

User Interface

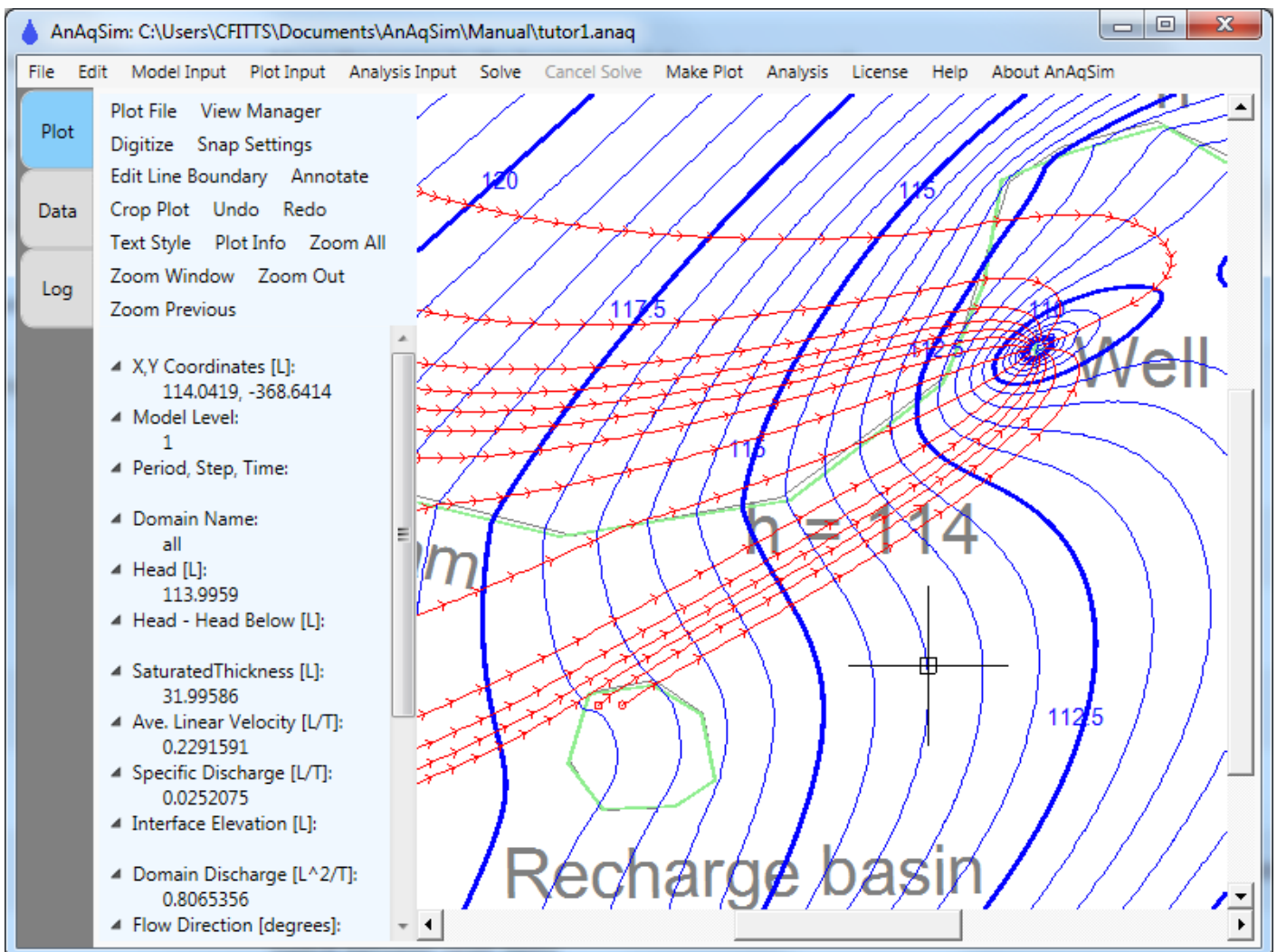
- [Overview](#)
- [Plot Tab](#)
- [Data Tab](#)
- [Log Tab](#)
- [Menu Keyboard Shortcuts](#)

Overview

The user interface consists of one main menu plus three different tabs:

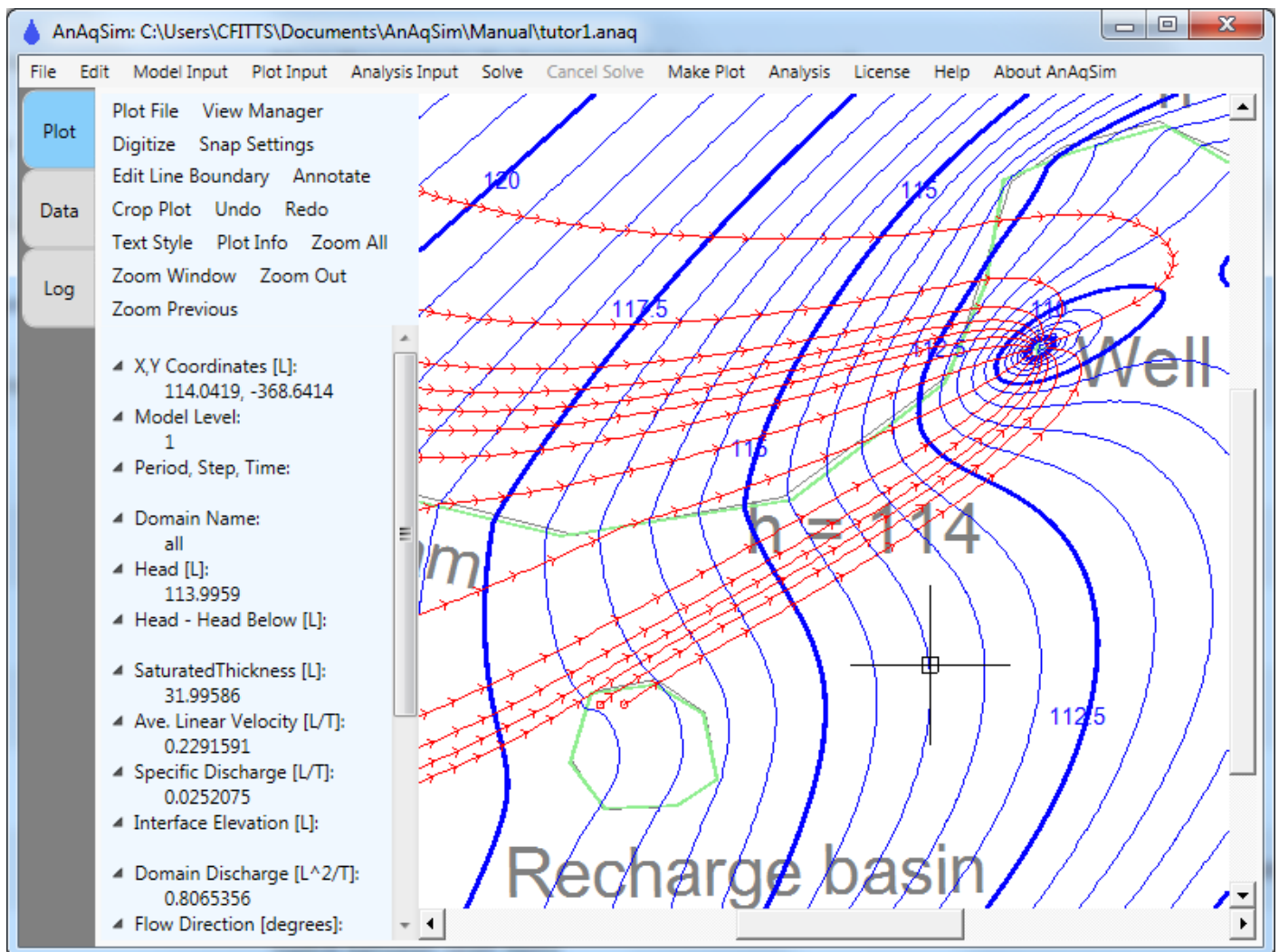
- **Plot:** for graphically displaying map-based inputs and outputs
- **Data:** for editing input data. The data view is automatically displayed when you choose to edit something under the Model Input, Plot Input, or Analysis Input menus.
- **Log:** for displaying text outputs. The run log displays information when files are opened or saved, when the Solve menu is executed, and in response to a number of choices under the Analysis menu.

More information about each of these tabs is given in the following pages. Change from one tab to another by clicking on the tab at the left. The current active tab is highlighted in blue. In the following image, the Plot tab is selected.



Plot Tab

The Plot tab shows the model inputs and results plotted in map view. This view is automatically shown after opening an existing model, and after making a plot under the [Make Plot](#) menu. Most of the Plot view is a map view of the model that can display a basemap, model elements, simulated heads, flow vectors, pathlines, etc. The plot area has scroll bars that allow you to shift the view left/right and up/down while retaining the same scale. The scroll wheel on most mouse devices will cause the view to zoom in and out. Pressing down on the scroll wheel and moving the mouse will allow you to pan the view with most mouse devices.



On the upper left is a separate [plot view menu](#) (Plot File, View Manager,...Zoom Previous) that applies just to the plot. Choices in this menu allow you to zoom to a different view, save or print the plot, digitize and edit coordinates, or add annotations to the plot. See the tutorial videos at the [website](#) to see the various functions in the plot view menu explained and demonstrated. Right-clicking your mouse over the plot brings up a [context menu](#) with many choices for digitizing,

editing line boundaries, and generating outputs related to the cursor location.

Context Menu (right click over plot)

When the cursor is over the plot, you may bring up a context menu by clicking your right mouse button. This menu allows the following options, which are handy short cuts while building, editing, and analyzing a model:

- Edit Domain Properties - switches to the data view with a domain data table displayed and one highlighted row: the row of the domain where the cursor is located.
- Edit Nearest Well - switches to the data view with a well data table displayed and one highlighted row: the row of the well nearest the cursor location.
- Edit Nearest Line Boundary - switches to the data view with a line boundary data table displayed and one highlighted row: the row of the line boundary nearest the cursor location.
- Edit Area Sink that Applies Here - switches to the data view with an area sink data table displayed and one highlighted row: the row of the area sink that applies at the cursor location.
- Digitize - includes the most commonly used commands (Point, Polyline, Clear Digitizing Marks) from the [Plot View/Digitize](#) menu, but without the pop-up instructional windows. These are efficient for experienced users.
- Edit Line Boundary - allows the same operations (Insert Vertex, Delete Vertex) as are found in the [Plot View/Digitize](#) menu, but without the pop-up instructional windows. These are efficient for experienced users. One line boundary, spatially-variable area sink (SVAS) polygon boundary, or vertical leakage polygon must be selected before executing either of these menu items.
- Set Plot Window to Current View - this is the same as [Plot Input/Set Plot Window to Current View](#), setting the window for subsequent plots to the current view.
- Set Plot Window to Entire Model - this is the same as [Plot Input/Set Plot Window to Entire Model](#), setting the window for subsequent plots to the entire model.
- Check Nearest Well Head and Discharge - this writes the head and discharge of the nearest well to the Log tab.
- Check Nearest Line Boundary Condition - this allows you to check the accuracy of the approximation of boundary conditions along the particular segment of a line boundary that is closest to the cursor when this is selected. This causes a graph to be made of the conditions on that boundary segment. The graphs vary depending on the type of line boundary. You may export a bitmap graphic of the graph or the underlying data (see the [Exporting X-Y Graphs](#) topic). Prior to release 2016-2, this feature was under the [Analysis menu](#).
- Graph Head Hydrograph(s) Here, All Levels - for a transient simulation, this causes hydrographs (graphs of head vs. time) to be made that show heads in all model levels at the location of the cursor. These graphs do not contain the initial heads (time=0) at these

locations. If you want hydrographs that include initial heads, use [Analysis Input/Hydrograph Points](#) and [Analysis/Head Hydrographs](#) and [Analysis/Drawdown Hydrographs](#).

Data at Cursor Location

To the left of the plot and below the plot menu is an area that displays the coordinates of the cursor, model information and model results (X,Y Coordinates, Model Level, Domain Name, Head,...) at the location of the cursor. As you move the cursor, this data updates based on the cursor location.

Each item may be hidden or visible; to toggle this, click on the triangle to the left of the label. If the window is not tall enough to display all items, use the scrollbar immediately to the right of this area.

In transient models, the values reflect the time step of the selected period, which ends at the time listed. Head and interface values are at the end of the time step at the time listed. Discharge-related values apply over the duration of the time step listed.

The model results shown are described below.

- **Head Above - Head** is the difference in head from the level above to the level of the plot.
- **Head - Head Below** is the difference in head from the level of the plot to the level below.
- **Interface Elevation** is the elevation of the fresh-salt interface (only applies in models with fresh-salt interface domains).
- **Domain Discharge** is the specific discharge times the saturated thickness, which equals the discharge in the domain per unit length normal to the discharge.
- **Flow Direction** is the direction of the specific discharge or average linear velocity vector, assuming the positive x axis is zero degrees.
- **Top of Model Condition** is the condition at the top of the topmost domain of the model at this x,y, defined by the area sink input.
- **Bottom of Model Condition** is the condition at the bottom of the bottommost domain of the model at this x,y, defined by the area sink input.
- **Modeled Extraction** is the extraction per area γ [L/T] that Anaqsim is modeling at the point, using the spatially-variable area sink functions. This quantity is defined by equation 13 of [Fitts \(2010\)](#).
- **Extraction from Heads** is defined as:

$$L_t + L_b + S \frac{\partial h}{\partial t}$$

where L_t is leakage out the top of the subdomain computed using head differences and vertical hydraulic conductivities, L_b is the leakage out the bottom of the subdomain computed similarly, and $S \partial h / \partial t$ is the transient discharge/area into storage computed

using the head change over a time step and storativity. At spatially-variable basis points, this equals the modeled extraction, but between basis points, the two are unequal, but close if the basis point spacing is small enough. This quantity is defined by equation 6 of [Fitts \(2010\)](#). In a transient simulation, this cannot be computed for the first time step, since the initial head at time zero at the cursor location is not known and therefore ∂h can't be computed.

- **Leakage out Top** is the leakage out the top of the domain, L_t as defined above [L/T].
- **Leakage out Bottom** is the leakage out the bottom of the domain, L_b as defined above [L/T].
- **Storage Flux** is the flux into storage in a transient model, $S \partial h / \partial t$, as defined above [L/T].
- **Leakage Factor Above** is the leakage factor (**LF**) computed for upward leakage from the subdomain. If the subdomain is at the top of the model with no overlying level, the leakage factor is computed as:

$$LF = \frac{Tb^{1/2}}{2 * K_v}$$

Where T is the transmissivity of the subdomain, b is the saturated thickness of the subdomain, K_v is the vertical hydraulic conductivity of the upper half of the subdomain.

If the subdomain has another subdomain overlying it, the leakage factor is computed with the following equation:

$$LF = \left(T * T_a \left(\frac{b}{2 * K_v} + \frac{b_a}{2 * K_{va}} \right) / (T + T_a) \right)^{1/2}$$

Where T_a is the transmissivity of the subdomain above, b_a is the saturated thickness of the subdomain above, and K_{va} is the vertical hydraulic conductivity of the lower half of the subdomain above. Leakage factors are used as guidance for determining appropriate basis point spacing for [spatially-variable area sinks](#). See the discussion under spatially-variable area sinks.

- **Leakage Factor Below** is the leakage factor (**LF**) computed for downward leakage from the subdomain. It is computed in a manner analogous to that described above for Leakage Factor Above.

Data Tab

The data tab allows you to edit [model inputs](#), [plot inputs](#), and [analysis inputs](#) with a data grid component that displays data from the underlying data tables. To display and edit a table, make a selection under the Model Input, Plot Input, or Analysis Input menus.

Label_unique	Level	Domain_Type	Top_elevation	Bottom_elevation	Average_head	Porosity	Storativity	Specific_yield
3D middle	2	confined	100	90	110	0.11	0.001	0.1
3D lower	3	confined	90	80	110	0.11	0.001	0.1
all	1	unconfined	110	82	110	0.11	0.001	0.1
recharge basin	1	unconfined	110	82	110	0.11	0.001	0.1
3D upper	1	unconfined	110	100	110	0.11	0.001	0.1

There is a context menu that pops up when you right-click over the grid. Options in this menu include

- Paste New Rows - this pastes in new rows of data that are tab-delimited between columns. This format allows you to paste data copied from spreadsheets like Excel.
- Copy Selected Rows - this copies the selected rows to the system clipboard, which can then be pasted into spreadsheets like Excel.
- Copy All Rows - this copies all rows in the spreadsheet to the system clipboard, which can then be pasted into spreadsheets like Excel.

Using the Data Grid

A data table is displayed in the grid when you select an item under the Model Input, Plot Input, or Analysis Input menus. The displayed data is linked to one of several database tables, and when you edit the displayed data, the underlying data table is updated.

The table of data is displayed with headers that define each column, such as Label, Domain, Parameters_per_line, ... as shown below.

Data: head-specified lines

Label	Domain	Parameters_per_line	Domain_Boundary	h_start	h_end	Coordinates	Off_periods
lake	all	6	<input checked="" type="checkbox"/>	104	104	Edit	
swamp	3D upper	3	<input checked="" type="checkbox"/>	101	100.5	Edit	

You can move from cell to cell with the arrow keys or using the mouse. The current row is highlighted blue. You can enter new values by navigating to a cell and typing a new entry, or you can double-click on a cell to edit cell contents with a text editor, as shown below, where the contents of the cell with "101" is being edited.

Domain_Boundary	h_start	h_end	Coordinates	C
<input checked="" type="checkbox"/>	104	104	Edit	
<input checked="" type="checkbox"/>	101	100.5	Edit	

When you enter a value in a grid cell, the underlying database is updated when you press enter or move to a different cell. At this point the cell value is checked to make sure it is compatible (e.g. a positive real number for hydraulic conductivity). If the value is incompatible, an error message is displayed and you must correct the cell entry. Be sure to remember to press enter or move to another cell after editing the value in a cell, otherwise the value will not be changed in the database.

New rows are created by editing the blank row at the bottom of the table. A new row of data is entered into the underlying data table only when you press enter after editing the row, at which point a new blank line appears below the line just entered. The following two screen shots shows a new 2nd row before it has been entered in the database (no blank row shows below it), and after (blank row below 2nd row).

Data: head-specified lines

Label	Domain	Parameters_per_line	Domain_Boundary	h_start	h_end	Coordinates	Off_periods
lake	all	6	<input checked="" type="checkbox"/>	104	104	Edit	
swamp	3D upper	3	<input checked="" type="checkbox"/>	101	100.5	Edit	

Data: head-specified lines							
Label	Domain	Parameters_per_line	Domain_Boundary	h_start	h_end	Coordinates	Off_periods
lake	all	6	<input checked="" type="checkbox"/>	104	104	Edit	
swamp	3D upper	3	<input checked="" type="checkbox"/>	101	100.5	Edit	
			<input type="checkbox"/>			Edit	

In data tables that contain multiple rows, the leftmost field is often called Label, and it is always displayed even if you scroll far to the right. No entry is required in this field, and it accepts any text. It is wise to fill in a text label in this field (e.g. "PW-103" for a pumping well). The label will help you know which feature this row represents, and many analysis outputs make use of this label. Also you can sort the data based on entries in this column to easily find the row you want. The contents of the table can be sorted by clicking on the column header. Clicking a second time reverses the sort order. It is a good idea to choose labels that easily allow you to sort features. For example, if you want to easily find a group of wells on property A, you could give them labels such as "A_MW102", "A_MW105", "A_MW113"... so they would be grouped together after sorting by the label column.

Column widths are automatically adjusted to fit the contents. You can increase or shrink column widths by dragging the left or right the vertical line that separates columns in the header (top) row. Double-clicking on this vertical line automatically resizes the column width to fit the contents.

Some columns, like the Parameters_per_line column in the table shown above, are edited using a drop-down list of choices. To see the list, double-click the cell, then select the item you want.

Other columns, like the [Coordinates](#) column, contain buttons to edit or select data; these are edited by clicking on the button.

Number Formats

All data grid cells that expect numerical input have common format constraints. You can input real numbers with formats such as the following:

- 1256.82
- 0.004
- 1.4e-2

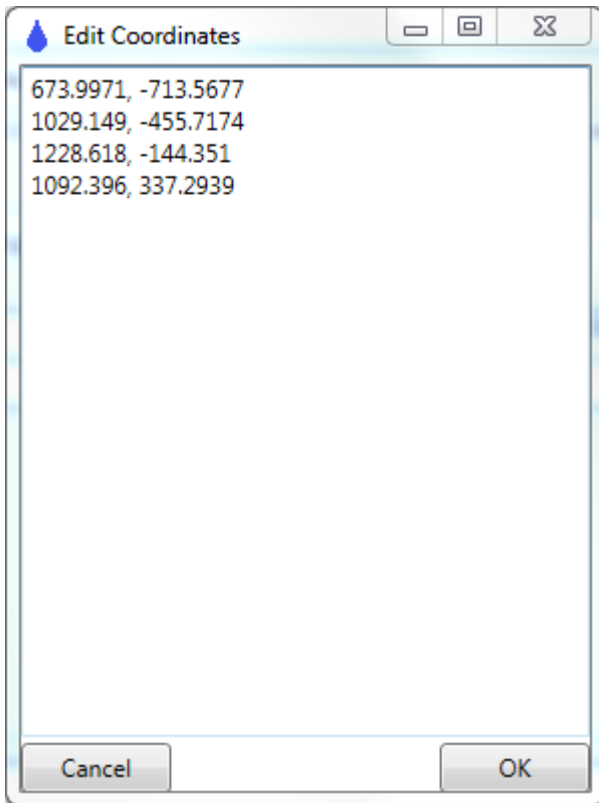
The last one is scientific notation for 1.4×10^{-2} .

You should not insert commas to mark thousands, millions, (e.g. 1,200,000) as the comma may be interpreted as a decimal mark. In North America, the convention is to use a period for the decimal marker. In Europe, the comma is often used as a decimal marker. There is a Windows operating system setting to switch between these modes. Often, European users need to adjust these

settings to use Anaqsim.

Editing Coordinates

In many of the data input tables, there are columns and cells that display an "Edit" button in the Coordinates column. When you click the button, a text box window pops up and you enter coordinate data there:



Often, you will digitize the coordinates in the plot tab and then paste the coordinates into this text box window. Alternately, you can just type coordinates in. The OK button records the edited coordinates and the Cancel button does not.

Once input, [coordinates can be edited graphically by selecting the line boundary and then moving the vertexes or inserting or deleting vertexes.](#)

Deleting Data Rows

Delete one or more rows of data in the data table by selecting rows and then pressing the Delete key. Row(s) are selected by clicking (and dragging for multiple rows) in the leftmost column of the grid. A dialog will ask you if you really want to delete those records from the data table.

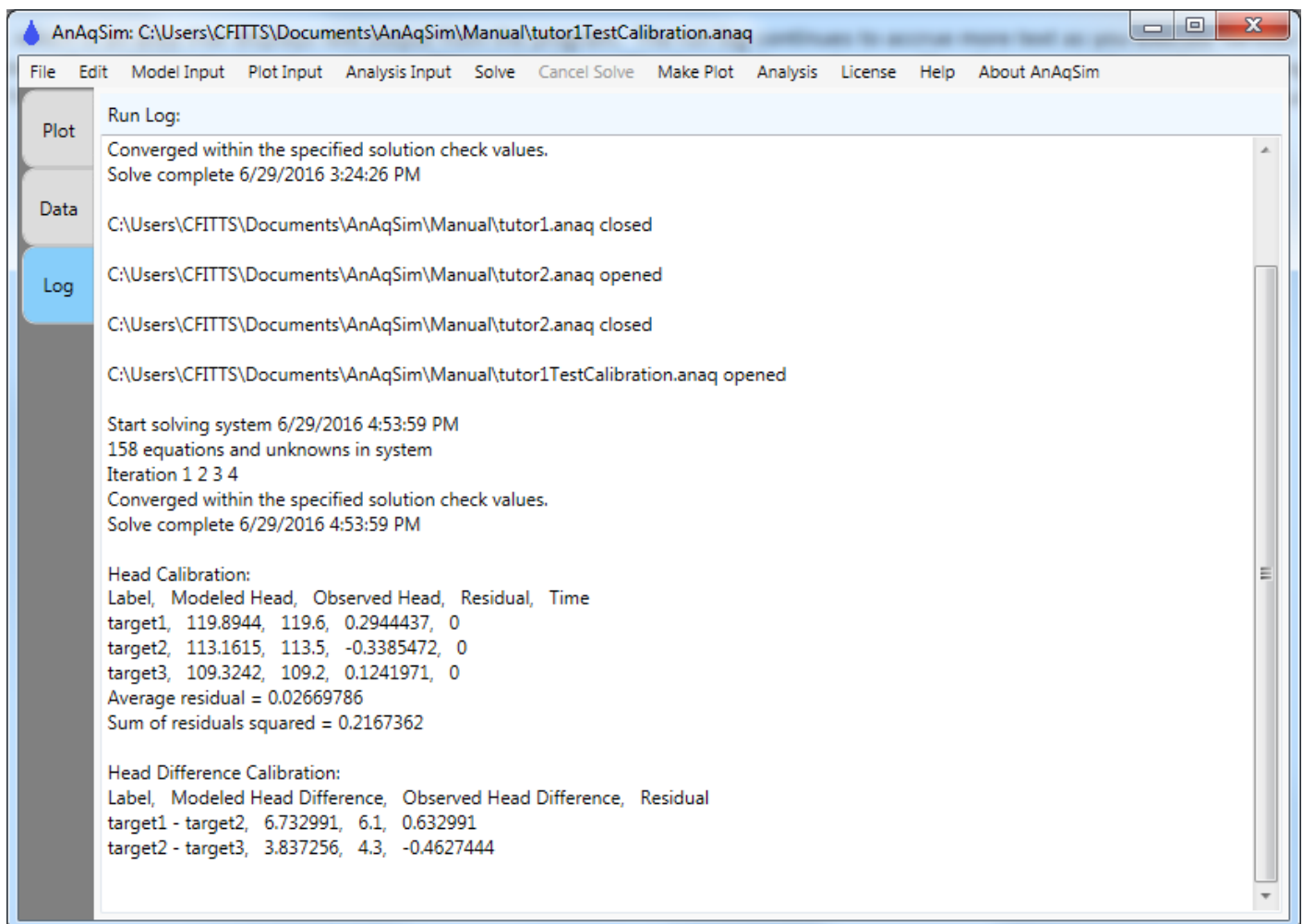
Importing and Exporting Data

To import data from Excel into a data table, highlight a block of data in an Excel sheet that corresponds to row(s) of data in a data table, copy that block in Excel, then right-click over the data grid and select Paste New Rows. This will add these copied rows to the data table. Make sure that the columns in the copied block match the columns in the data table. Data in Coordinates columns cannot be pasted in due to their multi-line structure, but all other columns can be pasted in. In the case of a Coordinates column, a paste operation leaves that blank and you must enter the coordinates by clicking on the Edit button in that column.

To export rows of data to Excel or a text file, select rows of data (see section above) and then right-click over the data grid and select Copy Selected Rows. After doing this the rows of data are in the computer's clipboard as tab-delimited data, which can then be pasted into Excel or into text files.

Log Tab

The Log tab holds the run log, which is an area that displays text output from the program. The run log continues to accrue more text as you execute various tasks such as updating license information, opening a file, solving the system of equations, checking boundary conditions, checking calibration results, or closing a file. If the text in the run log gets long enough, a scroll bar will appear to let you scroll through the entire log, as shown below. You can select all or a portion of the text in the run log and cut, copy, and paste this text. This is an easy way to move text results to another document.



Menu Keyboard Shortcuts

You can access common menu items with keyboard shortcuts by pressing the key sequences as listed below. Many are standard Windows shortcuts.

Ctrl-O	File/Open
Ctrl-S	File/Save
F12	File/SaveAs
Ctrl-W	File/Close
Alt-FE	File/Exit
Alt-S	Solve
Alt-PA	Make Plot/All Selected Features
Alt-PE	Make Plot/Elements Only
Alt-V	Switch View
Alt-H	Help