before running a SUTRA simulation, appropriate information must be entered into various layers to describe the physical geologic and hydrologic setting for the ground-water flow and transport, as well as the numerical discretization of the modeled domain. The particular layers available for data input within the Argus-SUTRA GUI are described in some detail in the sections: "SUTRA Model Layers" (page 53), "Geospatial Information Layers" (page 67) and "Map or Point Data Layers" (page 80). A general description of Argus ONE layer types follows.

Argus ONE supports several types of two-dimensional coverages or layers; these layer types are referred to as quadrilateral mesh, triangular mesh, finite-difference grid, density, map, domain, data and information layers. Any number of most layer types can be created. SUTRA is based on the finite-element method using quadrilateral elements, thus, the triangular mesh and finite-difference grid layers are not described here.

A ***domain layer*** type specifies the areal extent of the modeled domain, and contains information on the desired density of the mesh within the modeled domain. The nominal size of mesh elements is controlled by both the value assigned to ***domain layer*** objects, and to the spacing of vertices in the objects. The ***domain layer*** is used exclusively by the discretization layers such as the finite-element quadrilateral mesh layer when the mesh for the numerical simulation is created.

A ***quadrilateral finite-element mesh layer*** type allows a finite-element mesh containing quadrilateral elements to be generated, edited, imported, and exported. The density (or size) of elements in any part of the modeled domain is based on information recorded in the domain layer. Alternatively, the ***mesh layer*** can take advantage of a special type of information layer, referred to as a ***Density layer*** type which contains the desired spatial distribution of element size, to increase or decrease the density of quadrilateral elements in selected regions of the modeled domain. Locations of nodes can also be edited manually. The mesh contains information at each node and element that is linked to the various information layers. The meshing may be changed at any time and in any way desired without affecting the underlying geospatial information, and the new node and element information will be automatically recalculated as needed.

An ***information layer*** type contains the distribution of geospatial information in the form of closed contours (polygons), open contours, and point contours all referred to in the Argus ONE environment as "objects". The appearance of an ***information layer*** is similar to a topographic map with contours, and the information may extend beyond the domain boundary of the model specified in the domain layer. The contour information is interpreted by Argus ONE to define the spatial variability of the information in the layer, and a number of interpretation methods are available within Argus ONE. The information in an ***information layer*** may be linked to the nodes and elements in a mesh layer and eventually may be exported in a formatted manner to create the input files required by SUTRA. Geospatial information in ***information layers*** is completely independent of the finite-element mesh. ***Information layers*** are the most general layer type, and the majority of layers used in generating a SUTRA model are of this type.

A ***map layer*** type provides a visual base for other geospatial information layers; information in ***map layers*** cannot be linked to other layers for use in the simulation. ***Map layers*** also can be used to visualize point data from Argus ONE Data layers using a variety of visualization tools provided in Argus ONE. The post-processing visualization of pressure, saturation, velocity vectors, and concentration or temperature for SUTRA simulations (provided by the SUTRA PIE) is generated in a ***map layer*** type. In short, ***map layers*** are not used by the model, but are only for visualization purposes.

A ***data layer*** type contains point data (only) which can be linked to the finite-element mesh and visualized in map layers. The project geospatial data is most often imported into this layer type. Gridded or meshed information can be extracted from other layers into a set of points in a ***data layer***.

Additional information about the attributes of these types of layers and linking information among layers, as well as the methods of importing and exporting information to layers is found in the Argus ONE User's Guide. The SUTRA-Argus ONE GUI is already set up with appropriate links among layers, and a user will not need to modify this setup unless the user wishes to add new layers to store additional types of geospatial information, and the advanced user who makes changes or additions to the SUTRA code may also have to make modifications to the GUI. See section "Adding or Changing Layers" (page 52).

LAYERS AND THEIR PARAMETERS

Within each group shown in Figure 19 on page 42, there are a number of layers, each representing one kind of information belonging to that group. Each such layer contains one or more **parameters**. It is the parameters in any layer that actually contain the information in the GIS coverages (spatial distributions of values); each layer parameter contains one coverage. All SUTRA-Argus ONE GUI layers and their parameters are given in Table 3.

Each parameter has a default value for the entire workspace that may either be a constant or an expression. Initial parameter values, when the Argus-SUTRA GUI is started, are constants and are given in Table 6, "Default Background Values for Layer Parameters" (page 70). Additionally, parameters may vary among objects in the layer. When defining a new spatial object (closed, open or point contour) in one of these layers, all of the parameters associated with the layer must be defined for the object in the *Contour Information window* that appears. Thus, parameter values in a layer may vary among the objects defined for the layer and throughout the workspace based on some combination of both object parameter values and the parameters' background values. The methods by which object and background parameter values are interpreted are described in the following section, "Interpretation of Contours and Point Data" (page 45).

Some trade-offs in designing the Argus-SUTRA GUI have been made in keeping the number of layers low, implying that some spatially varying parameters must change together. If a simulation demands that parameters within a layer vary independently, then these would need to be broken out into separate layers and these new layers linked to the ***SUTRA Mesh*** layer.

> The Argus-SUTRA GUI uses the convention that layer names are "Capitalized" and parameter names are all "lower case". Accordingly the naming convention is "Layer.parameter". If, for example, ***Permeability*** is a layer with three parameters, maximum, minimum, and angle_of_max_to_x_axis, then the full names of the layer's parameters are: ***Permeability.maximum***, ***Permeability.minimum***, and ***Permeability.angle_of_max_to_x_axis***.

INTERPRETATION OF CONTOURS AND POINT DATA

The calculation of continually varying parameters between points, lines and polygons representing equal-value contours is referred to in the Argus ONE environment as *interpretation*. Contour and point data information in any information-type layer define the spatial variability of the information throughout a layer, according to one of a number of available interpretation methods. These methods do not always allow the user to directly obtain desired spatial distributions. Thus, the user should experiment with these interpretation methods to obtain the best possible results for any given spatially-varying parameter.

> The interpretation method is selected from a pull-down list found at the bottom of the *Layers dialog* for the layer highlighted in this window. This window may be displayed by clicking the **Layers** button in the *Layer List window*, and closed by clicking **Done**.

The following interpretation schemes are available.

Table 3: Argus-SUTRA GUI Layers and Parameters

Group	Layer	Parameter Name[1]	Interpretation
SUTRA Model	SUTRA Mesh	● NREG	None
		● THICK	
		● POR	
		▲ LREG	
		▲ PMAX	
		▲ PMIN	
		▲ ANGLEX	
		▲ ALMAX	
		▲ ALMIN	
		▲ ATMAX	
		▲ ATMIN	
		● QIN	
		● UIN	
		● time_dependent_fluid_sources	
		● QUIN	
		● time_dependent_energy_or_solute_sources	
		● PBC	
		● pUBC	
		● time_dependent_specified_head_or_pressure	
		● UBC	
		● time_dependent_specified_concentration_or_ temperature	
		● PVEC	
		● UVEC	
		● INOB	
	Fishnet Mesh	elements_in_x	Nearest
		elements_in_y	
	Domain Outline	element_size	Nearest
	Mesh Density	element_size	Nearest
	Observation	is_observed	Exact
Hydrogeology	Thickness	thickness	Nearest
	Porosity	porosity	Nearest
	Hydraulic Conductivity or Permeability	maximum	Nearest
		minimum	
		angle_of_max_to_x_axis	
	Dispersivity	longdisp_in_max_permdir	Nearest
		longdisp_in_min_permdir	
		transdisp_in_max_permdir	
		transdisp_in_min_permdir	
	Unsaturated Properties	region	Exact

Table 3: Argus-SUTRA GUI Layers and Parameters

Group	Layer	Parameter Name[1]	Interpretation
Hydrologic Sources	Sources of Fluid	total_source	Exact
		specific_source	
		concentration_of_source or temperature_of_source	
		time_dependence	
		RESULTANT_FLUID_SOURCE[2]	
		QINUIN	
	Sources of Solute or Energy	total_source	Exact
		specific_source	
		time_dependence	
		RESULTANT_SOLUTE_SOURCE[2] or RESULTANT_ENERGY_SOURCE	
Hydrologic Boundaries	Specified Hydraulic Head or Pressure	specified_hydraulic_head or specified_pressure	Exact
		concentration or temperature	
		time_dependence	
	Specified Concentration or Temperature	specified_concentration	Exact
		time_dependence	
Initial Conditions	Initial Hydraulic Head or Pressure	initial_hydraulic_head or initial_pressure	Nearest
	Initial Concentration or Temperature	initial_concentration or initial_temperature	Nearest
Map or Point data	Map	map (not available for editing)	None
	Point Data	value1	"Default"

1. A circle ● denotes a nodewise parameter, while a triangle ▲ denotes an elementwise parameter. Only ***SUTRA Mesh*** layer parameters have this distinction.
2. Uppercase parameters may not be modified by the user.

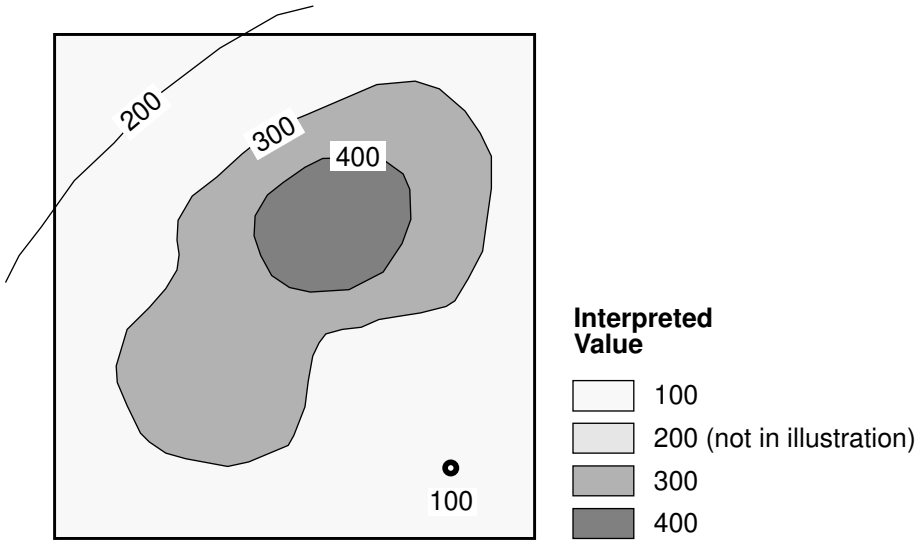


Figure 20. Exact Contour Method

The **Exact Contour method** works well with closed contours, and results in nested-plateau-type distributions of the parameters. In Figure 20 the area inside the 400 unit contour is interpreted to be 400, the area between the 300 and 400 unit contours is interpreted to be 300, and the area outside the 300 unit contour is interpreted to be the layer default value (0, set in *Layers dialog*). Open contours (such as the 200 unit contour) and points (such as the 100 unit point) are interpreted to have no effect on adjacent areas, and only affect nodes exactly beneath the contour or point. (Note that to force nodes to be exactly along an open contour or a point, these contours must be copied to the ***Domain Outline*** layer and the mesh regenerated.)

Typically, this method is used with 'zoned information' in which closed areas of constant value are defined. For example, when parameter estimation is used for ground-water models, aquifer properties such as permeability are often specified in zones. The default value of the layer is assigned to the region outside of closed contours.

An important application of this method is to define locations of specified heads and fluid sources, such as along streams or other surface-water bodies. Note that this is the default interpretation method for the groups ***Hydrologic Sources*** and ***Hydrologic Boundaries*** (see Table 3).

If the user plans to place point or contour objects in a layer to which other layers are linked, the interpretation method listed at the bottom of the *Layers dialog* must be set to **Exact Contour method**. Any other interpretation method will cause the objects to override the values from the linked layer everywhere in the layer.

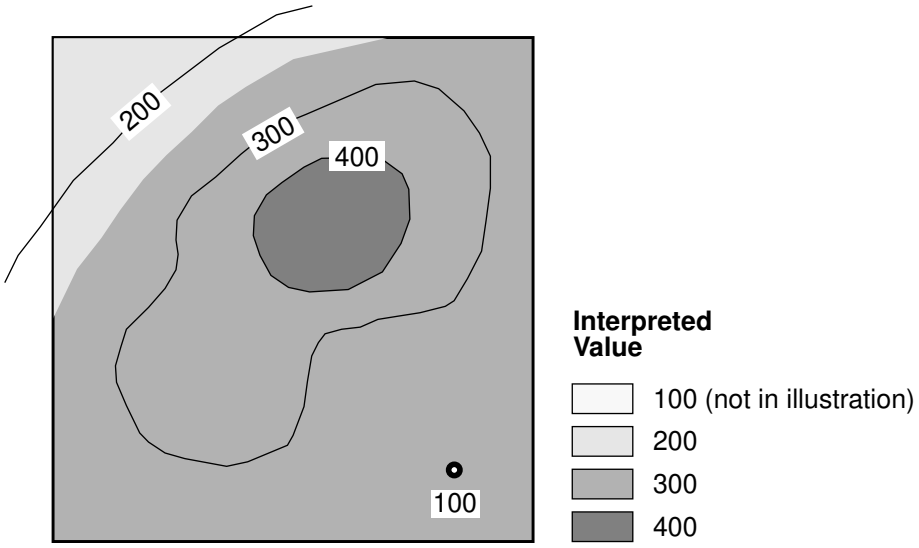


Figure 21. Nearest Contour Method

The **Nearest Contour method** is used with open and closed contours to define a spatial parameter distribution in any layer; it acts like the **Exact Contour method** for nested closed contours, but additionally gives steps in the parameter value halfway between other open and closed contours. This is the default scheme for most of the layers in the ***Hydrogeology group*** in the Argus-SUTRA GUI (see Table 3). In Figure 21 the area inside the 400 unit contour is interpreted to be 400, and the area between the 300 and 400 unit contours is interpreted to be 300. In the area between the 300 and 200 unit contours the value is interpreted to change from 300 to 200 along a line halfway between the contours, and the area outside of any contours is interpreted to be continue to be the nearest contour's value. Points do not affect adjacent areas, and must be *directly* over a node to affect its interpreted value.

Note that a single open or closed contour in a layer will set a value for the entire layer. Note also that the layer default value is ignored.

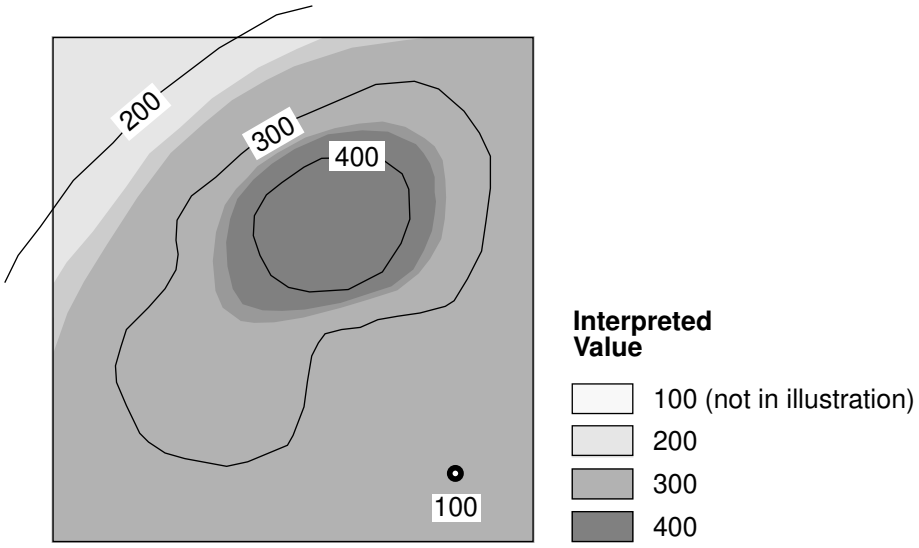


Figure 22. Interpolation Method

The **Interpolation Contour method** is usually used with scattered data points to create a smooth distribution and may also be used for interpreting between contours. When used with contours it will interpolate between the vertices (points) defining the contours. When there are few vertices, results are not always smooth. Note that the layer default value is ignored by this method.

This method is typically used with data known only at discrete locations that describe a property that varies continuously in space. An example would be top and bottom elevations of geologic formations known at well sites.

This method also is useful for data represented by open and closed contours that describe value topography. Examples are elevation or rainfall maps. The method produces an interpretation in which values change steeply at a point midway between contours (see Figure 22) which are based on many vertices (similar to results of the Nearest Contour Method but that changes halfway between all nested contours), and which may oscillate somewhat for contours based on few vertices.

These Argus ONE methods are further described in the Argus ONE Users Guide. Advanced users may write their own interpolation method as an Argus ONE PIE so that it appears in the Argus ONE menus.

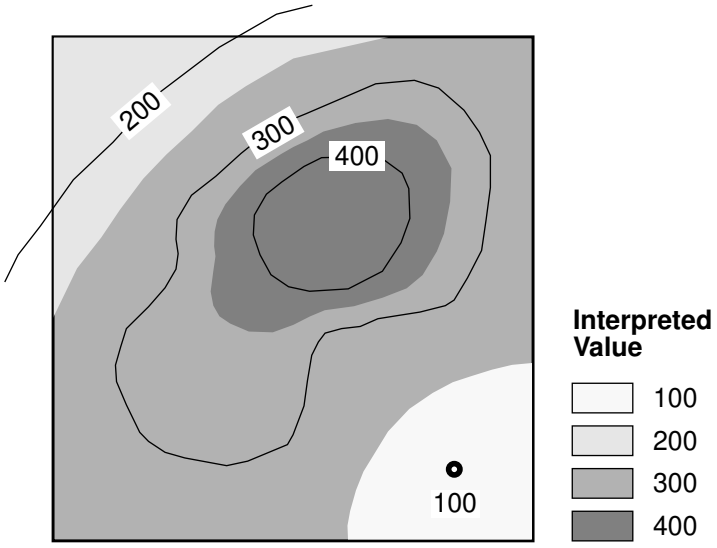


Figure 23. NN2D Contour Method

The **NN2D Contour method** is a non-smooth, nearest neighbor search that can be used with any kind of contour, point, open or closed. The value of the interpolation jumps abruptly at a distance midway between any contour. Figure 23 shows the effect as being like a very sharp edged variety of the **Interpolation Contour method**. This method has been contributed to Argus ONE by Subterranean Research, Inc.

Note that a single open or closed contour in a layer will set a value for the entire layer. Note also that the layer default value is ignored.

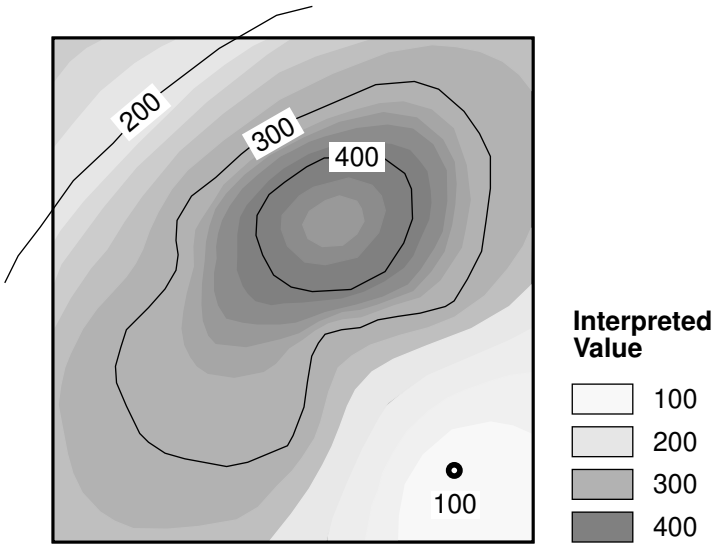


Figure 24. InvDistSq Contour Method

The **InvDistSq Contour method** is a smooth, inverse distance squared search that can be used with any kind of contour; point, open or closed. Open or closed contours are treated as though the vertices of the contours are distinct points, and so this method can show spurious interpolation artifacts along contours with few vertices (effect not shown here). This method shows a regression towards a local mean value that can be seen Figure 24 where the values outside of the level 200 contour begin to rise, and those inside the 400 level contour begin to fall. The interpolation is much smoother than the **Interpolation Contour method**, to which it bears the closest resemblance. This method has been contributed to Argus ONE by Subterranean Research, Inc.

> Note that a single open or closed contour in a layer will set a value for the entire layer. Note also that the layer default value is ignored.

The **ScanN2D Contour method** is a neural-network-based interpolation which requires some prior data manipulation and layer programming to use. This method has been contributed to Argus ONE by Subterranean Research, Inc. More information on these methods is available from the Argus World Wide Web site.

VISUALIZING AND INSPECTING VALUES IN LAYERS

Contours representing any layer information may be colored according to value. This is done by activating the layer of interest and then holding down the left mouse button on the arrow next to **Color** found along the left side of the *Argus ONE window*. This displays a list of available parameters of the layer. The one upon which the color scale should be based is then selected from the list, coloring the layer's contours.

A display of the selected parameter's value at any point in the workspace may be found above the lower left corner of the workspace in the window border. This value changes continuously as the cursor location changes.

ADDING OR CHANGING LAYERS

The Geospatial Information layers created by the Argus-SUTRA GUI and discussed below are directly correlated to the information used in the numerical model. In many SUTRA modeling projects, information required for direct input to the simulation may not be available, while proxy information that may be used to infer hydrogeologic properties may indeed be available. For example, instead of having information on hydraulic conductivity, the spatial distribution of soil type may be available. Because hydraulic conductivity may be inferred from soil type, the hydraulic conductivity can be linked to this information.

Geospatial Information layers can be added to the layers already created by the Argus-SUTRA GUI. For example, a geospatial information layer for "soil type" can be added, and a mathematical expression for the hydraulic conductivity as a function of soil type can be entered in the *Expression dialog* for the maximum and minimum parameters of the ***Hydraulic Conductivity*** layer that is accessed through the *Layers dialog*.

The Geospatial Information layers created in the Argus-SUTRA GUI may be regarded by advanced users as a basic set that can be augmented depending on the unique demands of each modeling project. By adding Geospatial Information layers, a convenient computing environment can be created for modeling with SUTRA within the Argus ONE application, that eliminates the need to perform correlations with proxy information externally before importing the results.

> Note: New users need not add any layers before basic modeling, as the set provided by the Argus-SUTRA GUI already allows all of the functionality of the SUTRA model to be accessed.

A Warning about Layer and Parameter Names

Although new Geospatial Information layers may be added, *the layers and parameters provided by the Argus-SUTRA GUI should not be deleted or renamed by the casual user*. Doing this may cause the application to fail or give incorrect results. The names and functionality of these layers are required to link the geospatial information to the finite-element mesh and to export this information in a form that can be read by SUTRA. Advanced users may, however, modify the functionality of these layers (in the *Expression dialog* discussed below) to extend the functionality of the GUI as needed.

SUTRA MODEL LAYERS

SUTRA Model is a group of layers that define the SUTRA simulation model geometry and discretization, as well as the locations in the model where observations of results will be made. These include the layers ***Domain Outline***, ***Mesh Density***, ***Fishnet Mesh Layout***, ***SUTRA Mesh***, and ***Observation***. When the user-specified SUTRA Interface Configuration requires a Fishnet Mesh, then the ***Fishnet Mesh Layout*** layer is included in the group, and the ***Mesh Density*** layer is not. When the user-specified SUTRA Interface Configuration requires an Irregular Mesh, then the ***Fishnet Mesh Layout*** layer is not included in the group.

DOMAIN OUTLINE

The ***Domain Outline*** is a special Argus ONE layer type that contains the closed contour lines representing the maximum areal extent of the simulated region when using an Irregular Mesh generated by Argus ONE. "Domain outline" refers to the contours that will exactly contain the entire finite-element mesh that is to be generated. Geospatial information coverage in other layers is required everywhere within the domain outline for the finite-element mesh, and may extend areally beyond the domain outline. Complex domain boundary shapes are possible; for example, islands of areas external to the SUTRA simulation can be placed within the outermost domain outline (Figure 25).

The contours in this layer also contain information controlling finite-element mesh density which defines the desired nominal size of elements in the discretization. This information, in conjunction with the layer ***Mesh Density*** and the geometry of objects in these layers, determines the distribution and size of mesh elements. Mesh density is strongly affected by the nearness of contours and the size of contour segments, as described in the Argus User's Guide.

Point and open contours may also be specified in the ***Domain Outline*** layer. While these do not determine the external boundaries of the mesh, they may be used to define the nominal size of elements along these objects, and both nodes and edges of elements will fall along these contours when the mesh is generated.

> Note: Any point or contour objects which must correspond with node location, such as contours in the ***Fluid Sources*** layer, must be copied to the ***Domain Outline*** layer before a useful mesh can be generated.

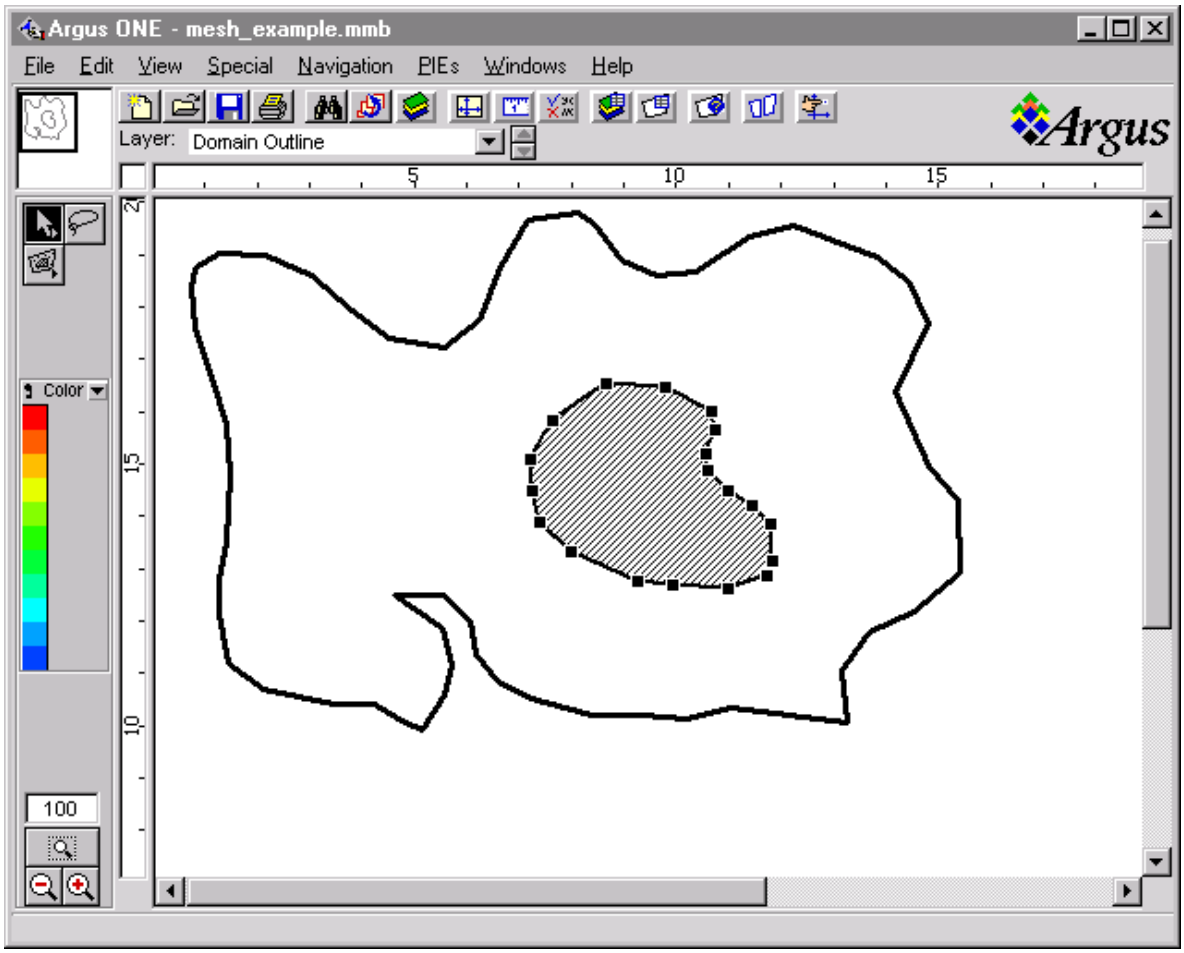


Figure 25. Domain Outline, example showing vertices on a closed contour

The procedure for creating a domain outline follows.

1. After the ***Domain Outline*** layer has been made active, the closed contours in the ***Domain Outline*** layer can either be generated with the contour drawing tool found in the tool palette at the left of the *Argus ONE window* or they can be imported from **TEXT**, **DXF** or ArcView **Shape** files.

2. To draw a domain-outline contour manually, first click on the closed-contour drawing tool. Then to specify the contour's position and shape, click in the white work area of the window in a sequence of positions to form a nearly-closed contour. To close the contour, double-click on the last position desired, then move the mouse away from the last point. The contour automatically closes and the *Contour Information dialog* appears (Figure 26).

3. This dialog is used to specify the size of quadrilateral elements. An **element_size** must be specified that refers to the desired typical size of an element side (in the distance units already specified for the project). This number is typed into the data box to the right of the word, "**element_size**" and just below the label, "**Value**". In Figure 26, an **element_size** value of 0.5 is specified. Click **OK** to exit the dialog box, and draw additional contours if desired.

   Note that Argus manuals refer to **element_size** as **mesh density**.

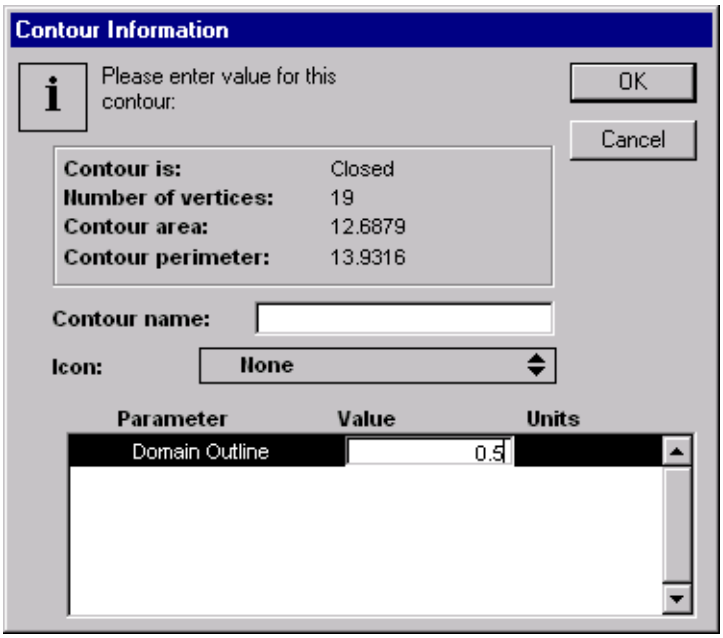


Figure 26. Contour Information Dialog

4. To draw an open or point contour instead, click and hold the mouse button down over the contour drawing tool (Figure 27) and make a selection from the menu that appears. Clicking in the white work area then draws the selected type of contour, and the *Contour Information dialog* again appears for entry of the **element_size** value when the contour is completed. Remember that the interpretation of values between open contours depends on the interpretation type for this layer.

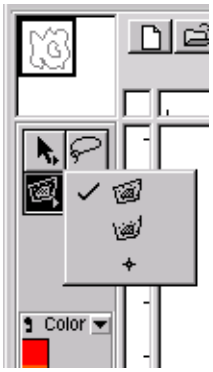


Figure 27. Contour Tool

5. Double-clicking on any contour object brings back the *Contour Information dialog* allowing modifications to be made to the values assigned to the object.

More information on drawing contours and on the drawing tools may be found in the Argus ONE User's Guide.

Note that the ***Domain Outline*** layer appears in the group even when the user-specified SUTRA Interface Configuration requires a Fishnet Mesh due to Argus ONE restrictions. However, it is not used at all to define the mesh boundaries in this case (see ***Fishnet Mesh Layout*** layer description "Fishnet Mesh Layout" (page 56) for more information), and should be ignored.

MESH DENSITY

Mesh Density, is an Argus ONE Information layer that is used by the ***SUTRA Mesh*** layer to define the refinement of the finite-element mesh (in addition to the **element_size** information contained in the ***Domain Outline*** layer). This layer may contain point contours (i.e. points), open contours, and closed contours representing the spatial distribution of **element_size** desired by the user. Each contour has a value of the **element_size** associated with it, similar to a topographic map, and together the contours define the **element_size** throughout the region of interest.

When regions where increased or decreased discretization within the model domain are desired by the user, the spatial distribution of **element_size** may be defined in this layer. Remember that the **element_size** along the domain boundary will still be based on the value and node-spacing in the ***Domain Outline*** layer.

When this layer is active, and as for all Information layers, the spatial distribution of **element_size** may be imported from **TEXT**, **DXF** or ArcView **Shape** files.

Alternatively, the point, line and closed contour tools may be used to manually specify the approximate size of quadrilateral elements in regions of the modeled domain. After using one of the contour tools to draw a contour, a dialog box appears where the **element_size** is entered; the **element_size** is the desired size of the quadrilateral elements in project units, prescribed at the contour just drawn.

Examples of using the ***Mesh Density*** layer to specify the finite-element discretization may be found in the Argus ONE User's Guide.

If the ***Mesh Density*** layer contains no contours (no objects), then the **element_size** specified for contours in the layer, ***Domain Outline***, is used in creating the mesh and this layer is ignored. When the user-specified SUTRA Interface Configuration requires a Fishnet Mesh, then the ***Mesh Density*** layer is not included in the group and is not available, nor is it needed to define the mesh.

FISHNET MESH LAYOUT

The ***Fishnet Mesh Layout*** layer is included in the group only when the user-specified *SUTRA Interface Configuration window* requires a Fishnet Mesh. This type of meshing is provided by the USGS SUTRA PIE. A Fishnet Mesh consists of superblocks (large contiguous quadrilaterals) each subdivided into a specified number of quadrilateral finite elements connected four to each internal node. The mesh may be considered similar to a deformed finite-difference grid.

This layer contains closed contours (only) in the form of quadrilateral "superblocks" that describe the external and internal boundaries of the Fishnet Mesh. Each superblock contour contains information describing the desired number of quadrilateral finite elements that will fill the superblock in the X and Y directions when the mesh is generated. The special procedure for generating a Fishnet Mesh is described here.

Prepare to draw the layout of the Fishnet Mesh

1. Create a problem setup using the *SUTRA Interface Configuration window* that uses a Fishnet Mesh.

2. Make the ***Fishnet Mesh Layout*** layer active.

3. From the **Special** menu along the top of the *Argus ONE window*, select **Allow Intersection**. This allows the superblocks to be joined perfectly.

Draw the first superblock

4. Select the closed-contour drawing tool at the left of the window.

5. Draw a quadrilateral in the white work area of the window by clicking at the desired location of each corner, proceeding either clockwise or counter-clockwise around the quadrilateral.

6. Double click the fourth (last) corner location, and move the mouse away from the point.

7. The *Contour Information* dialog box appears. Enter the desired number of elements in the X and Y directions next to the names **elements_in_x** and **elements_in_y** below the word **Value**. These values determine how many rows and columns of quadrilateral finite elements this superblock will be divided into when the mesh is generated. Then to exit the dialog, click **OK** to accept the values or **Cancel** to erase this superblock.

Depending on the orientation of the superblock, the X and Y directions referred to in the dialog box may be ambiguous requiring the user to experiment with the row and column directions for discretization.

superblocks may contain redundant information concerning number of rows and columns of elements. For example, each superblock in a pair of superblocks side-by-side in the X direction will have a value for **elements_in_y**. Because the Fishnet mesh design requires there to be the same number of rows of elements across both blocks, the user should provide each superblock with the same value for **elements_in_y**. The SUTRA PIE is designed so that, if these numbers do not agree, then the higher value is applied throughout that row or column of superblocks.

Draw additional superblocks

8. Usually, the user will want to join at least one side of each new superblock to a previously-drawn one. To draw the next superblock, move the cursor, a large plus sign (still selected from the closed-contour drawing tool), over one of the corners (vertices) of the previous superblock along the side to which the new one will be joined. When the cursor becomes hollow, then Argus ONE interprets its location as being exactly over the previous vertex (the **Allow Intersection** command enables this behavior of Argus ONE). Click on the corner.

9. If another vertex of a previous superblock is to be shared with the new superblock, then move the cursor over the next previously-existing vertex and click again when the cursor becomes hollow.

10. Then input the remaining corners of the new superblock in order, repeating steps 6 and 7 to complete the superblock. Each superblock must have four sides

11. Begin a new superblock according to step 8, if desired. If no more superblocks are needed, then continue with step 12.

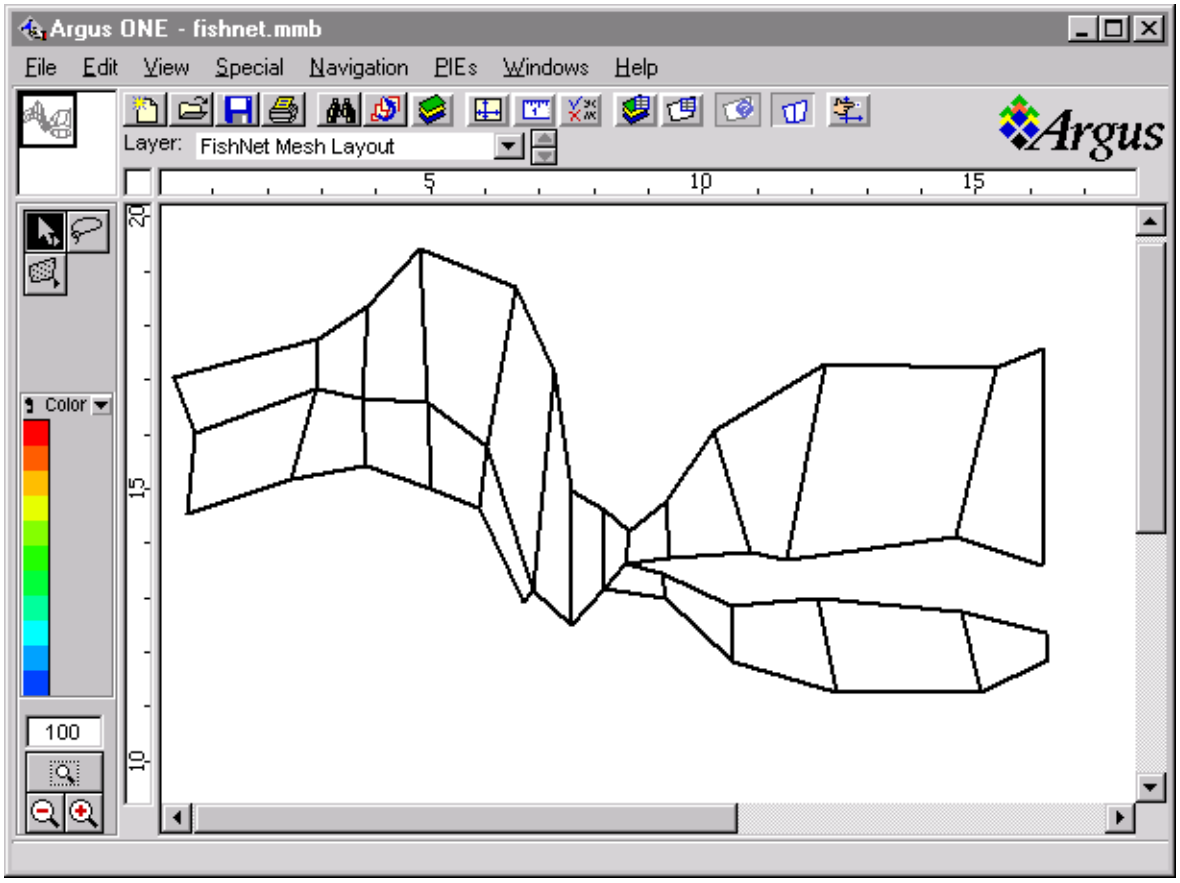


Figure 28. Fishnet Mesh Outline, example

Create the mesh from the layout of the Fishnet Mesh

12. Make the ***SUTRA Mesh*** layer active.

13. Select **Import SUTRA Mesh...** from the **File** menu along the top of the window, then select **Fishnet Mesh...** from the sub-menu that appears.

    Note that the ***SUTRA Mesh*** layer must be empty (i.e contain no old mesh) for these choices to appear.

    The SUTRA PIE then generates a mesh according to the specifications given in the ***Fishnet Mesh Layout*** layer and places the mesh into the ***SUTRA Mesh*** layer.

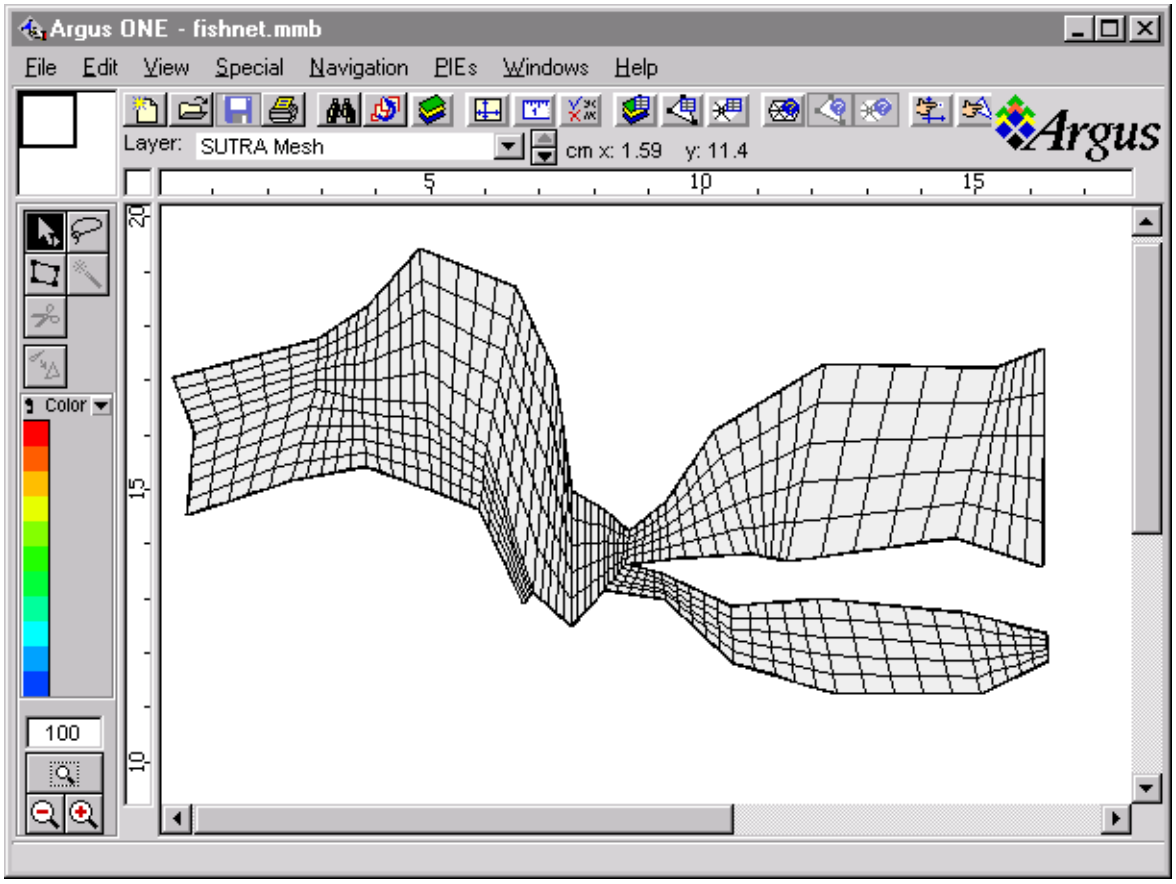


Figure 29. Completed Fishnet Mesh, for layout shown in Figure 28

A completed ***Fishnet Mesh Layout*** layer is shown in Figure 28 for the case where each superblock was divided into five rows and columns of elements (**elements_in_x** and **elements_in_y** are both set to 5). Almost any arrangement of superblocks is possible. The resulting mesh that appears in the ***SUTRA MESH*** layer is shown in Figure 29.

If desired, the coordinates of the vertices of superblocks may be imported into the ***Fishnet Mesh Layout*** layer, instead of drawing manually, after entering them into a text file using an editor. The proper format for this file may be obtained by drawing an arbitrary ***Fishnet Mesh Layout*** and exporting this as a text file using the **Export Fishnet Mesh Layout** command found under the **Export** command of the **File** menu along the top of the window. An example of the file for the first three superblocks of Figure 28 is shown below.

```
## Name:
## Icon:0
# Points Count   Value
5   5   5
# X pos   Y pos
8.21972222222222   14.5733333333333
8.21972222222222   13.1269444444444
8.61899544339292   13.6016475639678
8.67833333333333   14.1852777777778
8.21972222222222   14.5733333333333

## Name:
```

```
## Icon:0
# Points Count   Value
5   5   5
# X pos   Y pos
8.21972222222222   13.1269444444444
9.34861111111111   12.9505555555556
9.27805555555555   13.4091666666667
8.61899544339292   13.6016475639678
8.21972222222222   13.1269444444444

## Name:
## Icon:0
# Points Count   Value
5   5   5
# X pos   Y pos
8.67833333333333   14.1852777777778
8.61899544339292   13.6016475639678
9.41916666666667   13.6913888888889
9.34861111111111   14.7144444444444
8.67833333333333   14.1852777777778
```

The editor may be used to modify the exported text file containing the arbitrary superblock coordinates to obtain the desired number of superblocks with correct coordinates (as done in the section, "Henry Seawater Intrusion Problem with Variable-Density Flow" (page 19)). More information on importing contours may be found in the Argus User's Guide.

SUTRA MESH

For the case when the user-specified SUTRA Interface Configuration requires an Irregular Mesh, and after the domain outline has been specified in the layer, ***Domain Outline***, an irregular finite-element mesh can be generated automatically using the magic wand tool from the tool palette at the left side of the *Argus ONE window*. A discussion of the operation of the tools in a ***Mesh layer*** type can be found in the Argus ONE User's Guide. These tools may be used to inspect nodal and element geometries and parameters, to move nodes, and to override (change) node or element parameter values that were automatically assigned by Argus ONE. For the case when the user-specified SUTRA Interface Configuration requires a Fishnet Mesh, a different procedure is required to generate the mesh, as described in the previous section on "Fishnet Mesh Layout" (page 56).

To generate an Irregular Mesh, make the ***SUTRA Mesh*** layer active, click on the magic wand tool, and then click the magic wand cursor within the area of the domain boundary. Argus ONE then generates a mesh according to the density information contained in the ***Domain Outline*** layer and, if specified, in the ***Mesh Density*** layer.

If a mesh already exists in the layer, then the user is prompted to accept deletion of all existing elements before the new mesh is generated. An Irregular Mesh Is shown in Figure 30 for the domain shown in Figure 25 on page 54.

Note that if the view has been rescaled by user selection of Non-uniform in the Argus ONE Scale and Units dialog, then elongated elements are produced in an irregular mesh. Horizontally-long elements are often appropriate for cross-sectional modeling of aquifer systems.

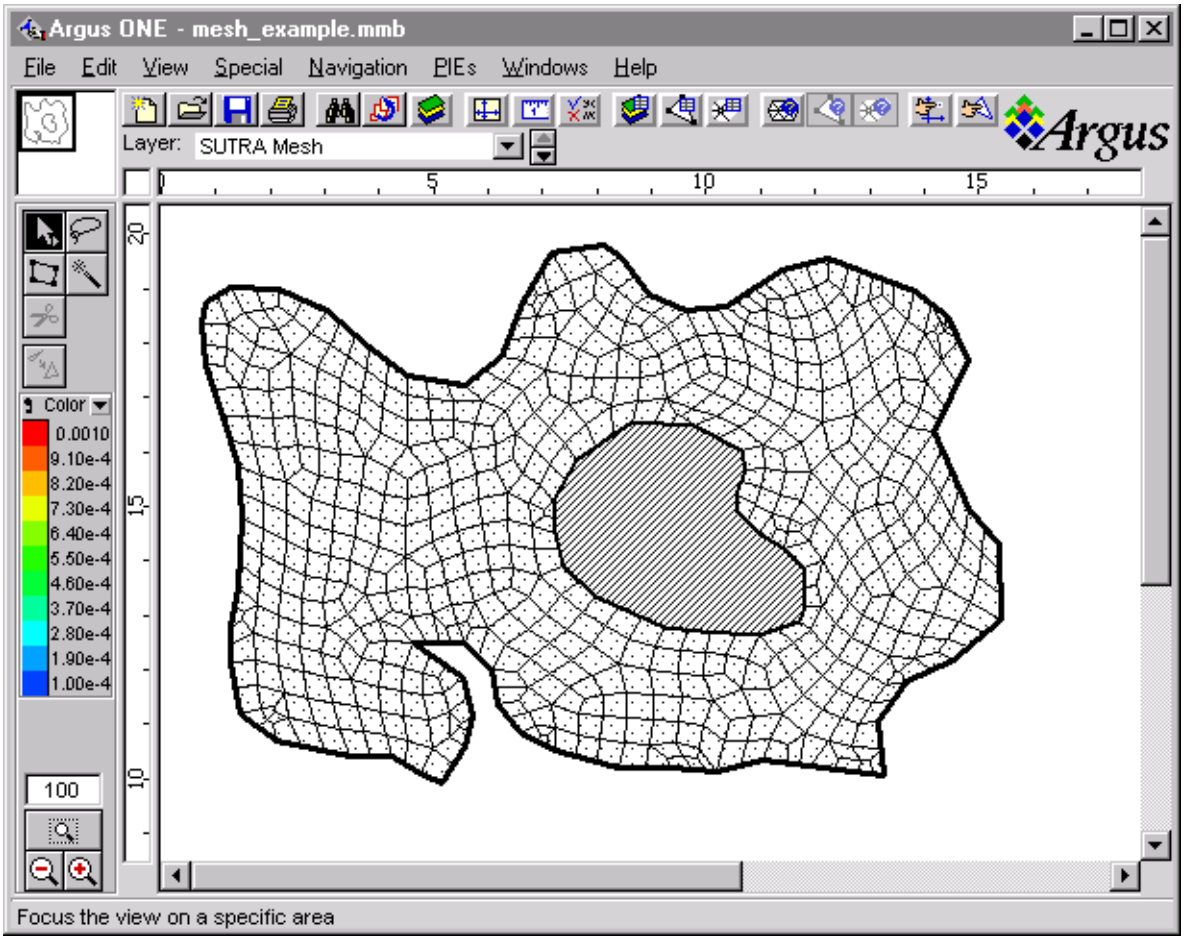


Figure 30. Irregular Mesh, with inactive area, for domain shown in Figure 25 on page 54

The ***SUTRA Mesh*** layer contains values for of all the spatially-distributed parameters required for running a SUTRA simulation, through references to and functions of the other layers. Depending on SUTRA's requirements for each parameter, as specified in the documentation for the SUTRA input data sets (Voss, 1984), a parameter may receive a value for each node or for each element in the mesh. These values are derived from the various Information layers described in following sections in a manner controlled by the file, **SUTRA.LAY**. The Argus ONE environment refers to the assignment of values in one Information layer to another as "linking". Table 4 lists the parameters of the ***SUTRA Mesh*** layer, their association with nodes or elements, and the name of the Information layer to which they are linked (i.e. from which their values are derived). The full Argus ONE name of each parameter is given as "Layer.parameter", for example, the first parameter, NREG, in the table is "***SUTRA Mesh.NREG***". The names, links to other layers, and logic associated with parameters in the SUTRA Mesh layer are defined in the **SUTRA.LAY** file.

The uppercase names of ***SUTRA Mesh*** parameters correspond exactly with variables used by the SUTRA model, and are described in the SUTRA documentation.

There is one exception to the above. The SUTRA parameter, "UBC", representing the concentration or temperature of any fluid that may flow into the model at a node where pressure or hydraulic head is specified has been changed to "pUBC" in this GUI to distinguish it from the specified concentration or temperature boundary condition type, which retains its original name ("UBC").

Two of the ***SUTRA Mesh*** parameters, QIN and QUIN, a fluid source and energy or solute source, respectively, have additional logic associated with their values. As noted in the description of the ***Sources of Fluid*** and ***Sources of Energy or Solute*** layers, to which these nodewise parameters are linked, these sources may be specified at point, line (open) or closed contours. Depending on which type of contour is specified in the "***Sources***" layer, the resultant source value provided as a parameter in the "***Sources***" layer may require multiplication by the open contour length or closed contour area associated with the node; these multiplications are carried out automatically.

Table 4: Parameters of the SUTRA Mesh Layer

Parameter	Type	Linked to:
NREG	Node	Unsaturated Properties.region (or missing for 'saturated-only' configuration)
THICK	Node	Thickness (or missing for 'cylindrical mesh' configuration)
POR	Node	Porosity
LREG	Element	Unsaturated Properties.region (or missing for 'saturated-only' configuration)
PMAX	Element	Permeability.maximum or Hydraulic Conductivity.maximum
PMIN	Element	Permeability.minimum or Hydraulic Conductivity.minimum
ANGLEX	Element	Permeability.angle_of_max_to_x_axis Hydraulic Conductivity.angle_of_max_to_x_axis
ALMAX	Element	Dispersivity.longdisp_in_max_permdir
ALMIN	Element	Dispersivity.longdisp_in_min_permdir
ATMAX	Element	Dispersivity.trandisp_in_max_permdir
ATMIN	Element	Dispersivity.trandisp_in_min_permdir
QIN	Node	Index(NodeAboveCntr(Sources of Fluid)+1, NodeEffectiveValue(Sources of Fluid.RESULTANT_FLUID_SOURCE), Sources of Fluid.RESULTANT_FLUID_SOURCE, Sources of Fluid.RESULTANT_FLUID_SOURCE* NodeEffectiveLength(Sources of Fluid), NodeEffectiveValue(Sources of Fluid.RESULTANT_FLUID_SOURCE))
UIN	Node	If(NodeAboveCntr(Sources of Fluid)=1 | NodeAboveCntr(Sources of Fluid)=2, Sources of Fluid.concentration_of_source, NodeEffectiveValue(Sources of Fluid.QINUIN)/ NodeEffectiveValue(Sources of Fluid.RESULTANT_FLUID_SOURCE)) or If(NodeAboveCntr(Sources of Fluid)=1 | NodeAboveCntr(Sources of Fluid)=2, Sources of Fluid.temperature_of_source, NodeEffectiveValue(Sources of Fluid.QINUIN)/ NodeEffectiveValue(Sources of Fluid.RESULTANT_FLUID_SOURCE))
time_dependent_fluid_sources	Node	Sources of Fluid.time_dependence

Table 4: Parameters of the SUTRA Mesh Layer

Parameter	Type	Linked to:
QUIN	Node	Index(NodeAboveCntr(Sources of Energy)+1, NodeEffectiveValue(Sources of Energy.RESULTANT_ENERGY_SOURCES), Sources of Energy.RESULTANT_ENERGY_SOURCES, Sources of Energy.RESULTANT_ENERGY_SOURCES* NodeEffectiveLength(Sources of Energy, NodeEffectiveValue(Sources of Energy.RESULTANT_ENERGY_SOURCES)) or Index(NodeAboveCntr(Sources of Solute)+1, NodeEffectiveValue(Sources of Solute.RESULTANT_SOLUTE_SOURCES), Sources of Solute.RESULTANT_SOLUTE_SOURCES, Sources of Solute.RESULTANT_SOLUTE_SOURCES* NodeEffectiveLength(Sources of Solute), NodeEffectiveValue(Sources of Solute.RESULTANT_SOLUTE_SOURCES))
time_dependent_energy_or _solute_sources	Node	Sources of Solute.time_dependence or Sources of Energy.time_dependence

PBC	Node	If(NodeAboveCntr(Specified Pressure),
		Specified Pressure.specified_pressure,
		$n/a) or
		If(NodeAboveCntr(Specified Head),
		Specified Head.specified_head,
		$n/a)

pUBC	Node	Specified Pressure.concentration or Specified Hydraulic_Head.temperature
time_dependent_specified _head_or_pressure	Node	Specified Pressure.time_dependence or Specified Hydraulic_Head.time_dependence

UBC	Node	If(NodeAboveCntr(Specified Concentration),
		Specified Concentration.specified_concentration,
		$n/a) or
		If(NodeAboveCntr(Specified Temperature),
		Specified Temperature.specified_temperature,
		$n/a)

time_dependent_specified _concentration_or _temperature	Node	Specified Concentration.time_dependence or Specified Temperature.time_dependence
PVEC	Node	Initial Pressure.initial_Pressure or Initial Hydraulic_Head.initial_hydraulic_head
UVEC	Node	Initial Concentration.initial_concentration or Initial Temperature.initial_temperature
INOB	Node	If(Observation.is_observed, NodeNumber(), 0)

Visualizing Geospatial Information Linked to SUTRA Mesh Elements

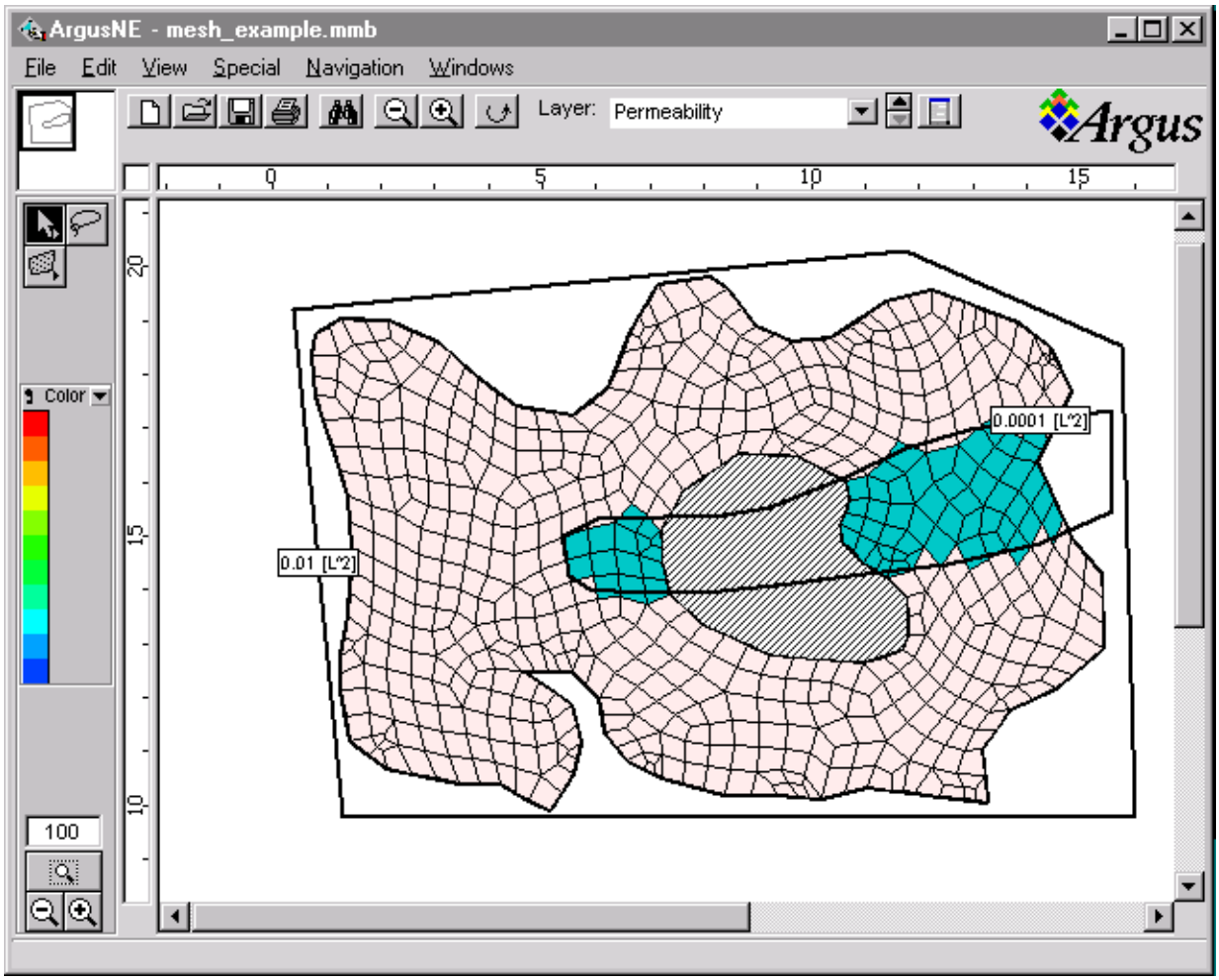


Figure 31. Elementwise Display of Parameter Values with a Color Map, example

It is important for the user to check whether the mesh parameters have been assigned by element as desired. The spatial distribution of any elementwise parameter of the ***SUTRA Mesh*** layer can be easily displayed as a color map, as follows.

When the ***SUTRA Mesh*** layer is active, holding the mouse button down over the triangle next to the word "Color" above the color band along the left edge of the *Argus ONE window* provides a list of possible elementwise mesh parameters by which elements can be colored.

Figure 31 shows the list and a mesh colored by parameter PMAX, where the closed contours are regions of constant permeability specified for the ***permeability.maximum*** layer parameter.

> It is sometimes useful to make the ***SUTRA Mesh*** layer transparent, which can be done by turning off the option "Opaque Elements" under the menu **View**.

The properties assigned to any element may be viewed (and modified) by double-clicking on the element of interest while the ***SUTRA Mesh layer*** is active.

Visualizing Geospatial Information Linked to SUTRA Mesh Nodes

It is important for the user to check whether the mesh parameters have been assigned by node as desired. The spatial distribution of any nodewise parameter of the ***SUTRA Mesh*** layer can be easily displayed as a color map, as follows.

1. Activate the ***Map layer***, or create a new map type layer and activate it.

2. Hold the mouse button down over the post-processing tool at the lower right of the tools found at the upper left edge of the *Argus ONE window*, and select the contour drawing tool from the pull-down list (Figure 32); this is the tool with concentric circles.

3. Click and drag the cursor anywhere in the workspace forming a small rectangle and release the mouse. This opens the *Contour Diagram dialog*.

4. In the "**Layer:**" box, select **SUTRA Mesh** (if it is not already selected).

5. In the "**Value:**" box, select the nodewise parameter of the ***SUTRA Mesh layer*** that should be contoured.

6. Click the **Position** tab along the top of the window and then click on "**Overlay Source Data**" to force the plot to exactly overlay the model domain.

7. Click **OK** and the plot is drawn. This creates a contour plot of the nodewise parameter of interest overlaying the mesh. Double-clicking this plot with the arrow tool brings back the *Contour Diagram Dialog window*, allowing modifications to be made to plot style. See "Post-Processing Plot Manipulation" (page 87) for more information.

If the mesh or the Information layer (to which the nodewise parameter plotted is linked) are changed, this process must be repeated in order to visualize the new nodewise parameter distribution.

.

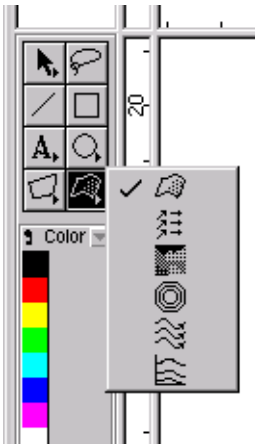


Figure 32. Post-Processing Tool, drawing contours

Nodewise values can also be inspected and modified individually in the ***SUTRA Mesh*** layer by double-clicking the node of interest, which opens a window containing all information about the node.

OBSERVATION AREAS

The ***Observation*** layer is an Information layer that contains the spatial distribution of locations at which the user desires "observation-node output" from SUTRA. This output, described fully in the SUTRA documentation, provides detailed time tracks of pressure, concentration or temperature, and saturation at user-selected nodes and at a user-selected time-step interval. This type of output is usually used for plotting time-drawdown curves or solute breakthrough curves.

> The user may select point, open or closed contours as regions where observations should be made during the SUTRA simulation. For each contour object, the value of the contour's parameter, ***is_observed***, must be set to **True (or 1)**. Non-spatial information concerning the frequency of the observations is also required and is specified in the OUTPUT pane of the *SUTRA Non-Spatial Information dialog*.

Nodes in the ***SUTRA Mesh*** layer falling within the selected areas, or directly above the open contours or selected points will be marked as observation nodes for input to SUTRA.

> Note that for the point and open contours, some nodes will fall exactly above these locations when the mesh is generated only if these contours are first copied to the ***Domain Outline*** layer. Information about copying and pasting objects in Argus ONE can be found in the Argus ONE User's Guide.

The ***Observation*** layer uses the **Exact Contour method** of interpretation so that observations will only be defined inside of closed contours, or directly below open and point contours.

GEOSPATIAL INFORMATION LAYERS

The geology and hydrology of an area are stored in Argus ONE as two-dimensional maps of the various parameters. These coverages are referred to as Geospatial Information layers and the specific ones used by the Argus-SUTRA GUI are described below. Separation of the underlying physical information from the simulation model provides a powerful hydrogeologic tool. The GIS coverages, described below and prepared for SUTRA simulation, may be used interchangeably for any simulation model coupled with Argus ONE.

The Geospatial Information layers in the Argus-SUTRA GUI in layer groups ***Hydrogeology*** and ***Initial Conditions***, are initially assigned the interpretation method **Nearest Contour method** (see "Interpretation of Contours and Point Data" (page 45)). The layer groups ***Hydrologic Sources*** and ***Hydrologic Boundaries*** are assigned the interpretation method **Exact Contour method**. An exception to the groupings mentioned above is layer ***Unsaturated Properties*** in layer group ***Hydrogeology***, which is assigned the **Exact Contour method**.These defaults are specified in the **SUTRA.LAY** file and may be changed interactively by the user.

Some user experimentation with the interpretation methods may be necessary in order to achieve the best possible spatial distribution for a given parameter.

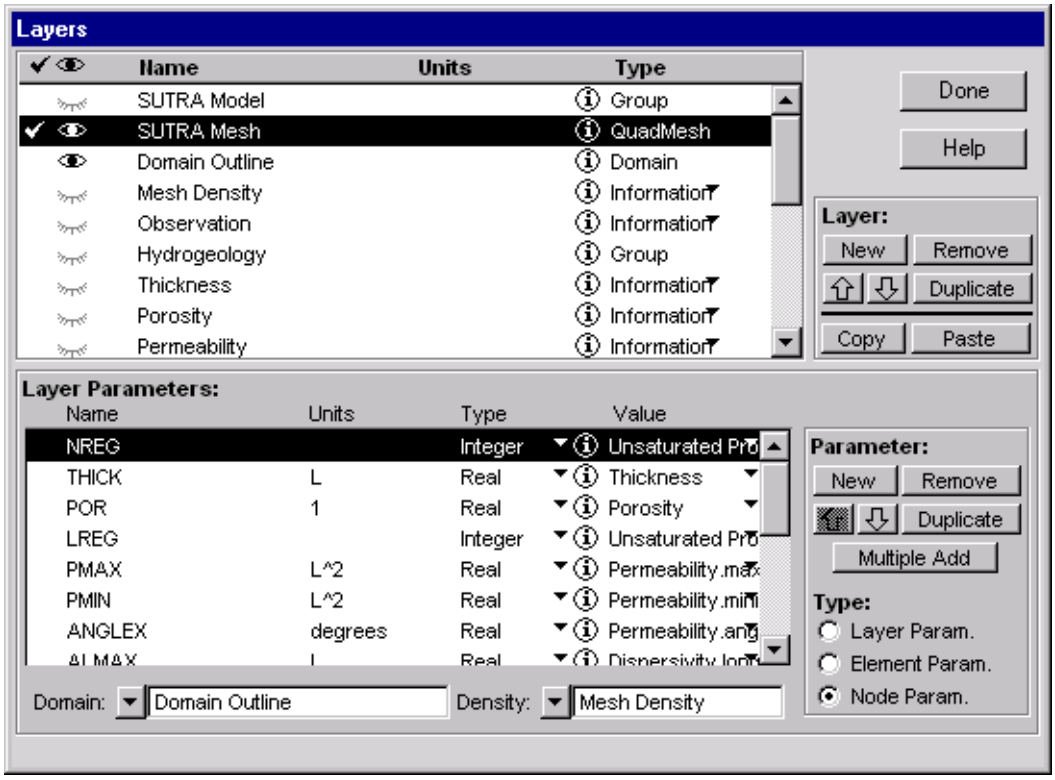


Figure 33. Layers Dialog

To specify a constant or default background value of a parameter, or to program a spatially-varying value based on a mathematical function, the *Layers dialog* (Figure 33), must be used. The list at the top of this window is the list of layers. The list at the bottom displays all the parameters associated with objects in the layer.

The *Layers dialog* may be displayed by clicking the **Layers** button in the *Layer List window*.

To assign a constant value, enter a function or a link to another layer to a particular parameter of a layer:

1. Highlight the layer under consideration (in the top box) by clicking it with the cursor. This shows the parameters associated with that Information layer in the list in the bottom box of the window.

2. Move the cursor to the **Value** column for the parameter for which a value will now be assigned, click and hold to cause the **Expression...** box to appear, and while holding, move the cursor over **Expression...** and release, causing the *Expression dialog* to appear.

3. Any numerical value, mathematical expression, parameter or function of a parameter in any layer may be used as the expression describing the spatial variation of this parameter.

4. After programming in the desired link, constant or function, click **OK** to exit the dialog or **Cancel** to exit without modifying the value.

More detailed information on using the *Layers dialog* and *Expression dialog* can be found in the Argus ONE User's Guide. (See also "Adding and Linking New Layers" (page 96).)

Occasionally, the user may have only scattered data points (rather than contours) as geospatial information, for example, measurements of the thickness of a hydrogeologic unit only where a number of boreholes penetrate to the bottom of the unit. These data can be imported into a layer if arranged in the format required by Argus ONE for scattered data files, as described in Supplement 2.5 of the Argus ONE User's Guide.

To use such data, the user must access the *Layers dialog* (Figure 33), activate the layer of interest (for example ***Thickness***), and select **Data** from the pull-down menu found in the **Type** column in the upper portion of the window. This changes the layer type from **Information** to **Data**.

After selecting **Done** to exit the window, the user may import a data file by selecting **Import...** in the **File** menu along the top of the *Argus ONE window*.

Interpolation of the data is automatic, and should be checked by contouring the data points as described in the section "Visualizing Geospatial Information Linked to SUTRA Mesh Nodes" (page 66). Various types of interpolation among data points are available through PIEs found on the Argus Web Site.

The Geospatial Information layers for the Argus-SUTRA GUI are grouped below by general type: ***Hydrogeology***, ***Hydrologic Sources***, ***Hydrologic Boundaries***, and ***Hydrologic Initial Conditions***. Each group contains a number of layers, each layer contains one or more parameters. After introduction of each parameter in the following sections, units are given for the parameter in square brackets; e.g. for ***Permeability.maximum***, the units are given as [L^2] representing arbitrary length-units squared. Table 5 shows units used and displayed in the GUI with their meanings.

.

Table 5: Unit Abbreviations

Unit	Meaning
[C]	any consistent concentration units
[degC]	Celsius degrees
[degrees]	angle in degrees
[E]	any consistent energy units
[L]	length units
[M]	mass units
[s]	seconds (time)
[T]	temperature (Celsius)
[1]	unitless, like porosity
[1,0]	true or false
x^2, y^3	x quantity squared, y quantity cubed

Parameter names such as ***Fluid Sources.RESULTANT_FLUID_SOURCE***, that are uppercase, are calculated, or linked to other layer parameters and should not be modified directly. In most cases modifications to such parameters can be done by changing the lowercase parameters of the layers. Advanced users who understand the functions or references in these parameters may modify them to suit specific modeling

needs. New users should consider parameters with uppercase names as "read-only" variables. Section "Adding and Linking New Layers" (page 96) explains how to add new layers and link the information from these layers into a SUTRA project.

Depending on the choice of problem type made by the user in the *SUTRA Interface Configuration window*, some of the layer and parameter names and the units of the parameters will be different. All of the variations are included in the descriptions that follow.

Each parameter of each layer is assigned a default value by the SUTRA PIE. The background value specifies a constant value of a parameter for the entire workspace that will be assumed in case no other user input is provided to that parameter. The default layer values are specified in file **SUTRA.LAY** with modifications from the **SUTRA.INI** file depending on the project type, and are given in Table 6.

Table 6: Default Background Values for Layer Parameters

Layer	Parameter	Value[1]
Fishnet Mesh	elements_in_x	$n/a
	elements_in_y	$n/a
Domain Outline	element_size	0
Mesh Density	element_size	0
Observation	is_observed	0
Thickness	thickness	1
Porosity	porosity	0.1
Permeability or Hydraulic Conductivity	maximum	1.0E-10 or 1.0E-3
	minimum	1.0E-10 or 1.0E-3
	angle_of_max_to_x_axis	0
Dispersivity	longdisp_in_max_permdir	0.5
	longdisp_in_min_permdir	0.5
	trandisp_in_max_permdir	0.1
Unsaturated Properties	region	0

Table 6: Default Background Values for Layer Parameters

Layer	Parameter	Value[1]
Sources of Fluid	total_source	$n/a
	specific_source	$n/a
	concentration_of_source or temperature_of_source	$n/a
	time_dependence	0
	RESULTANT_FLUID_SOURCE	$n/a
	QINUIN	$n/a
Sources of Energy or Sources of Solute	total_source	$n/a
	specific_source	$n/a
	time_dependence	0
	RESULTANT_ENERGY_SOURCES RESULTANT_SOLUTE_SOURCES	$n/a
Specified Pressure or Specified Head	specified_pressure or specified_head	$n/a
	concentration or temperature	$n/a
	time_dependence	0
Specified Concentration or Specified Temperature	specified_pressure or specified_head	$n/a
	time_dependence	0
Initial Pressure or Initial Head	initial_pressure or initial_head	0
Initial Concentration or Initial Temperature	initial_concentration or initial_temperature	0
Map	map	0
Point Data	value1	0

1.$n/a is a flag for an unassigned variable.

NOTE: Default values are only initial suggestions, and must be checked and reset, if needed, by the user to appropriate values for the user's project.

HYDROGEOLOGY

This group of Information layers contains information on the physical and material properties of the hydrogeologic unit of interest.

Thickness

The layer, ***Thickness***, contains information on the spatial distribution of the thickness of the hydrogeologic unit to be modeled, or for generic studies, the thickness of the two-dimensional medium being modeled. The layer contains only one parameter, ***Thickness.thickness*** [L], which contains the thickness value.

Porosity

The layer, ***Porosity***, contains information on the spatial distribution of the porosity of the hydrogeologic unit to be modeled, or for generic studies, the porosity of the two-dimensional medium being modeled. The layer contains only one parameter, ***Porosity.porosity*** [1], which contains the porosity value.

Permeability / Hydraulic Conductivity

The layer called ***Permeability*** (for cases using Pressure) or ***Hydraulic Conductivity*** (for simulations using Hydraulic Head), contains information on the spatial distribution of the principal values of permeability/hydraulic conductivity, and the direction of the anisotropy for the hydrogeologic unit to be modeled, or for generic studies, for the two-dimensional medium being modeled. The layer contains three parameters:

Permeability.maximum [L^2] (or ***Hydraulic Conductivity.maximum*** [L/s]) contains information on the spatial distribution of the maximum component of permeability (or hydraulic conductivity) of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

Permeability.minimum [L^2] (or ***Hydraulic Conductivity.minimum*** [L/s]) contains information on the spatial distribution of the minimum component of permeability (or hydraulic conductivity) of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

Permeability.angle_of_max_to_x_axis (or ***Hydraulic Conductivity.angle_of_max_to_x_axis***) [degrees] contains information on the spatial distribution of the direction of the maximum component of permeability (hydraulic conductivity) of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled. The angle is given in degrees measured counter-clockwise from the +X direction.

If the system is isotropic then both the maximum and minimum values must be set equal to the desired value, and the angle may be set to any arbitrary number. It may be convenient for isotropic systems to link the value of one of the permeability minimum or maximum parameters to the other. See "Adding and Linking New Layers" (page 96).

Dispersivity

The layer, ***Dispersivity***, contains information on the spatial distribution of the longitudinal and transverse dispersivities of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled. The layer contains four parameters:

Dispersivity.longdisp_in_max_permdir [L] contains information on the spatial distribution of the longitudinal dispersivity for flow in the maximum permeability (or hydraulic conductivity) direction of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

Dispersivity.longdisp_in_min_permdir [L] contains information on the spatial distribution of the longitudinal dispersivity for flow in the minimum permeability (or hydraulic conductivity) direction of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

Dispersivity.trandisp_in_max_permdir [L] contains information on the spatial distribution of the transverse dispersivity for flow in the maximum permeability (or hydraulic conductivity) direction of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

Dispersivity.trandisp_in_min_permdir [L] contains information on the spatial distribution of the transverse dispersivity for flow in the minimum permeability (or hydraulic conductivity) direction of the hydrogeologic unit to be modeled, or for generic studies, of the two-dimensional medium being modeled.

The additional components of the dispersivity values are extensions to the classical dispersion process provided by SUTRA (Voss, 1984; transverse extension added in 1990 revision). To obtain the classical dispersion model used by most other transport models, set the values of the longitudinal dispersivity in the maximum and minimum permeability directions equal to each other, and the values of the transverse dispersivity in the maximum and minimum permeability directions equal to each other. It may be convenient for isotropic systems in which variations in dispersivity are being investigated to link one of the dispersivity minimum or maximum parameters to the other for both the longitudinal and transverse components.

Unsaturated Properties

The layer, ***Unsaturated Properties***, contains information on the spatial distribution of regions having constant unsaturated properties in the hydrogeologic unit to be modeled, or for generic studies, in the two-dimensional medium being modeled. This layer only appears when a saturated-unsaturated problem type has been selected in the *SUTRA Interface-Configuration window*. The layer contains only one parameter, ***Unsaturated Properties.region*** [integer], which contains the identification number of the region. Usually the **Exact Contour method**, already specified as the default selection, should be used to define the location of regions in this layer, in which case only closed contours should be used for specifying regions. After closing a contour, its region number must be assigned by the user in the *Contour Information window* that appears; any number of contours may share the same region number. Each region represents one type of material having uniform unsaturated property functions. For example, region 1 may be a fine silt, region 2 may be clay, and region 3 may be coarse sand, and each type of material may occur in a number of places within the model domain. The default region number is zero. To create regions that share a boundary (such as geologic layers), turn on option **Allow Intersection** under menu **Special** before drawing contours.

If the user wishes to have nodes and element edges occur exactly at region boundaries, then for a Fishnet Mesh, these boundaries need to be defined as superblocks in a Fishnet Mesh layout, or for an irregular mesh, these must be defined as open contours in the ***Domain Outline*** layer. (Presently, the Argus ONE irregular meshing engine does not fill closed contours occurring within the outermost domain outline with elements. The recommended procedure would be to define a nearly-closed open contour in which the distance between the ends of the contour is roughly the length of the side of a typical element.)

> Unsaturated-flow simulation with SUTRA requires some user programming in SUTRA subroutine UNSAT in order for the user to specify the unsaturated functions for each region defined in this layer as described in Voss (1984). The FORTRAN source code for SUTRA and a FORTRAN compiler are required to use this feature.

HYDROLOGIC SOURCES

This group of Information layers contains information on fluxes (quantity per time) of fluid, solute, and energy entering or leaving the hydrogeologic unit.

The SUTRA-Argus ONE GUI allows specification of fluid sources (or sinks) and energy/solute sources (or sinks) at points, along curves and for delimited areas. Sources have positive values, whereas sinks are specified with a negative number. Sources of fluid may be either specified as a total amount of fluid per second (i.e. ***total_source***), or as the (total amount of fluid per second) per length or area of the source object (i.e. ***specific_source***). Sources of energy/solute may be either specified as a total amount of energy/solute per second (i.e. ***total_source***), or as the (total amount of energy/solute per second) per length or area of the source object (i.e. ***specific_source***).

Point sources may be assigned only a ***total_source***. These are created with the point contour drawing tool located along the left side of the *Argus ONE window*. These typically represent injection or withdrawal wells for fluid sources. The points representing point sources must be copied to the ***Domain Outline*** layer in order to force a node in the finite element mesh to be located at each point source when the mesh is regenerated.

Line or curve sources may be assigned either a ***total_source*** representing the amount of fluid or energy/solute for the entire object, or a ***specific_source***, representing the amount of fluid or energy/solute per length of the object. These are created with the open contour drawing tool located along the left side of the *Argus ONE window*. A fluid line source may represent, for example, recharge occurring along a reach of a river or irrigation canal. Depending on whether the user knows the total recharge along the canal reach, or the recharge per length of canal, would determine whether the user should specify ***total_source*** or ***specific_source*** in the ***Sources of Fluid*** layer. The contours representing these sources must be copied to the ***Domain Outline*** layer in order to force nodes in the finite element mesh to be located along each source contour when the mesh is regenerated.

Area sources may be assigned either a ***total_source*** representing the amount for the entire object, or a ***specific_source***, representing the amount of fluid or energy/solute per area of the object. These are created with the closed contour drawing tool located along the left side of the *Argus ONE window*. For fluid, an area source may, for example, represent ground-water recharge below a lake (then ***total_source*** could be specified for the entire lake as one object) or rainfall recharge rate (volume per area) for a particular area (specified as a ***specific_source*** in units [L/s] for the object). For energy, an area source may, for example, represent heat production in a magma body (then ***total_source*** could be specified for the entire body as one object) or radiogenic heat production rate (energy per area) for a particular area (specified as a ***specific_source*** in units [E/s/L^2] for the object).

> Because contours may not cross one another in Argus ONE, only one type of object may be specified at any location. Furthermore, a limitation of the SUTRA code is that it accepts only one source per location; thus, two sources, e.g. rainfall and a well, cannot be specified separately at the same location, and must be summed externally by the user before entering values into the Argus-SUTRA GUI.

To obtain areally-distributed values affecting nodes throughout the mesh, rather than exactly along contours in this layer, a new layer should be created and linked to the ***specific_source*** parameter. See "Adding a Precipitation Data Layer and Linking It to Fluid Sources Layer" (page 96).

Sources of Fluid

The layer, ***Sources of Fluid,*** contains information on the spatial distribution of the fluid sources and sinks (such as wells and recharge from rainfall) for the hydrogeologic unit to be modeled, or for generic studies, for the two-dimensional medium being modeled. Additionally the layer contains information on the solute concentration or temperature of the source fluids. The layer contains six parameters, however, two of these (the ones with the uppercase names) are automatically calculated by the Argus-SUTRA GUI from the others and must not be altered by the user. For any given object in the layer, the user may only specify three of the remaining four parameters, because a choice must be made between two of them as described below. When pressure has been selected as the hydraulic variable in the *SUTRA Interface Configuration window*, fluid sources use units of [M/s]; when hydraulic head has been selected, units of [L^3/s] are used. The parameters follow:

Sources of Fluid.total_source [M/s or L^3/s], allows the user to specify the total fluid source over any object specified in the layer.

> If an object for which ***total_source*** is specified extends outside of the domain boundary, not all of this total will be included in the model.

Sources of Fluid.specific_source [(M/s or L^3/s)/(L or L^2)], allows the user to specify the fluid source per length of an open contour object (a curve) or per area of a closed contour object (an area) in the layer.

Sources of Fluid.concentration_of_source [C] (or Sources of Fluid.temperature_of_source [degC]), contains the value of the concentration (or temperature) of inflowing fluid for each object in the layer.

Sources of Fluid.time_dependence [1,0], contains an indicator as to whether any of the parameters in an object will be specified as time-dependent and require user-programming in SUTRA subroutine, BCTIME. A value of 1 indicates that there is time-dependence, and a value of 0 (default) indicates that all parameters in the object do not vary with time.

The following 2 parameters may not be altered by the user for any object.

Sources of Fluid.RESULTANT_FLUID_SOURCE, is calculated by Argus ONE as specified in the file, **SUTRA.LAY**. The UPPERCASE parameter name is intended to mark this parameter as being an internally-calculated value that should not be modified directly. This parameter contains the resulting fluid source for each object, which is automatically calculated as follows. For a point object, it contains the ***total_source*** value specified by the user. For an open or closed contour, it contains either the ***specific_source***, if this was specified by the user, or the ***total_source*** divided by the length or area, respectively, of the contour. Thus, for non-point objects, ***RESULTANT_FLUID_SOURCE*** always contains a specific source value (i.e. total source per length or area of object).

Sources of Fluid.QINUIN,is calculated by Argus ONE as specified in the file, **SUTRA.LAY**. The UPPER-CASE parameter name is intended to mark this parameter as being an internally-calculated value that should not be modified directly. This parameter is an intermediate value used by the **UIN** parameter of the ***SUTRA Mesh layer***. The expression given for this parameter applies as the default value for the parameter in the entire layer. However, its actual purpose is only to correctly determine the net concentration (or temperature) of all source fluids assigned to nodes near closed contours (within one-half an element distance). Its value is the product of the fluid source and its concentration (or temperature). An expression for this parameter similar to the following is used in the **SUTRA.LAY** file (relations shown for solute):

> ***QINUIN = RESULTANT_FLUID_SOURCE * concentration_of_source***

The value of the ***SUTRA Mesh layer*** parameter ***UIN*** (concentration or temperature of fluid source at a node) then is defined using the parameter in the following logic.

For a point or open contour object:

UIN = ***Sources of Fluid.concentration_of_source***

Otherwise (i.e. for background and on or in closed contour objects):

UIN = NodeEffectiveValue(***Sources of Fluid.QINUIN***)
/ NodeEffectiveValue(***Sources of Fluid.RESULTANT_FLUID_SOURCE***)

The Argus ONE function, NodeEffectiveValue, gives an area-weighted average of its argument (see Argus ONE Users Guide). For each node within one-half element of closed contours, this second relation correctly determines the net concentration of all background and closed contour sources affecting the node, while the first simply assigns the point or line-source concentration to the nodes exactly below these source types.

In the case where the sources affecting a node are both inflows and outflows, there is an approximation the user must be aware of, as described in the following. Because SUTRA accepts only one source and source concentration (or temperature) per node, the various sources (and sinks) affecting each node are summed by the GUI giving a net source for each node. For a positive net source (i.e. an inflow) at a node, a net concentration of the fluid mixture must also be specified to SUTRA. When all the sources summed are inflows, calculation of a net concentration value of the net source is unambiguous because the concentrations of all inflowing fluids are given by the user as usual. When all of the sources affecting a node are outflows, *no* concentration needs to be specified to SUTRA at that node as this value will be a simulation result, as is usual with SUTRA. However, when some of the sources in the GUI affecting a node are inflows and some affecting the same node are outflows, and the resultant net source is an inflow, then the net concentration to be applied for this net inflow is ambiguous. This is because the concentration of the outflows to be summed with the inflows will generally be unknown to the user until after the simulation is complete. In this situation, the user should *guess* the concentration of outflowing fluid when specifying ***Sources of Fluid.concentration*** in order to approximate the net concentration. This need to guess outflow concentration prior to running a simulation will be a limitation until the SUTRA code is revised to accept multiple independent sources at each node. Note again that a guess is required only when both inflows and outflows occur in the neighborhood of a node, and the resulting source is an inflow.

To specify ***total_source*** or ***specific_source***, the user must place a numeric value for one of these in the ***VALUE*** column of the *Contour Information window* that appears after completion of drawing an object. The value next to the other type of source parameter, which is not being used, must be set to **$n/a**, the Argus ONE symbol for a value that is "Not Assigned". The value of ***concentration*** (or ***temperature***) must be set to a numeric value replacing the **$n/a** that appears for each new object. To switch between the two types of source specification, the user must place the **$n/a** in the type not being used. Using zero rather than **$n/a** will not work.

If the user desired to spread a single source evenly over the model domain, such as a recharge from rainfall, there would be two ways this could be accomplished. A closed contour could be drawn around the entire model domain, or the background value of ***Sources of Fluid.specific_source*** in the *Layers dialog* could be defined, setting it to a constant. Note that the interaction of background values and objects depends on the interpolation type (see "Interpretation of Contours and Point Data" (page 45)).

A visually appealing way to surround the model area with a closed contour would be to copy the contour representing the domain outline to this layer and assign it a source value.

Setting a value of 1 or "true" for ***Sources of Fluid.time_dependence*** will cause node numbers in the input file for SUTRA created by the Argus-SUTRA GUI to be negative. This, in turn, causes SUTRA to refer to the subroutine BCTIME at the beginning of each time step to obtain the time-dependent values for each node with a negative node number.

> BCTIME must be user programmed to obtain the desired type of time variation as described in Voss (1984). The FORTRAN source code for SUTRA and a FORTRAN compiler are required to use this feature.

A limitation of this version of the Argus-SUTRA GUI is that the separation of nodes into groups, where each group has a different time-dependence user-programmed in BCTIME, is not possible. The is because SUTRA does not currently accept such group input. Thus, negative node numbers will have to be manually discovered by the user for the purposes of programming BCTIME after mesh generation if there is to be more than one group of time-varying fluid source nodes. This may be done by using the **Search For...** command found under **Edit** along the top of the *Argus ONE window*, or by exporting the SUTRA dataset for the simulation and examining the file.

Sources of Solute / Energy

The layer, ***Sources of Solute*** (or ***Sources of Energy***) contains information on the spatial distribution of the solute (or energy) sources and sinks (such as dissolving minerals, radiogenic heat production or thermal conduction through a boundary) for the hydrogeologic unit to be modeled, or for generic studies, for the two-dimensional medium being modeled. The layer contains four parameters, however, one of these (the one with the uppercase name) is automatically calculated by Argus ONE from the others and may not be altered by the user although. For any given object in the layer, the user may only specify two of the remaining four parameters, because a choice must be made between two of them as described below. The parameters follow:

Sources of Solute/Energy.total_source [C/s or E/s], allows the user to specify the total source over any object specified in the layer.

Sources of Solute/Energy.specific_source [(C/s or E/s)/(L or L^2)], allows the user to specify the source per length of an open contour object (a curve), or the source per area of a closed contour object (an area) in the layer.

Sources of Solute/Energy.time_dependence [1,0], contains an indicator as to whether any of the parameters in an object will be specified as time-dependent and require user-programming in SUTRA subroutine, BCTIME. A value of 1 indicates that there is time-dependence, and a value of 0 (default) indicates that all parameters in the object do not vary with time.

The following parameter may not be altered by the user for any object.

Sources of Solute/Energy.RESULTANT_SOLUTE/ENERGY_SOURCE, is calculated by Argus ONE as specified in the file, **SUTRA.LAY**. The UPPERCASE parameter name is intended to mark this parameter as being different from the others. This parameter contains the resulting source for each object, which is automatically calculated as follows. For a point object, it contains the **total_source** value specified by the user. For an open or closed contour, it contains either the **specific_source**, if this was specified by the user, or the **total_source** divided by the length or area, respectively, of the contour. Thus, for non-point objects, ***RESULTANT_SOLUTE/ENERGY_SOURCE*** always contains a specific source value (i.e. total source per length or area of object).

To specify ***total_source*** or ***specific_source***, the user must place a numeric value for one of these in the **VALUE** column of the *Contour Information window* that appears after completion of drawing an object. The value next to the other type of source parameter, which is not being used, must be set to **$n/a**. To switch between the two types of source specification, the user must place the **$n/a** in the type not being used; using zero rather than **$n/a** will not work.

Setting a value of 1 or "true" for ***Sources of Solute/Energy.time_dependence*** will cause node numbers in the input file for SUTRA created by the Argus-SUTRA GUI to be negative. This, in turn, causes SUTRA to refer to the subroutine BCTIME at the beginning of each time step to obtain the time-dependent values for each node with a negative node number.

BCTIME must be user programmed to obtain the desired type of time variation as described in Voss (1984). The FORTRAN source code for SUTRA and a FORTRAN compiler are required to use this feature.

A limitation of this version of the Argus-SUTRA GUI is that the separation of nodes into groups, where each group has a different time-dependence user-programmed in BCTIME, is not possible, because SUTRA does not presently accept such group input. Thus, negative node numbers will have to be manually discovered by the user for the purposes of programming BCTIME after mesh generation if there is to be more than one group of time-varying solute/energy source nodes.

HYDROLOGIC BOUNDARIES

This group of Information layers contains information on the known levels (or changes in levels over time) of pressure, hydraulic head, solute concentration and temperature in the hydrogeologic unit, at locations where these levels are controlled by forces external to those occurring within the unit or generic model.

Specified Pressure / Hydraulic Head

The layer, ***Specified Pressure*** (or ***Specified Hydraulic Head***) contains information on the spatial distribution of known or fixed pressures or hydraulic heads within the hydrogeologic unit to be modeled, or for generic studies, within the two-dimensional medium being modeled. The layer contains three parameters:

Specified Pressure.specified_pressure [M/(Ls^2)] (or ***Specified Hydraulic Head.specified_hydraulic_head*** [L]), allows the user to specify the fluid pressure (or hydraulic head) over any object specified in the layer.

Specified Pressure.concentration [C] (or ***Specified Pressure.temperature*** [degC]), contains the value of the concentration or temperature of any fluid inflow that may occur for each object in the layer as a result of the specified pressure (or hydraulic head).

Specified Pressure.time_dependence [1,0], (or ***Specified Hydraulic Head.time_dependence*** [1,0]), contains an indicator as to whether either of the parameters in an object will be specified as time-dependent and require user-programming in SUTRA subroutine, BCTIME. A value of 1 indicates that there is time-dependence, and a value of 0 (default) indicates that all parameters in the object do not vary with time.

The SUTRA-Argus ONE GUI allows specification of pressures (or hydraulic head) at points, along curves and for delimited areas. To specify pressure, the user must place a numeric value in the **VALUE** column of the *Contour Information window* that appears after completion of drawing an object. The value of ***concentration*** (or ***temperature***) must be set to a numeric value replacing the **$n/a** that appears for each new object; this is the concentration or temperature for any fluid that enters the model as a result of the pressure (or hydraulic head) being held constant.

When the Argus ONE interpretation scheme is set to its default for this layer, (**Exact Contour method**), point, open and closed contour objects in this layer will only affect nodes in the ***SUTRA Mesh*** layer that occur exactly over (and within) these objects. This requires that the user copy open and point contours to the ***Domain Outline*** layer, and that the mesh be regenerated, to force the mesh to have nodes exactly along these contours. Some nodes naturally fall within the area delimited by closed contours, and these contours should not be copied to the ***Domain Outline*** layer (this would create holes in the mesh).

Setting a value of 1 or "true" for ***Specified Pressure.time_dependence*** (or ***Specified Hydraulic Head.time_dependence***) will cause node numbers in the input file for SUTRA created by the Argus-SUTRA GUI to be negative. This, in turn, causes SUTRA to refer to the subroutine BCTIME at the beginning of each time step to obtain the time-dependent values for each node with a negative node number. BCTIME must be user programmed to obtain the desired type of time variation as described in Voss (1984).

A limitation of this version of the Argus-SUTRA GUI is that the separation of nodes into groups, where each group has a different time-dependence user-programmed in BCTIME, is not possible, because SUTRA does not presently accept such group input. Thus, negative node numbers will have to be manually discovered by the user for the purposes of programming BCTIME after mesh generation if there is to be more than one group of time-varying specified pressure (or hydraulic head) nodes.

Specified Concentration / Temperature

The layer, ***Specified Concentration*** (or ***Specified Temperature***) contains information on the spatial distribution of known or fixed values of solute concentration (or temperature) within the hydrogeologic unit to be modeled, or for generic studies, within the two-dimensional medium being modeled. This specification refers not only to the fluid entering the model, but also to the exiting fluid and the fluid within the unit itself. The layer contains two parameters:

Specified Concentration.specified_concentration [C] (or ***Specified Temperature.specified_temperature*** [degC]), allows the user to specify the solute concentration (or temperature) over any object specified in the layer.

Specified Concentration.time_dependence [1,0], (or ***Specified Temperature.time_dependence*** [1,0]), contains an indicator as to whether either of the parameters in an object will be specified as time-dependent and require user-programming in SUTRA subroutine, BCTIME. A value of 1 indicates that there is time-dependence, and a value of 0 (default) indicates that all parameters in the object do not vary with time.

> The SUTRA-Argus ONE GUI allows specification of concentrations (or temperatures) at points, along curves and for delimited areas. To specify concentration (or temperature), the user must place a numeric value in the **VALUE** column of the *Contour Information window* that appears after completion of drawing an object. The value of ***concentration*** (or ***temperature***) must be set to a numeric value replacing the **$n/a** that appears for each new object; this is the value for any fluid already within the hydrogeologic unit or model and for any fluid that enters or leaves at this object.

When the Argus ONE interpretation scheme is set to its default for this layer, (**Exact Contour method**), point, open and closed contour objects in this layer will only affect nodes in the ***SUTRA Mesh*** layer that occur exactly over (and within) these objects. This requires that the user copy open and point contours to the ***Domain Outline*** layer, and that the mesh be regenerated, to force the mesh to have nodes exactly along these contours. Some nodes naturally fall within the area delimited by closed contours, and these should not be copied to the ***Domain Outline*** layer (this would create a hole in the mesh).

Setting a value of 1 or "true" for ***Specified Concentration.time_dependence*** (or ***Specified Temperature.time_dependence***) will cause node numbers in the input file for SUTRA created by the Argus-SUTRA GUI to be negative. This, in turn, causes SUTRA to refer to the subroutine BCTIME at the beginning of each time step to obtain the time-dependent values for each node with a negative node number. BCTIME must be user programmed to obtain the desired type of time variation as described in Voss (1984).

A limitation of this version of the Argus-SUTRA GUI is that the separation of nodes into groups, where each group has a different time-dependence user-programmed in BCTIME, is not possible, because SUTRA does not presently accept such group input. Thus, negative node numbers will have to be manually discovered by the user for the purposes of programming BCTIME after mesh generation if there is to be more than one group of time-varying specified concentration (or temperature) nodes.

HYDROLOGIC INITIAL CONDITIONS

This group of Information layers contains information on the complete spatial distribution of pressures (hydraulic heads), and solute concentrations (temperatures) at a given point in time, within the hydrogeologic unit being considered.

Initial Pressure/Hydraulic Head

The layer, ***Initial Pressure***, (or ***Initial Hydraulic Head***), contains information on the spatial distribution of pressure (or hydraulic head) at a given moment in time within the hydrogeologic unit to be modeled, or for generic studies, within the two-dimensional medium being modeled. The layer contains only one parameter, ***Initial Pressure.initial_pressure*** [M/(Ls^2)], (or ***Initial Hydraulic Head.initial_hydraulic_head*** [L]), which contains the pressure (or hydraulic head) values.

Initial Concentration/Temperature

The layer, ***Initial Concentration***, (or ***Initial Temperature***), contains information on the spatial distribution of solute concentration (or fluid temperature) at a given moment in time within the hydrogeologic unit to be modeled, or for generic studies, within the two-dimensional medium being modeled. The layer contains only one parameter, ***Initial Concentration.initial_concentration*** [C], (or ***Initial Temperature.initial_temperature*** [degC]), which contains the concentration or temperature values.

MAP OR POINT DATA LAYERS

MAP LAYER

The layer, ***Map***, is a special type of Argus ONE layer that contains an image of contours and points that cannot be modified or linked to other layers. The image in a Maps layer can serve as the pattern upon which information in other layers is superposed. Map layers can be used to visualize point data from Argus ONE Data layers using a variety of visualization tools provided in Argus ONE. Such visualization of point data is described in the section, "Visualizing Geospatial Information Linked to SUTRA Mesh Elements" (page 65). The post-processing visualization of pressure (hydraulic head), saturation, velocity vectors, and con-

centration (or temperature) for SUTRA simulations and provided by the SUTRA PIE is generated in a Map layer created by the PIE. The user may create additional Map layers, and one, ***Map***, is provided as a convenience in the Argus-SUTRA GUI.

Map information is only used as a visual basis for other Geospatial Information layers. Objects in map layers (lines, points and other objects) can be copied from the Map layer and pasted into Geospatial Information layers where attributes associated with the geospatial information can be assigned to these objects.

> For example, if the boundary line on a map is associated with a prescribed hydraulic head, the boundary line can be copied from the Map layer to the ***Specified Hydraulic Head*** layer. Information about copying and pasting objects in Argus ONE can be found in the Argus ONE User's Guide.

POINT DATA LAYER

The layer, ***Point Data***, is a special type of Argus ONE layer that contains scattered, gridded or meshed (point-wise) data which can be linked to the finite-element mesh or visualized in map layers. Data are most often imported into this layer type. Instructions on interpolating and contouring these data are given in the Argus ONE Users Guide (Supplement Version 2.5), and a brief description of this is given in the section, "Visualizing Geospatial Information Linked to SUTRA Mesh Nodes" (page 66). One Data layer, ***Point Data***, is provided as a convenience in the Argus-SUTRA GUI, and the user may create additional ones as needed.

CREATING SUTRA INPUT-DATA FILES AND RUNNING SUTRA

CREATING THE MESH AND RENUMBERING

When all data has been entered into the layers and the non-spatial information has been entered in the *SUTRA Non-Spatial Information dialog* (or at any time after completing input to the ***Domain Outline*** and ***Mesh Density*** layers, or alternatively to the ***Fishnet Mesh Layout*** layer) the finite-element mesh should be created as described in "SUTRA Mesh" (page 60) for irregular meshes, or as described in "Fishnet Mesh Layout" (page 56) for fishnet meshes.

After the mesh is then created, the discretization obtained may be evaluated by visualizing elementwise and nodewise mesh parameter distributions to see whether the desired spatial distributions are satisfactorily reflected in the mesh, as described in the section, "Visualizing Geospatial Information Linked to SUTRA Mesh Elements" (page 65), and, "Visualizing Geospatial Information Linked to SUTRA Mesh Nodes" (page 66). If needed, modifications may be made to the mesh density in the ***Domain Outline*** and ***Mesh Density*** layers (or to the ***Fishnet Mesh Layout*** layer), and the mesh regenerated as described above.

When the mesh is satisfactory, the user should optimize the bandwidth by renumbering the nodes.

1. Click on the **Special** pull-down menu along the top of the *Argus ONE window.*
2. Select **Renumber...**. this brings up the *Renumber window.*
3. In the *Renumber window*, select **Optimize Bandwidth**, and then **OK**.

To see the bandwidth value and other mesh information use menu **Navigation** along the top of the *Argus ONE window*, and select **Mesh Info...**. This may be done both before and after the optimization to see the change.

At present, the distribution of node numbers within the mesh cannot be controlled by the user and is selected by Argus ONE.

EXPORTING DATA AND RUNNING SUTRA

The SUTRA code requires two text files of input data as described in Voss (1984),

the "K1" file containing all of the simulation data except initial conditions,

the "K2" file containing initial conditions,

and a text file for file control,

the "K0" file containing a list of input/output file assignments, called, SUTRA.FIL.

These three files are automatically created by the Argus-SUTRA GUI as follows.

When the ***SUTRA Mesh*** layer is active in the *Argus ONE window*:

1. In the **PIEs** pull-down menu along the top of the window, choose **Run SUTRA**.

2. A dialog box appears with two options, **Create SUTRA Input Files** and **Create SUTRA Input Files and Run SUTRA**. The first option only creates the two text files. The second option creates these files and then executes SUTRA, after which control is returned to the *Argus ONE window*. (Also listed in this dialog box is an option to edit or check non-spatial project information; clicking the **Edit Params** button returns to the *SUTRA Non-Spatial Information dialog*.) After making a selection, click **OK**.

3. A file/directory dialog box will appear. The user should select the desired directory into which Argus ONE should place the SUTRA input files. Also the user must specify the desired "filename" which will be used as the prefix for these two filenames.

   The Argus-SUTRA GUI will create two files using this specified prefix: "**filename.d5**" and "**filename.d55**", which are the "K1" and "K2" files, respectively, as well as the file, **SUTRA.FIL**.

4. Click **OK** to start the export process.

   The moving diagonal stripes indicate that export is underway.

Some warnings or errors may appear during export. Some of these may require user corrections to the information in the project (see "Setting the Level of Error Checking Done for SUTRA Input Data" (page 95)). If a critical error is found that stops the export process, then the information is recorded in a file, **sutra.log**, to which the user should refer to correct the error.

If the user has chosen to also run SUTRA, another DOS window appears when export is completed that tracks the SUTRA code execution. When this window disappears, the SUTRA execution has terminated.

If problems have occurred in the SUTRA execution, the user should seek clues as to the cause of failure by inspecting the SUTRA output files: **SUTRA.ERR**, and **filename.d6**, as well as the input files, **filename.d5**, and **filename.d55**, using any available text editor outside of Argus ONE.

If desired, users familiar with the file structure and data input for SUTRA may use a text editor to view and edit the latter two SUTRA input files if minor (e.g. non-spatial data) changes are needed for additional simulations. Such simulations would need to be started from outside of the Argus ONE environment (see "Running SUTRA Outside of Argus ONE" (page 84). However, if Argus ONE export is quick, it may be easier to make these changes in the Argus-SUTRA GUI and then export the file again.

The file, **SUTRA.MET**, in the ArgusPIE\SUTRA directory is the export template used by Argus ONE to export and format the two text files required by SUTRA. The **SUTRA.MET** file refers to the project information and geospatial information linked to the finite-element mesh and exports this information to two files in exactly the format required for these SUTRA input files.

FILES CREATED BY SUTRA SIMULATIONS

When SUTRA is successfully executed following export from Argus ONE, four text files are created by the simulation. Depending on the prefix, "**filename**", selected by the user (see above) the files are:

filename.d6
filename.d66
filename.d3
filename.d4

The first two are the normal SUTRA output files "K3" and "K4" (see SUTRA revision in 1990) containing simulation output and restart data, respectively. The latter two are new files output by SUTRA, and are used by the Argus-SUTRA GUI for visualization of simulation results (see "SUTRA Output files for Post-Processing, d3 and d4" (page 91).

- The "**d3**" file contains the simulation results for all nodes in the mesh every NPRINT time steps, as specified in the *SUTRA Non-Spatial Information dialog*. Following some header lines required by both Argus ONE and the Argus-SUTRA GUI, the data are arranged in columns, from left to right, containing the node number, the node's X-coordinate, Y-coordinate, pressure (or hydraulic head), concentration (or temperature), and saturation.

- The "**d4**" file contains the velocity results for all elements in the mesh every NPRINT time steps, as specified in the *SUTRA Non-Spatial Information dialog*. Following some header lines required by both Argus ONE and the Argus-SUTRA GUI, the data are arranged in columns, from left to right, containing the element number, the element centroid X-coordinate, Y-coordinate, X-velocity component, and Y-velocity component. If velocity output was not user-requested in the *SUTRA Non-Spatial Information dialog*, this file exists but contains blank data.

In both the "**d3**" and "**d4**" files, results for each new time step are preceded by a header line.

RUNNING SUTRA OUTSIDE OF ARGUS ONE

When the Argus-SUTRA GUI runs a SUTRA simulation once, two files are created in the working directory (i.e. where the SUTRA input and output files are placed) that may be used to run SUTRA simulations from outside of the Argus user environment. These are:

- the file **sutra.fil**, which contains the file assignments for the SUTRA simulation and may be modified in any text editor, if necessary, before running SUTRA, and

- the file, **Runsutra.bat**, which may be executed to run the SUTRA executable. Alternatively, the full path of the **SUTRA.exe** file may be typed at a DOS prompt to start a simulation, if full paths to all input and out files are included in the **sutra.fil** file.

Note that if changes are also made to **d5** or **d55** input files for SUTRA using a text editor rather than the Argus-SUTRA GUI, then the Argus ONE project will not contain the revised information when the next export is done, and the revised information will be lost in the **d5** and **d55** files produced automatically.

POST-PROCESSING: VISUALIZING SUTRA SIMULATION RESULTS

Visualization of simulation results is an important part of the modeling process. What the user learns from inspecting the results guides choices in changing project and geospatial information. This is part of the iterative process of conducting the hydrogeologic analysis at hand. When simulations, based on visualization-motivated revisions to the model information, can be made conveniently and quickly, the analysis becomes much more effective.

For this purpose, the USGS SUTRA PIE provides the possibility of automatic visualization of simulation results in the Argus ONE application. The post-processing uses the "**filename.d5**" SUTRA input text file created by the Argus-SUTRA GUI and Argus ONE, and also uses the "**filename.d3**" and the "**filename.d4**" SUTRA output text files, both created by the SUTRA simulation.

STARTING THE POST-PROCESSOR

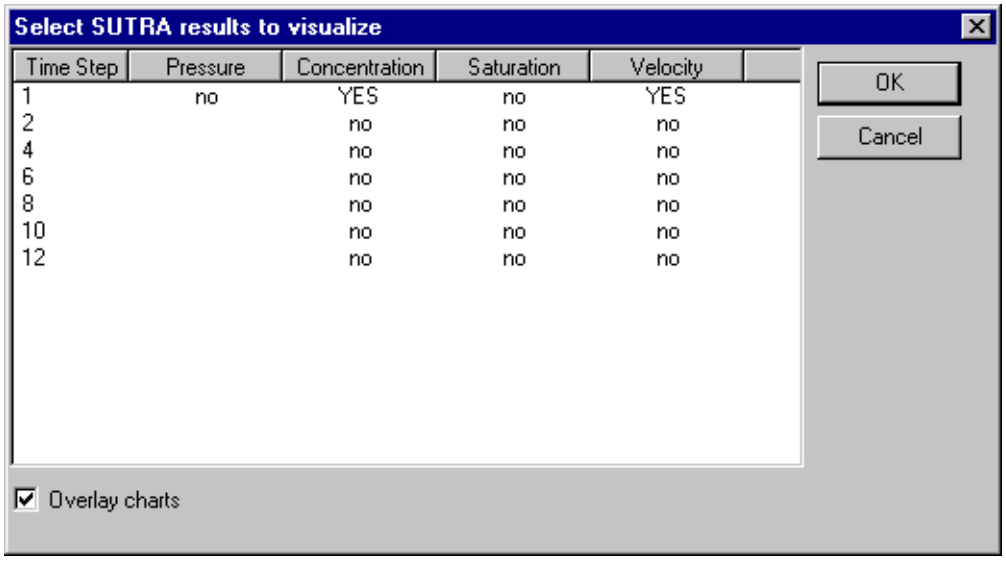


Figure 34. Plotting Dialog Window

To start the post processing visualization,

1. From the *Argus ONE window* menu **PIEs**, select **SUTRA Post Proc...**. This causes the *"Please select d5 file:" window* to appear.

2. Select the "**filename.d5**" file, associated with the SUTRA simulation output that the user wishes to view, associated with the current project.

   Argus ONE reads the file that is chosen and the *Plotting Dialog window* appears giving a listing of simulation results available in the "**filename.d6**" SUTRA output file (Figure 34). This window lists each time step available to plot and each of the SUTRA output variables that may be plotted for each time step.

3. Select the desired time step, and the variables to be plotted by clicking on the appropriate table cells changing **no** to **YES** (e.g. Pressure, Concentration, Saturation, Velocity). Clicking on a column header will toggle the entire column.

   The plot will overlay the model domain by default; to obtain separate small plots, deselect **Overlay** by clicking at the lower left corner of the window.

4. Clicking **OK** exits the window and the chosen contours and/or velocity vectors are plotted in the workspace (Figure 35).

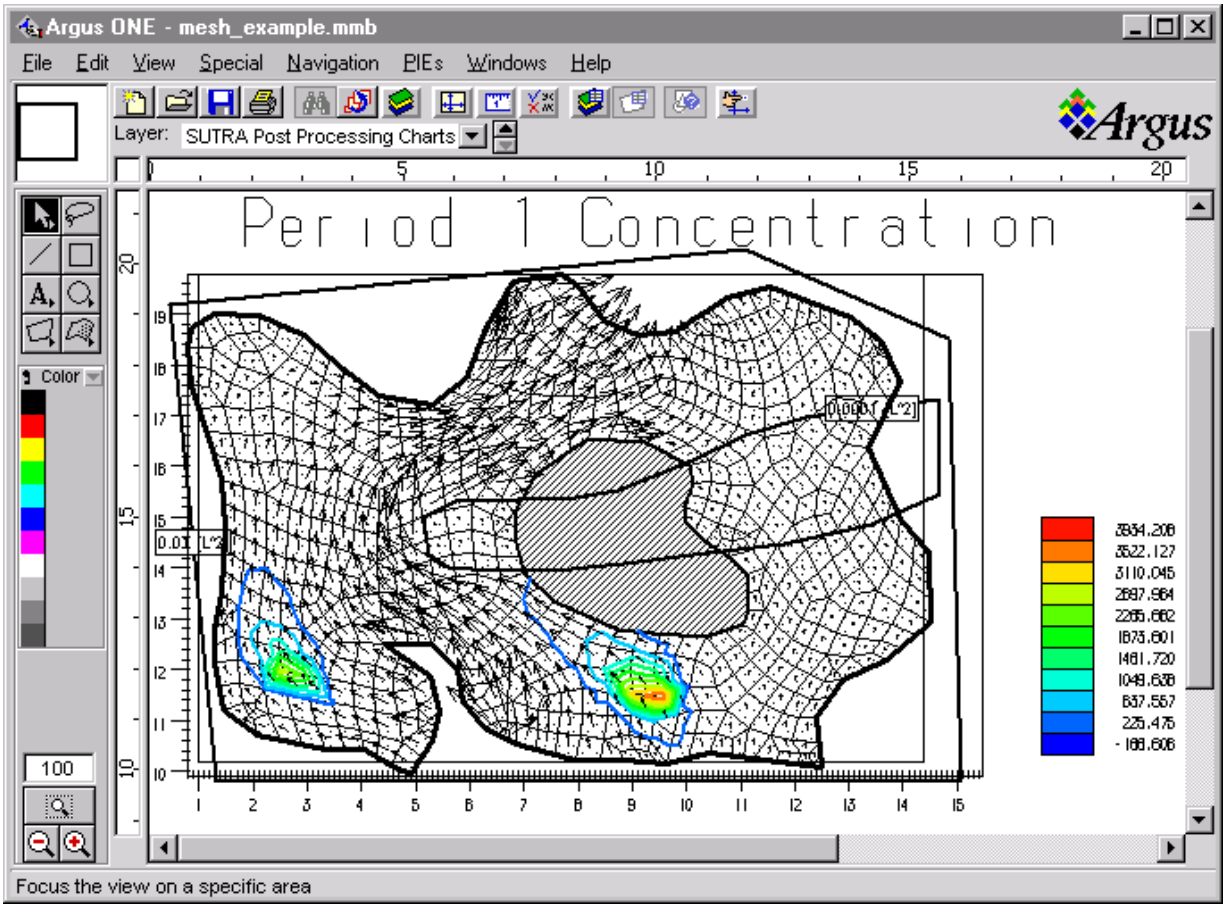


Figure 35. Post-Processing Graphics, Concentration and Velocity, example

The post processing step creates three additional Argus ONE layers: ***d3 Data***, ***d4 Data***, and ***SUTRA Post-Processing*** (a Maps layer).

- The ***d3 Data*** layer contains the values of the SUTRA output parameters at the nodal locations.
- The ***d4 Data*** layer contains the values of the X- and Y-velocity components at element centroid locations as output by SUTRA.
- The ***SUTRA PostProcessing*** layer contains the actual plots visualizing the user-selected simulation results. Each SUTRA output variable selected appears in an individual plot with a plot frame line, and these plots are superimposed on each other and the domain if "overlay" was selected.

If the post processing step fails, it is often due to a SUTRA simulation that has not terminated normally. See "Appendix B" (page 93) for troubleshooting help.

POST-PROCESSING PLOT MANIPULATION

The plot characteristics may be manipulated as follows.

1. Make the ***SUTRA PostProcessing*** layer active, and click just inside the plot frame line for the plot which is to be modified. This pops up a *Contour Diagram window* (or the *Vector Diagram window*) in which some modifications can be made. If both velocity and contour plots exist, some experimentation may be required to find the appropriate point on the screen to access each plot's controls.

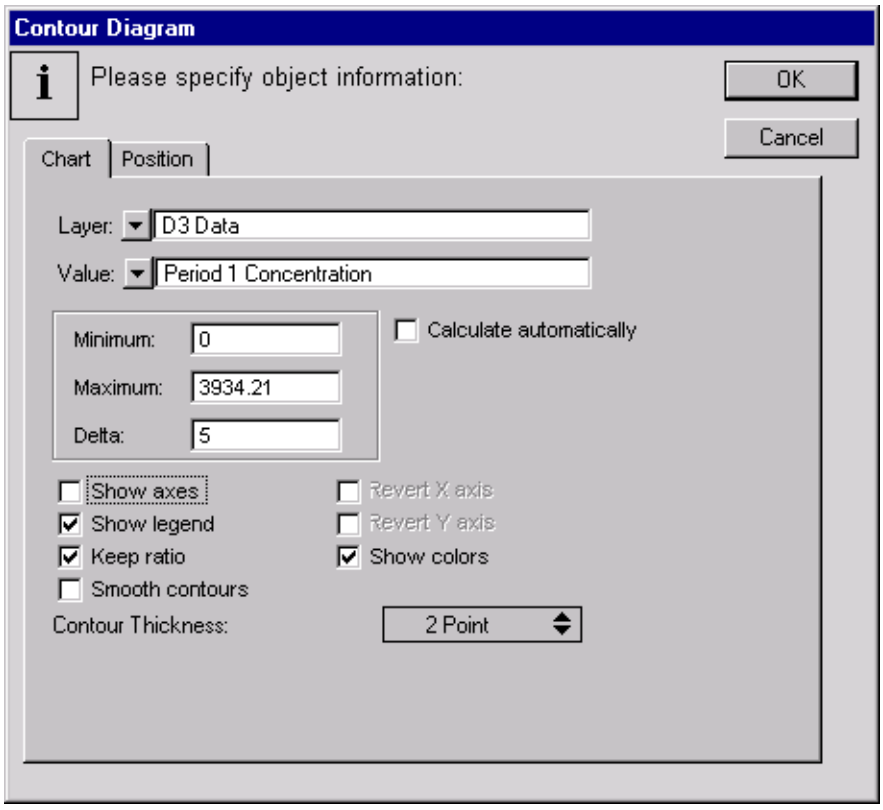


Figure 36. Contour Diagram Window

2. For example, in the *Contour Diagram window* (Figure 36), the contours may be changed by setting the desired **Minimum**, **Maximum** and **Delta** for the plotted contours. The data range may be obtained by clicking **Calculate Automatically** on and off.

3. To thicken the contours, select **Contour Thickness**, providing a pull-down menu of thicknesses, and select **3**. A useful effect is to set **Delta** to be one or two orders of magnitude smaller than the number automatically selected, creating a set of overlapped contours which shows as rainbow-color-filled areas.

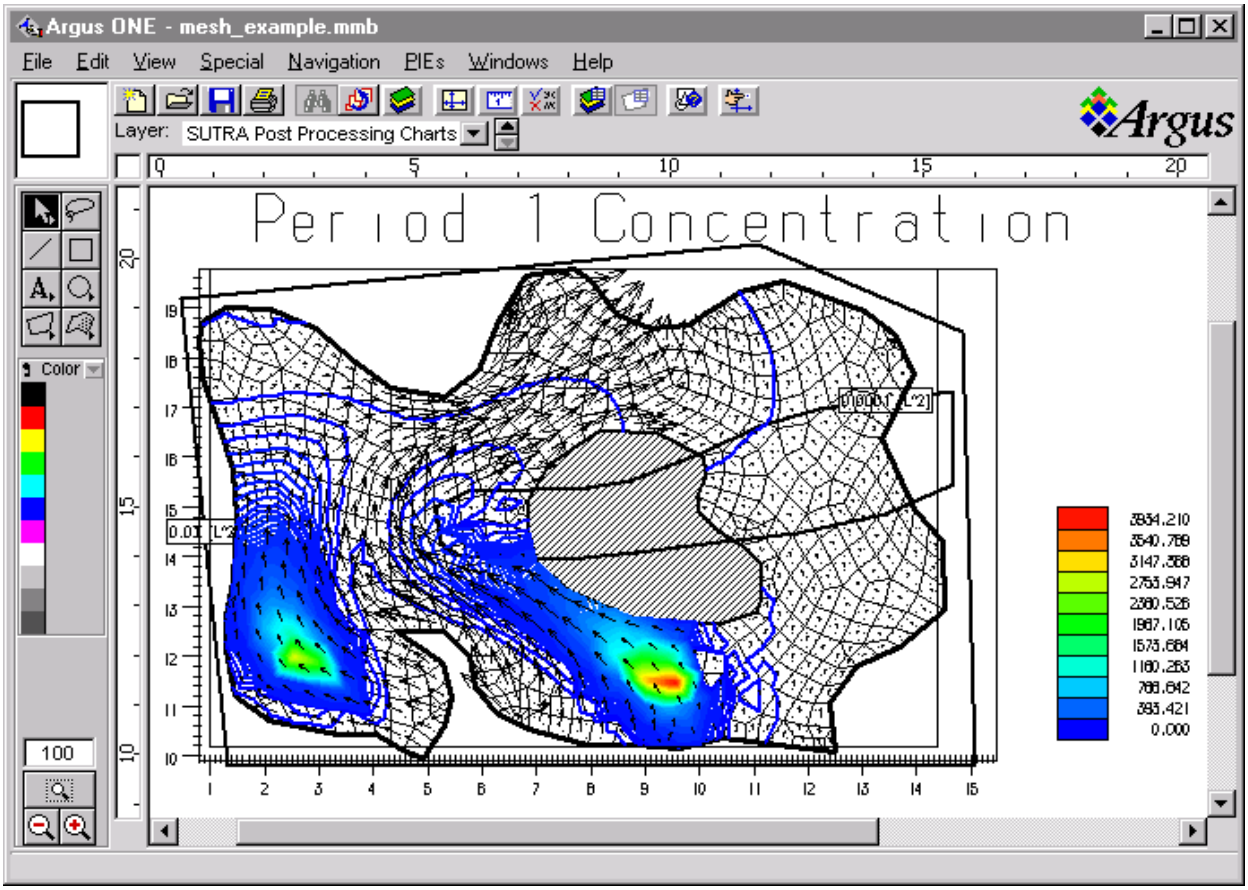


Figure 37. Concentration Contours with Small Delta, example

A modified *Contour Diagram window* is shown in Figure 36, and the modified plot is shown in Figure 37. A velocity vector plot may be similarly modified.

EXPORTING POST-PROCESSING PLOT GRAPHICS

Better manipulation of these plots, particularly for preparation of publication graphics is possible, but this is under development. At present, to generate publication-ready graphics, it is suggested that the following method be used.

The user should "print" the ***SUTRA PostProcessing*** layer to a file, by selecting **Print**, found in the **File** menu, and then selecting **Write To File** in the *Print/Write Dialog window* that appears. This produces a postscript file with the filename selected by the user in this dialog. This "ps" file may be imported into various graphics packages and manipulated. It may be necessary to convert the "ps" file to "eps" (encapsulated postscript) before importing. Neither the conversion program nor the graphics package is provided at this time by Argus ONE. Note that this method assumes that the printer selected uses a Postscript driver.

REFERENCES CITED

Argus Interware, 1997a, Argus Numerical Environments and MeshMaker User's Guide, <url: http://www.argusint.com/Download.html#olguide>, 519p.

Argus Interware, 1997b, Argus Numerical Environments Plug-In Extensions, Argus-PIEs Technical Notes, <url: http://www.argusint.com/Download.html#olguide>, 31 p.

Harbaugh, A. W., and McDonald, M. G., 1996, User's documentation for MODFLOW-96, An update to the U.S. Geological Survey modular finite-difference ground-water flow model: U.S. Geological Survey Open-File Report 96-485, 56 p.

Konikow, L.F., Goode, D.J. and Hornberger, G.Z., 1996, A three-dimensional method-of-characteristics solute-transport model (MOC3D), U.S. Geological Survey Water-Resources Investigations Report WRIR 96-4267, <url: http://water.usgs.gov/software/moc3d.html>, 87 p.

Shapiro, A. M., Margolin, J., Dolev, S. and Ben-Israel, Y., 1997, A graphical-user interface for the U.S. Geological Survey modular three-dimensional finite-difference ground-water flow model (MODFLOW-96) using Argus Numerical Environments: U.S. Geological Survey Open-File Report 97-121, <url: http://h2o.usgs.gov/software/modflow-gui.html>, 50 p.

Voss, C.I., 1984, SUTRA - Saturated-Unsaturated Transport: 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 WRIR 84-4369 (revised 1990, 1997), 409 p.

APPENDIXES

APPENDIX A

SUTRA OUTPUT FILES FOR POST-PROCESSING, D3 AND D4

SUTRA can optionally create two special output files that can be used for post-processing by Argus ONE or other programs. This requires use of SUTRA, version V09972D or later. The SUTRA file unit "K5" is assigned to file **d3**, and SUTRA file unit "K6" is assigned to file **d4**.

- The "**d3**" file contains the simulation results for all nodes in the mesh every NPRINT time steps, as specified in the *SUTRA Non-Spatial Information dialog*. Following some header lines required by both Argus ONE and the Argus-SUTRA GUI, the data are arranged in columns, from left to right, containing the node number, the node's X-coordinate, Y-coordinate, pressure (or hydraulic head), concentration (or temperature), and saturation.

- The "**d4**" file contains the velocity results for all elements in the mesh every NPRINT time steps, as specified in the *SUTRA Non-Spatial Information dialog*. Following some header lines required by both Argus ONE and the Argus-SUTRA GUI, the data are arranged in columns, from left to right, containing the element number, the element centroid X-coordinate, Y-coordinate, X-velocity component, and Y-velocity component. If velocity output was not user-requested in the *SUTRA Non-Spatial Information dialog*, this file exists but contains blank data.

There is a header section at the beginning of **d3** and **d4** files that specifies the SUTRA output titles (two lines), the type of information contained in the file (nodewise or elementwise), the number of nodes or elements, the number of time steps available, and a table of elapsed time, existence of head (or pressure), concentration (or temperature) for the time step, and time step. Existence is marked by a 1 (does exist) or a 0 (does not exist).

The existence of various results is pre-calculated by SUTRA before completing simulation of all time steps. In the event that the simulation stops prematurely (i.e. for non-convergence of an iterative solution) the header will be incorrect. See "Plotting a Non-Converged Solution" (page 94), in this case.

In both files, results for each time step are followed by a blank line and results for each new time step are preceded by three header lines specifying the elapsed time, the time step number, and the column headings.

An example of the header format for a **d3** file:

```
## SUTRA Model created using ArgusONE
## USGS example
## Nodewise Results
##     123 nodes
##    4 Time Steps printed
##    Current Time   Calculated
##      in Seconds   Head  Conc  Time step
##      1.0000D+00   1     1     1
##      2.0000D+00   1     1     2
##      3.0000D+00   1     0     3
##      4.0000D+00   1     1     4
## Elapsed Time     1.0000D+00 Seconds
## Time Step     1, Duration      1.0000D+00 Seconds
##  Node              X              Y       Head      Concentration     Saturation
       1  1.2835616E+01  1.6661957E+01  3.5785491E+00  1.6645971E-97  1.0000000E+00
       2  1.3687778E+01  1.6707639E+01  3.1634323E+00  7.1279230-106  1.0000000E+00
       3  1.2967556E+01  1.7456813E+01  1.9738776E+00  7.4044157E-98  1.0000000E+00
```

An example of the header format for a **d4** file:

```
## SUTRA Model created using ArgusONE
## USGS example
## Velocity Results
##     102 elements
##    4 Time Steps printed
##    Current Time   Calculated
##      in Seconds   Head  Conc  Time step
##      1.0000D+00   1     1     1
##      2.0000D+00   1     1     2
##      3.0000D+00   1     0     3
##      4.0000D+00   1     1     4
## Elapsed Time     1.0000D+00 Seconds
## Time Step     1, Duration      1.0000D+00 Seconds
## Element       X origin       Y origin     X velocity     Y velocity
       1    3.7427179E+00  1.8345237E+01  0.0000000E+00  0.0000000E+00
       2    4.6319645E+00  1.8370438E+01  0.0000000E+00  0.0000000E+00
       3    5.5364395E+00  1.8326056E+01  0.0000000E+00  0.0000000E+00
```

The post-processing program reads the header information in these files in the following manner.

1. The first 4 lines are skipped.

2. The first two characters of the current line are verified to be "##" and then skipped.

3. The next number encountered is taken to be the number of periods
   ## 2 Time Steps printed

4. The next two lines are skipped.

5. For the number of periods, time, Head and Concentration flags, and the time step number are read for a **d3** file, and velocity vectors are read for a **d4** file.

APPENDIX B

TROUBLESHOOTING FAILED SUTRA SIMULATIONS

If the SUTRA simulation seems to work, and the post-processing step complains that there is something wrong with the **d3** or **d4** files, then it is not possible to obtain plots of simulation results. Often, this is caused by a failed SUTRA simulation due to missing values or errors in some of the Argus-SUTRA GUI layers as entered by the user. Sometimes these problems are flagged by error messages that appear on the screen during export. However, when the source of error is unclear, the following approach should help to determine it.

1. In any text editor available, check the end of the **d6** file output created by the SUTRA run to determine whether or not the simulation has terminated normally. If simulation was successfully completed, then the last line states this:

   ```
   SUTRA SIMULATION TERMINATED AT COMPLETION OF TIME STEPS (or PERIOD)
   ```

2. Most likely, this statement will be missing and the simulation did not terminate normally. In this case, scan backward in the file to find the last text output to detemine the last successful step of the simulation.

3. Then open the **d5** SUTRA input file created by the Argus-SUTRA GUI in a text editor. Scan this file from the top to see if there are any "$N/A" or "NAN" entries in the file. If so, these indicate that data was missing from the GUI layer that provides this data, and these have caused the simulation to fail. The type of data missing may be determined by reference to the SUTRA manual's list of input data.

4. In the Argus-SUTRA GUI, correct the missing or incorrect data, and re-export and run SUTRA again.

APPENDIX C

PLOTTING A NON-CONVERGED SOLUTION

Sometimes it is useful to troubleshoot non-convergent solutions from SUTRA by visualizing the results completed just before the non-convergent simulation is terminated. To plot SUTRA results from a non-converged iterative solution, the user must make some modifications to the header lines in the **filename.d3** and **filename.d4** files to report the actual number of time steps that exist in the file before non-convergence stopped the SUTRA simulation. The modifications may be made with any text editor available.

The header of the two files is created before the simulation by SUTRA and the anticipated time steps that should appear in the SUTRA output are listed. When the simulation does not converge at a given time step, no other steps are output by SUTRA, although they are listed in the header. Thus, the user must remove the non-existent time steps.

After opening the files in an editor, replace the number on line 5 with the number of time steps actually printed in the **d6** file. The lines to be modified state:

"## 2 Time Steps printed".

Finally, delete the lines from the time-step list (starting three lines below), for the time steps that were not printed.

APPENDIX D

SETTING THE LEVEL OF ERROR CHECKING DONE FOR SUTRA INPUT DATA

By default there are a series of consistency checks done during the export of data for the SUTRA simulation. Some checks may result in warnings if there seems to be a condition which could result in an invalid simulation. Other checks report errors that will cause SUTRA to be unable to run. In case of such a critical error, a **sutra.log** file is generated that contains information about the error. The level of error reporting can be customized by setting a global variable in the project.

1. From the menu **Edit** select **Edit Global Variables...**

2. In the *Global Variables dialog*, click on **Add**.

3. Create a variable named "Alert" and set the value accordingly:
   0 - show all alert messages
   1 - turn off warnings, alert messages that may cause problems
   2 - turn off alert messages on conditions that will certainly produce invalid model results

APPENDIX E

ADDING AND LINKING NEW LAYERS

Two examples presented below which demonstrate a few aspects of adding and linking new layers. The first example creates an areal fluid source from a precipitation map and the second defines the thickness of the aquifer from top and bottom elevations.

Adding a Precipitation Data Layer and Linking It to Fluid Sources Layer

1. Bring up the *Layer List window* by clicking on the **Layers...** button along the top of the Argus ONE window.

2. Click on the **Layers** button at the top of the *Layer List window*. This brings up the *Layers dialog* (see Figure 33).

3. To create the new layer (which will contain the areal distribution of rainfall), use the down arrow to position the highlighted layer to ***Sources of Fluid***, and click on button **New** below the label "Layer" (not below the label "Parameter, which would add a parameter to layer ***Sources of Fluid***).

4. Set the layer name by typing "Rainfall" to replace the default layer name, "New Layer".

5. Change the **Type** to **Data** (from the default of **Information**) by clicking on the word "Information", which will bring up a menu of choices.

6. Before the ***Rainfall*** layer can be linked to the fluid sources layer, data needs to be imported into the new layer. See the Argus User's Guide for information on importing data. A number of parameters may be associated with the imported data; assume that the first parameter contains the rainfall amounts in units [m/year].

7. To link the imported information, click on the ***Sources of Fluid*** layer line to highlight it.

8. In the parameter listing area, highlight the line containing **specific_source**, then press and hold the mouse button below the label "Value" near the arrow pointing downwards. A button labeled "Expression..." appears. Move the mouse over this button and release.

9. In the *Expression dialog* scroll the left scroll bar until you can see an entry for layer, ***Rainfall***.

10. Click on entry ***Rainfall***. The parameter names imported into the layer will then appear in the box to the right.

11. The first parameter, "Imported Parameter 1" contains the rainfall values for this example. Double-click on the entry in the right-hand box, and the parameter name will appear in the expression box at the top of the *Expression dialog*.

12. To convert the value to [m/s] append the expression "/(365.25*86400)" to divide the rainfall by the number of seconds in a year. To indicate that only 25% of the rainfall becomes ground-water recharge, click to the left of the parameter name and type "0.25*".

13. Click button **OK**.

14. If the user plans to add point or contour objects to the ***Sources of Fluid*** layer in addition to the recharge from rainfall, the interpretation method listed at the bottom of the *Layers dialog* must be set to **Exact Contour method**. Any other interpretation method will cause the objects to override the distributed rainfall source everywhere in the layer.

15. In the *Layers dialog* click **Done**. The default value of the parameter ***Sources of Fluid.specific_source*** is now set to the recharge calculated from the rainfall distribution.

Calculating a Thickness From Two New Layers or Two New Parameters

In the following dual example, it is shown how thickness may be calculated in two ways. In the first, two new layers are added, the elevations for the top and bottom of the aquifer. These layers are then linked to the thickness layer which calculates the aquifer thickness. In the second example, two new parameters are instead added to the ***Thickness*** layer. The first example follows:

1. Bring up the *Layer List window* by clicking on the **Layers...** button along the top of the Argus ONE window.

2. Click on the **Layers** button at the top of the *Layer List window.*

3. In the *Layers dialog* that appears (see Figure 33 on page 68), use the down arrow to position the highlighted layer to be ***Thickness***, and click on button **New** below the label **Layer** (not below the label **Parameter**, which would add a parameter to layer ***Thickness***).

4. Set the layer name by typing "Top" to replace the default layer name, "New Layer". This layer will be used to set the elevation of the top of the aquifer. The default interpretation type is **Nearest Contour**.

5. Add a second new layer and name it "Bottom".

6. Click on the ***Thickness*** layer line to highlight it.

7. In the parameter listing area, press and hold the mouse button below the label **Value** near the arrow pointing downwards. A button labeled **Expression...** appears. Move the mouse over this button and release.

8. In the *Expression dialog* scroll the left scroll bar until you can see an entry for layer ***Top***.

9. Click on entry "Top". That entry will then appear in the box to the right.

10. A list of parameters of this layer appears in the right hand box. In this case there is only one parameter, "Top". Double-click on the entry in the right-hand box, and the layer (and implicitly for single parameter layers, the parameter "Top" in layer ***Top***) name will appear in the expression box at the top of the *Expression dialog*.

11. Insert a minus sign after "Top" (see Figure 38).

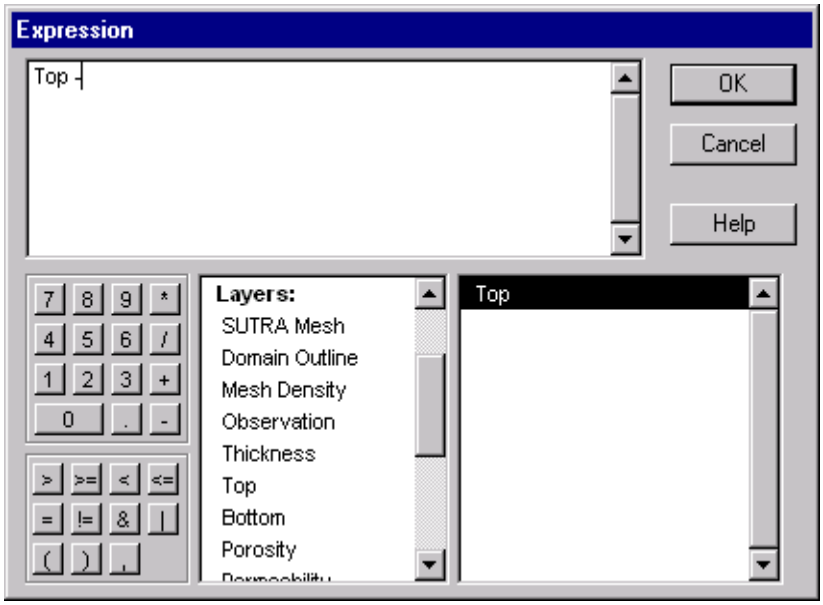


Figure 38. Constructing an Equation in the Expression Dialog, example

12. Repeat step 10. to add layer ***Bottom*** to the expression. You should now have the expression "Top - Bottom" in the upper box.

13. Click **OK**.

14. If the user plans to add point or contour objects to the ***Thickness*** layer in addition to the thickness calculated from top and bottom elevations, the interpretation method listed a the bottom of the *Layers dialog* must be set to **Exact Contour method**. Any other interpretation method will cause the objects to override the calculated thickness everywhere in the layer.

15. In the *Layers dialog* click **Done**.

From now on elevations of objects in layer ***Bottom*** will be subtracted from elevations in layer ***Top*** to calculate values for layer ***Thickness***.

Note: Objects with specified thickness value drawn directly into layer ***Thickness*** would override the calculation.

The Argus ONE User's Guide provides details of adding Information layers and linking Information layers to each other using mathematical expressions.

In the second example, thickness may be calculated from "Top" and "Bottom" information contained in a single layer. In this case it would be necessary to add two parameters to the ***Thickness*** layer as follows.

1. Bring up the *Layer List window* by clicking on the **Layers...** button along the top of the Argus ONE window.

2. Click on the button **Layers** at the top of the *Layer List window.*

3. In the *Layers dialog* that appears (see Figure 33 on page 68), use the down arrow to position the highlighted layer to be "Thickness", and click on button **New** below the label "Parameter" twice (not below the label "Thickness, which would add a layer). Each click creates a new parameter.

4. Set the first parameter name by typing "Top" to replace the default parameter name after clicking on "New Parameter". This parameter will be used to set the elevation of the top of the aquifer.

5. Set the second parameter name by typing "Bottom" to replace the second default parameter name after clicking on "New Parameter 1". This parameter will be used to set the elevation of the bottom of the aquifer.

6. In the parameter listing area, click on the line containing parameter "thickness" in the list found on the lower part of the dialog. Then press and hold the mouse button below the label "Value" near the arrow pointing downwards in the line. A button labeled "Expression..." appears. Move the mouse over this button and release.

7. In the *Expression dialog* scroll the left scroll bar until you can see an entry for layer "Thickness" and click on it. Click on entry "Top". That entry will then appear in the box to the right.

8. A list of parameters of this layer appears in the right hand box. In this case there are three parameters. Double-click on the "Top" entry in the right-hand box, and the parameter "Top" will appear in the expression box at the top of the *Expression dialog*.

9. Insert a minus sign after "Top".

10. Repeat step 8. to add parameter "Bottom" to the expression. You should now have the expression "Thickness.top - Thickness.bottom" in the upper box.

11. Click **OK**.

12. In the *Layers dialog* click **Done**.

From now on elevations of objects in parameter "Bottom" will be subtracted from elevations in parameter "Top" to calculate values for parameter "Thickness". This method somewhat restricts the flexibility of the distributions possible for "Top" and "Bottom" as they are both defined as parameters of the same graphic objects. For the previous example, where new layers were defined, the contours of "Top" and "Bottom" could, in effect, cross one another.

APPENDIX F

CURRENT LIMITATIONS

There are a number of limitations of the Argus-SUTRA GUI that users will find. Some of these will disappear as new versions of the Argus ONE software and the Argus-SUTRA GUI are released. This list will be updated. At present (**Argus** ONE **version 4.0, Argus-SUTRA GUI version 1.0**), the limitations are as follows:

Illegal quadrilateral elements generated

Occasionally, the quadrilateral mesh generator will give triangular elements with a node along a flat side, or will overlap or leave space between elements. Usually, a small change to mesh density or location of the domain boundary will eliminate this problem. However, the user is requested to save the "mmb" project file containing the error and contact Argus with the information, so that improvements can be made in the meshing engine.

No contour intersection allowed in the *Domain Outline* layer

In many problems, particularly in cross sections, it would be desirable for geologic units to have a distinct top and bottom with finite elements lined up along these boundaries. In order to do this, the tops and bottoms must be drawn in the ***Domain Outline*** layer, and these would need to intersect the outside boundaries of the domain. At present, such intersection is not allowed in Argus ONE for the irregular mesh. A Fishnet Mesh may be used to avoid this limitation.

No variable properties along open contours

Argus ONE does not allow specification of a parameter variation along a contour (although this limitation may, in effect, be overcome in some cases, e.g. see "Henry Seawater Intrusion Problem with Variable-Density Flow" (page 19)). This would be desired, for example, along an open contour representing a river with varying elevation as a specified head boundary. At present, the user must break the contour into pieces, each having a constant value of the parameter.

Minimal Graphics Export Capability.

Postscript files may be created by printing to a file (if the printer selected is a Postscript printer).

Spatial system is not georeferenced

Users of other available GIS systems such as ARC/Info™ may perceive a limitation in that coordinates of imported datasets are taken exactly as specified in the imported files, and no georeferencing is done by Argus ONE. Argus ONE does not use latitude-longitude coordinates, and these must be converted to some consistent length units such as feet, meters, or miles before import. No cartographic projections are available within Argus ONE.

Minimal Graphics Export Capability

The only output option for graphics is printing; Postscript files can be generated by printing to a file.

No user option for warnings during data export

Argus ONE does not provide an error window option to ask the user whether or not to continue after an error has been detected. As a result there is no way to keep Argus ONE from exporting the SUTRA datasets and attempting to run SUTRA after non-fatal errors have been detected.

There is some ability to configure which warnings will be displayed, see "Setting the Level of Error Checking Done for SUTRA Input Data" (page 95).

No grouping of time-variable boundary conditions or sources

SUTRA does not accept grouping of nodes for the purposes of assigning different time variations to each group in SUTRA subroutine BCTIME. All time-dependent nodes are passed to the SUTRA input files in one group. The user must separate them manually if more than one type of time variation is required. This is not, strictly speaking, a limitation of the GUI, but of the SUTRA program itself that is highlighted by the GUI.

No multiple sources at a single location

SUTRA allows only one fluid source and only one solute/energy source per node. Multiple sources must be summed by the user before input to the GUI. This is not, strictly speaking, a limitation of the GUI, but of the SUTRA program itself that is highlighted by the GUI.

Node numbering cannot be controlled by user

Regular rectangular meshes are numbered automatically by Argus ONE in a non-uniform pattern of node numbers resulting in non-optimal band widths, even after automatic band-width optimization. For such meshes and other meshes, it may be desirable for the user to assign the location and pattern of node numbering, both to minimize the band width and to allow for easier user recollection of node number locations for purposes of modeling.

Contour vertices cannot be placed exactly during interactive input

When inputting point, open or closed contours, it is required that the user edit a data file that will be imported into Argus ONE in order to place contour vertices at exact locations. At present, numerical entry of vertex coordinates cannot be done through text input while entering such objects interactively.

Symbols

$n/a, unset numerical value 76, 93
.d3 file suffix 83
.d4 file suffix 83
.d6 file suffix 83
.d66 file suffix 83
.dll file suffix 6
.err file suffix 83
.fil file suffix 84
.ini file suffix 6
.lay file suffix 6
.log file suffix 82
.met file suffix 6
.mmb, file suffix 41
.val file suffix 6

A

Activating a layer 43
Active layer 43
Adding or Changing Layers 52
ALMAX, SUTRA variable 63
ALMIN, SUTRA variable 63
ANGLEX, SUTRA variable 63
Anisotropy direction 72
ArcView Shape files 7, 43, 54, 56
Area sources 74
Areal model, selecting 28
Argus Interware 1, 3, 5
Argus Open Numerical Environments (Argus ONE) 1
ATMAX, SUTRA variable 63
ATMIN, SUTRA variable 63
AutoCAD, DXF format 7

B

Background value, defining 68
Background value, variable 21, 68
Bandwidth of mesh, optimizing 11, 81
Band-width optimization 11
BCTIME, SUTRA subroutine 75, 77, 78, 79, 80
Boundary conditions 9, 78
Boundary conditions, time-dependent, FORTRAN compiler needed 6

C

C++ 4, 5, 6
Canal 74
concentration_of_source 75, 76
Contour data, using for input 50
Contour Diagram dialog 18
Contour drawing tool 55
Contour Information dialog 9, 10
Contour plot 87
Contour vertices, exact placement 20
Contours, by percent 23
Contours, copying to another layer 21, 22
Contours, editing in text editor 20
Coordinates, exact entry 60
Creating a domain outline 54
Creating a Fishnet mesh 57
Creating an irregular mesh 60
Creating SUTRA input-data files 81
Cross-sectional model 19
Cross-sectional model, mesh 60
Cross-sectional model, selecting 28
Cylindrical thickness, selecting 28

D

d3 Data layer 86
D3 file 83, 86, 91
d4 Data layer 86
D4 file 83, 86, 91
D5 file 83
D55 file 83
D6 file 83
D66 file 83
Data layer 44, 69, 81, 86
Datum 41
Default interpretation types 46
Default value, parameters 70
Default Values for SUTRA for Special Cases 31
Default Values for SUTRA Model for General Case 30
Default values, non-spatial information 29
Deleting layers created by GUI 53
DELT, SUTRA variable 18
Density layer 44
Dipped model, selecting 28
Dispersion, classical 73
Dispersion, flow-direction-dependent 72
Dispersivity layer 23, 72
Dissolving minerals 77
DLL file 4
Domain layer 44
Domain Outline layer 10, 53
Drawing a contour 9, 10
Drawing size 20, 40
DXF files 7, 43, 54, 56
Dynamically linked library, DLL 4

E

Editing project information 18
Element shape, elongated 60
element_size parameter and element sizes 56
Elementwise distributions, visualizing 65
Elementwise parameters, list 46, 63

Elongated elements 60
Energy transport model, selecting 28
Equation in Expression dialog 22
Error checking, setting level of 95
Exact Contour method 16, 48
Exact entry of coordinates 60
Exaggerated view of workspace 41
Examples, step by step 13, 15, 18, 19
Exporting 11, 13, 20
Exporting a contour to a file 20
Exporting a Fishnet mesh 59
Expression dialog 22, 52, 68
Eye 8, 10

F

Failed SUTRA Simulations 93
Files Created by SUTRA Simulations 83
Fishnet Mesh Layout layer 20
Fishnet mesh, creating 57
Fishnet mesh, selecting 28
Fishnet meshes 4, 7, 56
Fluid Properties 37
FORTRAN 6, 32, 73, 77, 78

G

Generating an irregular mesh 11
Georeferencing, availability 100
Geospatial information 4, 5, 7, 8, 67
GNUP, SUTRA variable 18
Gravity vector 28, 39
GRAVY, SUTRA variable 20
Ground-water flow model, areal, example 8
Groups list 46

H

Headings, SUTRA model 32
Heat production 74
Hydraulic Conductivity layer 15, 72
Hydraulic head as a variable, selecting 28
Hydraulic Head layer 78
Hydrologic Boundaries 78

I

Importing information 7, 40, 43, 59
Inactive layers 43
Information layer 44
Initial Concentration layer 80
Initial Conditions, hydrologic 80
Initial Hydraulic Head layer 80
Initial Pressure layer 80
Initial Temperature layer 80
INOB, SUTRA variable 64
Installing Argus ONE 6
Interface-Configuration Window 27
Interpolation Contour method 50
Interpretation methods 67
Interpretation of contours and data 41
Interpretation of Contours and Point Data 45
Interpretation type, defaults 46
InvDistSq Contour method 52
Irregular mesh, creating 60
Irregular mesh, selecting 29
Irregular meshes 4, 7
Iteration 36

K

K1 SUTRA input file 29, 82
K2 SUTRA input file 82

L

Lake as an areal source 74
Layer Floater 8, 41
Layer groups 42
Layer List window 8, 21, 41, 42
Layer names, naming convention 45
Layer names, warning about changing 53
Layer parameters 45
Layer Types 43
Layer visibility 42
Layers dialog 21, 43, 68
Layers, adding or changing 52
Layers, linking new 96
Layers, list of 46
Line or curve sources 74
Linking 44, 48
Linking new layers 96
Links, SUTRA Mesh parameters 63
LREG, SUTRA variable 63

M

Magic wand 11
Magma body 74
Map layer 44, 80
Maps, displaying 80
Mesh Density layer 53, 56
Mesh type, selecting 28
Modes, Numerical Controls 33
Modification of GUI 1
Modifying plots 18

N

Naming convention, layer names 45, 61
Naming SUTRA input files 12
NAN, not a number 93
Nearest Contour method 49
New SUTRA Project, menu option 7
NN2D Contour method 51
Nodewise distributions, visualizing 66

Nodewise mesh parameters, list 46
Nodewise parameters,
creating contour map of 66
Nodewise parameters, list 63
Non-uniform scaling 41, 60
NPRINT, SUTRA variable 18, 20
NREG, SUTRA variable 63

O

Objects, adding objects to linked layers 48
Observation layer 67
Optimize Bandwidth 81
Output 35
Over-ride values, clearing 22

P

Parameter names, warning about changing 53
Parameters list 46
Parameters of a layer 45
Parameters, default values 70
Path for Argus-SUTRA GUI files 6
Path for SUTRA files 6
PBC, SUTRA variable 64
Permeability layer 23, 72
PIE, Plug-In Extension 3, 4, 5, 89
Plots, displaying in Map layer 80
Plots, modifying 14, 18
Plots, removing 13
Plots, saving 19
Plots, selecting 14
Plotting contours 12
Plotting velocity vectors 12
PMAX, SUTRA variable 63
PMIN, SUTRA variable 63
Point Data layer 81
Point sources 74
POR, SUTRA variable 63
Porosity layer 23, 72
Post-processing 6, 7, 12, 84
Post-processing layers 86
Post-processing, troubleshooting 93
Pressure as a variable, selecting 28
Production of energy or solute 39
Project Information Dialog 18, 25
pUBC, SUTRA variable 62, 64
Public domain code 4, 5
PVEC, SUTRA variable 64

Q

QIN, SUTRA variable 62, 63
QINUIN, internal variable 75
Quadrilateral Finite-Element Mesh Layer 44
QUIN, SUTRA variable 62, 64

R

Radial cross-section, selecting 28
Radiogenic heat production 74, 77
Rainfall 74
Recharge 74
Reference elevation 41
Renumber nodes 11, 81
RESULTANT_FLUID_SOURCE,
internal variable 75
RESULTANT_SOLUTE/ENERGY_SOURCE,
internal variable 77
Retrieving an Argus ONE-SUTRA project 13, 41
River 74
Running SUTRA 13, 82
Runsutra.bat file 84

S

Saturated flow model, selecting 28
Saving an Argus ONE-SUTRA project 11, 41
Scale and units 40
ScanN2D Contour method 52
Scattered data, using 69
Seawater intrusion model 19
Select SUTRA results to visualize window 18, 19
SHAPE files 7
Shape, ArcView files 43, 54, 56
SIGMAW, SUTRA variable 20
Simulation problem type, selecting 27
Simulation results for all nodes,
SUTRA output file d3 83
Simulation type, choosing 7
Solid Matrix, Adsorption 38
Solute transport model, areal, example 13
Solute transport model, selecting 28
Solute transport model,
transient areal, example 18
Source, line 21
Sources of Energy layer 77
Sources of Fluid layer 21, 75
Sources of Fluid.time_dependence 75
Sources of Solute layer 77
Sources of Solute/Energy.time_dependence 77
Sources, area 74
Sources, areally-distributed 74
Sources, concentration 75
Sources, line 74
Sources, multiple 74, 76
Sources, point 74
Sources, specific 74
Sources, specifying 21
Sources, temperature 75
Sources, total 74

Specific source, solute or energy 77
specific_source 74, 75, 76
Specified Concentration layer 79
Specified Hydraulic Head 9, 13
Specified Hydraulic Head.time_dependence ... 78
Specified Pressure layer 21, 78
Specified Pressure.concentration 78
Specified Pressure.temperature 78
Specified Pressure.time_dependence 78
Specified Temperature layer 79
Specifying concentration or temperature of inflowing fluid at specified head 78
Specifying hydraulic head 9, 78
Specifying pressure 78
Starting Argus ONE 25
Starting SUTRA GUI 25
Steady-state transport 18
Step-by-step examples 13, 15, 18, 19
Super-blocks 56
SUTRA 3, 6, 89
SUTRA input files 3, 6, 7, 11
SUTRA input files, creating 11
SUTRA input files, editing with text editor 83
SUTRA input files, naming 12
SUTRA Interface-Configuration window ... 8, 27
SUTRA manual 4
SUTRA Mesh layer 7, 10, 53, 60
SUTRA Mesh parameters 46, 62
SUTRA Mesh parameters, links 63
SUTRA Mesh parameters, list 63
SUTRA Model Layers 53
SUTRA Non-Spatial Information dialog 8, 18, 29
SUTRA output files 3, 83
SUTRA Post Processing Charts layer . 18, 19, 86
SUTRA problem type, selecting 27
SUTRA simulation window 12, 13
SUTRA, failed simulations 93
SUTRA.DLL 4, 6, 25
SUTRA.ERR file 83
SUTRA.EXE, a revised version 6
SUTRA.FIL file 84
SUTRA.INI 6, 70
SUTRA.LAY 5, 6, 67, 70
sutra.log, SUTRA output file 82
SUTRA.MET 5, 6, 83
SUTRA.VAL 6

T

Tabbed-dialog box 7, 29
temperature_of_source 75
Temporal Controls 34
Thermal conduction through boundary 77
THICK, SUTRA variable 63
Thickness layer 72
Thickness of model, selecting type 28
Time dependence, solute or energy source 77
time_dependence 75
time_dependence, fluid sources 77
time_dependence, specified concentration or temperature 79
time_dependence, specified head or pressure .. 78
Title Window 26
TITLE1, SUTRA variable 20
TMAX, SUTRA variable 18
Total source, solute or energy 77
total_source 74, 75, 76
Transient solute transport 18
Troubleshooting 93

U

UBC, SUTRA variable 62, 64
UIN, SUTRA variable 63, 76
Unsaturated and saturated flow model, selecting 28
Unsaturated flow simulation, FORTRAN compiler needed 6
Unsaturated Properties layer 73
Uppercase parameters 70, 75, 77
UVEC, SUTRA variable 64

V

Variable background value 21
Variable-density flow model, example 19
Variable-density model, selecting 28
Velocity results for each element, SUTRA output file d4 83
Version of Argus ONE 4, 5
Version of SUTRA 6
Vertical exaggeration for cross-sections 41
Visibility of Layers 42
Visualizing elementwise distributions 65
Visualizing nodewise distributions 66
Visualizing SUTRA Simulation Results 1, 12, 84

W

Walk-Throughs 13, 15, 18, 19
Wells 74
World Wide Web site for Argus Interware 4
World Wide Web site for USGS software 4

Z

Zoned information, using as input 48