

Analysis Menu

Under this menu there are many choices for analyzing a model after the system of equations has been solved. Remember to Solve the system after making input changes and before using this menu. Some choices generate graphs, others generate text output to the run log. Many of the functions under this menu are demonstrated in detail in the tutorial videos at the [website](#).

Check Boundary Conditions at Latest Iteration

Selecting this causes the program to run through a check of all boundary conditions at the end of the most recent iteration (solved model of last time step if it converged, or last iteration in the case of a model that didn't converge), to see if they are met within the tolerances specified under [Model Input/Solution/Check Settings](#). The results are written to the run log and the view is switched to the view with the run log. This can be useful to see which boundary conditions are causing non-convergence, if your model doesn't converge. It may steer you to relax certain check settings or reconfigure the model.

In the following example, all of the boundary conditions met the check tolerances, so no specific boundary condition is listed:

Run Log:

```
Begin check of boundary conditions 5/31/2011 4:57:04 PM ...
The following will list just specific boundary conditions that did not meet the following criteria specified in model input / solve:
1) Head conditions [L]: deviation of modeled values are no more than 1.000E-003
2) Discharge (Q) conditions [L^3/T]: deviation of modeled values are no more than 1.000E-003
3) Discharge per length (Q/L or Qn) conditions [L^2/T]: deviation of modeled values are no more than 1.000E-004
4) Extraction (Q/area) conditions [L/T]: deviation of modeled values are no more than 1.000E-006
Checking 1 head-specified line boundaries...
Checking 3 river line boundaries...
Checking 1 Qn-specified line boundaries...
Checking 1 inter-domain line boundaries...
Checking 231 basis points at spatially variable area sinks...
End of list - all boundary conditions have been checked.
```

After decreasing the Qn_check_tolerance to 2×10^{-9} and the Extraction_check_tolerance to 4×10^{-7} , the same model had some of the boundary conditions exceeding their check tolerances:

Run Log:

```
Begin check of boundary conditions 5/31/2011 5:01:00 PM ...
The following will list just specific boundary conditions that did not meet the following criteria specified in model input / solve:
1) Head conditions [L]: deviation of modeled values are no more than 1.000E-003
2) Discharge (Q) conditions [L^3/T]: deviation of modeled values are no more than 1.000E-003
3) Discharge per length (Q/L or Qn) conditions [L^2/T]: deviation of modeled values are no more than 2.000E-009
4) Extraction (Q/area) conditions [L/T]: deviation of modeled values are no more than 4.000E-007
Checking 1 head-specified line boundaries...
Checking 3 river line boundaries...
2.012E-009 = modeled discharge per length deviation at river control point at (133.242, -79.310) in domain well upper
2.110E-009 = modeled discharge per length deviation at river control point at (137.418, -73.785) in domain well upper
Checking 1 Qn-specified line boundaries...
Checking 1 inter-domain line boundaries...
Checking 231 basis points at spatially variable area sinks...
5.571E-007 = modeled deviation in extraction rate at spatially-variable basis point at (225.764, -55.658) in domain well upper
5.571E-007 = modeled deviation in extraction rate at spatially-variable basis point at (225.764, -55.658) in domain well middle
End of list - all boundary conditions have been checked.
```

This output may also be written in [automated runs](#).

Head-Specified Well Discharges

Under this menu, selecting **Write Discharges to Run Log** writes a listing of head-specified well discharges to the run log. If the model is steady, it will list one discharge per well. If the model is transient, it will list period, step, time, and discharge at each time step. The time listed is the time at the end of the step. The discharge listed applies from the start to the end of the step.

Discharges of such wells are not known prior to solving the system ([Solve](#)).

If you select **Write Discharges to CSV**, a dialog pops up to select an output file location. The discharges of all head-specified wells are then written to that file location, providing period, step, time and discharge at each time step.

If you select **Graph Transient Discharges**, a dialog pops up where you select one well, and a graph is made of the discharge vs. time at the selected well. Discharges are constant during a time step, so plots show abrupt changes in discharge between steps. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Discharge-Specified Well Heads

Under this menu, selecting **Write Heads to Run Log** writes a listing of heads at all discharge-specified wells to the run log. This is head at the specified well radius. If the model is steady, it will list one head per well. If the model is transient, it will list period, step, time, and head at each time step. Heads at such wells are not known prior to solving the system ([Solve](#)).

If you select **Write Heads to CSV**, a dialog pops up to select an output file location. The heads of all discharge-specified wells are then written to that file location, providing period, step, time and

head at each time step.

If you select **Graph Transient heads**, a dialog pops up where you select one well, and a graph is made of the head vs. time at the selected well. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Multi-Domain Well Discharges by Domain

Under menu, selecting **Write Discharges to Run Log** writes a listing of the discharge contributed by each domain in a discharge-specified multi-domain well to the run log. If the model is steady, it will list one discharge per domain. If the model is transient, it will list period, step, time, and discharge at each time step in each domain. The time listed is the time at the end of the step. The discharge listed applies from the start to the end of the step. The discharges in each domain are not known prior to solving the system ([Solve](#)).

If you select **Write Discharges to CSV**, a dialog pops up to select an output file location. The discharge of all multi-domain wells are then written to that file location, providing period, step, time and discharge at each time step in each domain.

If you select **Graph Transient Discharges**, a dialog pops up where you select one discharge-specified multi-domain well, and a graph is made of the discharge contributed by each domain vs. time at the selected well. Discharges are constant during a time step, so plots show abrupt changes in discharge between steps. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Internal Head-Specified Line Boundary Discharges

Under this menu, selecting **Write Discharges to Run Log** writes a listing of internal head-specified line boundary discharges to the run log. If the model is steady, it will list one total discharge per line boundary. If the model is transient, it will list total discharges vs. time, with period, step, time, and discharge at each time step. The time listed is the time at the end of the step. The discharge listed applies from the start to the end of the step. The discharges of such line boundaries are not known prior to solving the system ([Solve](#)).

If you select **Write Discharges to CSV**, a dialog pops up to select an output file location. The discharges of all internal head-specified lines are then written to that file location, providing period, step, time and discharge at each time step.

If you select **Graph Transient Discharges**, a dialog pops up where you select one internal head-specified line boundary, and a graph is made of the total discharge vs. time at the selected line boundary. Discharges are constant during a time step, so plots show abrupt changes in discharge between steps. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

River Line Boundary Discharges

River line boundary discharges are split into two sections, individual reaches and grouped reaches. Anaqsim will sum a group of river line boundary discharges if your river line boundaries have labels with a common root. A common root consists of any label that contains the common character string "____: #". For example, if you had three river line boundaries and their labels were dark creek: #1, dark creek: #2, and dark creek: #3, the discharges of each of these 3 are available individually as well as their total discharge. The discharges of river line boundaries are not known prior to solving the system ([Solve](#)).

Selecting **Write Discharges to Run Log** writes a listing of river line boundary discharges to the run log. If the model is steady, it will list one total discharge per line boundary. If the model is transient, it will list period, step, time, and total discharge at each time step. The time listed is the time at the end of the step. The discharge listed applies from the start to the end of the step. Discharges are negative for flow out of the domain into the river and positive for flow from the river into the domain.

If you select **Write Discharges to CSV**, a dialog pops up to select an output file location. The discharges of all river line boundaries are then written to that file location, providing period, step, time and discharge at each time step.

If you select **Graph Transient Discharges**, a dialog pops up where you select one river line boundary, and a graph is made of the total discharge vs. time at the selected line boundary. Discharges are constant during a time step, so plots show abrupt changes in discharge between steps. This option is not yet available for grouped reaches. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Conditions Along a Line

This selection allows you to create graphs of head, domain elevations, interface elevations, aquifer discharge, or extraction rate along a line.

There are three options:

- Writing outputs from pre-defined lines to a CSV file,
- Graphing outputs from pre-defined lines,
- Graphing outputs from a new line

Pre-defined lines are defined prior to solving the model, and must be defined in [Analysis](#)

[Input/Transient Line Conditions](#). Graphing outputs from a new line can be specified after model solve, but graphs are limited to a single time step.

Whilst the predefined lines are called 'Transient Line Conditions' they can be used to provide outputs from both steady and transient models.

The options for choosing a variable to plot and/or write to CSV file are:

- Head. Creates graphs of head vs. time along the line. The profile includes the initial ($t=0$) heads along the line. Heads shown are at the end of the time step listed.
- Discharge tangent to line. Creates graph of discharge vs time along the line. Discharge tangent to line is the component of discharge in the domain tangent to the line [$L^3/T/L = L^2/T$], positive for flow proceeding from the first towards the second point.
- Discharge normal to line. Creates graphs of discharge vs time along the line. Discharge normal to line is the component of discharge in the domain normal to the line [$L^3/T/L = L^2/T$], positive for flow from left to right as you proceed from the first to the second point.
- Extraction. Creates graphs of extraction vs. time along the line. Extraction is the extraction [$L^3/T/L^2 = L/T$] due to recharge, leakage out the top, leakage out the bottom, and storage fluxes. Both modeled extraction and extraction computed from heads are shown. In a well-defined model these two curves will be close to each other. If they are not, you probably need denser basis point spacing in your [spatially-variable area sinks](#). The extraction shown applies over the time step that ends at the time listed in the plot legend.
- Leakage at Top. Creates graphs of vertical leakage out the top of the domain vs. time along the line. Leakage [$L^3/T/L^2 = L/T$] is computed based on head differences, vertical hydraulic conductivities ($K3$) and saturated thicknesses. The leakage shown applies over the time step that ends at the time listed in the plot legend.
- Leakage at Bottom. Creates graphs of vertical leakage out the bottom of the domain vs. time along the line. Leakage [$L^3/T/L^2 = L/T$] is computed based on head differences, vertical hydraulic conductivities ($K3$) and saturated thicknesses. The leakage shown applies over the time step that ends at the time listed in the plot legend.
- Storage Flux. Creates graphs of flux into storage in the domain vs. time along the line. Flux [$L^3/T/L^2 = L/T$] is computed based on the change in head during the time step listed, and applies over the time step that ends at the time listed in the plot legend

When Graphing outputs from a new line, some additional options are present. A dialog box pops up which requires the following input:

- Model Level(s) selects the model level(s) of the graphed condition. You may select one or multiple levels (e.g. you may want to plot a profile of heads in all levels along the line).
- Period, Step, Time selects the snapshot in time that you graph for a transient run. Starting with release 2016-2, this allows selection of the solution at the end of any time step. Prior to 2016-2, only the final time step solution was available for this function.
- Line Coordinates are the coordinates of the end points of the line. The coordinates are two lines or input, each with two real numbers delimited by a comma (X, Y). You may digitize the coordinates with the [Digitize/Polyline](#) selection in the plot view menu and then paste them in. This [line can be displayed on plots](#). The line may also be edited graphically, like can be done with line boundaries: click on a line, then drag the purple squares at the vertexes to new locations.
- Number of points along the line is the number of points where the head, normal discharge, or extraction is evaluated along the line. More data points make for a smoother plot.
- Select whether you want to graph a Vertical profile (Head), Discharge tangent to line, Discharge normal to line, Extraction, Vertical Leakage (which includes both leakage at Top and Leakage at Bottom) and Storage Flux vs. distance along the line.

If you select Vertical profile, you have the option of showing head, domain boundaries, fresh/salt interface elevations, pathlines, and/or pathline time markers. Depending on what you select under Level, you will make a profile of the selected levels.

When you have made a graph, it shows the selected item(s) vs. distance along the selected line. You may export a bitmap graphic of the graph or the underlying data (see the [Exporting X-Y Graphs](#) topic). For a transient simulation, this shows the conditions at the end of the simulation.

The location of this [line can be displayed on plots](#). The line may also be edited graphically, like can be done with line boundaries: click on a line, then drag the purple squares at the vertexes to new locations.

Fresh/salt interface profiles

There are two options for plotting fresh/salt interface profiles. The Dupuit option computes the interface position using the model-simulated head for the domain, regardless of the elevation of the interface within the domain. This assumes that there is no vertical resistance to flow within a domain and no vertical head gradients within a domain (this is commonly called the Dupuit approximation, which is inherent in Anaqsim's domain discharge model). The Dupuit interface position is always computed using the model-simulated head, and assuming pressure equilibrium between this fresh water head and static salt water.

The Dupuit plotting option can be used for single- and multi-level domain interface models. In multi-level models where there are significant head differences between levels, the Dupuit option will generate an interface that has discontinuities at boundaries between levels. This saw-tooth shaped interface is consistent with the Dupuit and Ghyben-Herzberg approximations used in

Anaqsim. With the Dupuit approximation, head jumps abruptly at horizontal boundaries between domains at different levels, so the interface elevation computed with the Ghyben-Herzberg equation must also jump.

Such discontinuities in the interface may be realistic for cases where a thin aquitard, included in the Anaqsim model by adjusting the K3 of an aquifer domain above or below, is the main cause of vertical resistance between levels. On the other hand, if the main source of vertical resistance is not an aquitard but the actual K3 of the domains involved, the interface discontinuities are just an artifact of the Dupuit approximation. These discontinuities can be smoothed by selecting the Non-Dupuit plotting option.

The Non-Dupuit option uses heads that are calculated in a post-processing step, heads that do vary with elevation within a domain. In multi-level models simulating vertical leakage between levels, there is information available to approximate the vertical distribution of head within a domain.

Using the simulated heads in the domain and in overlying or underlying domains, and the vertical hydraulic conductivity of the domains (K3), and assuming that the simulated domain head applies at the mid-height of a domain, it is possible to construct an estimated vertical profile of heads within a domain at a point. This profile is non-Dupuit, since head varies with elevation in a domain. For example, if the K3 of the lower half of level 1 domain A equals K3 of the upper half of level 2 domain B, the approximated head would vary linearly from the midpoint of domain A to the midpoint of level B. On the other hand, if the K3 of the lower half of level 1 domain A were twice the K3 of the upper half of level 2 domain B, the approximated head would vary linearly in the lower half of domain A and linearly in the upper half of domain B, but the vertical head gradient in the upper half of B would be twice that in the lower half of A. If you elect the Non-Dupuit option for interface position, the interface position is computed so that this approximated head balances pressure with static salt water. The Non-Dupuit option will have these characteristics compared to the Dupuit option:

Where no vertical leakage can be computed, no non-Dupuit interface can be computed, so none can be displayed. This is true in single-level areas of models.

- Interfaces will not abruptly jump at the boundary between different levels like with the Dupuit option, but will instead be continuous since the approximated head profile is continuous.
- In areas with an upward component of vertical flow, the non-Dupuit interface will be steeper than the Dupuit interface.
- In areas with a downward component of vertical flow, the non-Dupuit interface will be shallower than the Dupuit interface.
- In areas with little or no component of vertical flow, the non-Dupuit and Dupuit interfaces will be similar.

Pathlines

When you check pathlines, the profile will show thin lines along each pathline, which is projected onto the line using a normal (right angle) projection. If you check pathline time markers, a dot is displayed at each time marker (arrow in the plot) location. The pathlines and markers drawn will be

those from the most recent plot made with the open model. In a transient model, the pathline is transient, and not based on the flow field just at the selected time step.

This output may also be written in [automated runs](#).

Sum Discharge Across a Polyline

This sums the domain discharge [L³/T] across a polyline that has been digitized and is currently residing in the clipboard after being digitized. You must first digitize a polyline in the plot view ([Digitize/Polyline](#)) before making this selection. The discharge is written to the run log. Discharge is positive for flow going left to right across the polyline as you proceed from the first vertex to the last. For a transient simulation, discharges at each time step are written to the run log, so you can see how discharge evolved during the simulation.

The discharge is computed by numerically summing normal components of aquifer discharge at 1000 equally-spaced points on each segment of the digitized polyline.

Sum Discharge Across External Boundaries

This sums the domain discharge [L³/T] across all external boundaries ([head specified](#), [normal flux-specified](#), and [head-dependent normal flux](#)). Discharge is positive for flow going left to right across the boundary as you proceed from the first vertex towards the last. The discharge is computed by numerically summing normal components of aquifer discharge at 1000 equally-spaced points on each segment of the boundary. The polyline vertexes that are used for this computation are very slightly displaced into the interior of the boundary from the actual vertexes (within 10⁻⁴ of the segment length from the actual vertexes).

Selecting **Write Heads to Run Log** writes these discharges to the run log. For a transient simulation, discharges at each time step are written to the run log, so you can see how these discharges evolve during the simulation.

If you select **Write Discharges to CSV**, a dialog pops up to select an output file location. The heads of all discharges across external boundaries are then written to that file location, providing period, step, time and discharge at each time step.

If you select **Graph Transient Discharges**, a dialog pops up where you select a boundary, and a graph is made of the discharge vs. time at the selected boundary. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Vertical Leakage Over Polygon Areas

This causes Anaqsim to estimate and output leakages [L3/T] over polygon areas. The leakages will be computed in all model levels over x-y plane areas defined in [Analysis Input / Vertical Leakage Over Polygon Area](#). These [polygons can be displayed on plots](#). They may also be edited graphically, like can be done with line boundaries: click on a polygon, then drag the purple squares at the vertexes to new locations. You may also [insert or delete vertexes](#) once the polygon is selected.

Under this menu, selecting **Write Leakage to Run Log** writes a listing of leakage at all defined polygons to the run log. If the model is steady, it will list one leakage per model level per polygon. If the model is transient, it will list period, step, time, and leakage at each time step. The output text also includes the area of the polygon.

If you select **Write Leakage to CSV**, a dialog pops up to select an output file location. The leakage of all polygons are then written to that file location, providing period, step, time, level and head at each time step.

If you select **Graph Transient Leakage**, a dialog pops up where you select one polygon, and a graph is made of the leakage vs. time at the selected polygon. Exporting the graph or the underlying data is described [here](#).

This output may also be written in [automated runs](#).

Write Number of Displayed (Captured) Pathlines to Run Log

Selecting this causes the number of displayed pathlines to be written to the run log. This can be handy if you are trying to optimize the capture of pathlines originating from a contaminant source area. Use in combination with the [capture constrain](#) option, so only pathlines captured by selected wells are displayed. This information can also be written to the text output file generated by an [automated Anaqsim run](#), which allows you to use PEST or other parameter estimation software to perform the optimization.

Head Hydrographs

After solving a transient simulation, this calculates head vs. time data at the hydrograph points input under the [Analysis Input/Hydrograph Points](#) menu. The data points relate to the end of each time step in the simulation, as defined in the [Model Input/Time Steps](#) menu. To calculate this data, which include initial heads, the following steps must be taken in this sequence:

1. Hydrograph points must be defined in [Analysis Input/Hydrograph Points](#),
2. Create initial head files as outlined in the [Save Locations for Initial Transient Heads](#) topic.
3. [Solve](#).
4. Select this menu choice, which outputs the data in 1 of 3 ways:

If you select **Write Head Hydrographs to Run Log**, the period, step, time and head will be written at each time step to the log file.

If you select **Write Head Hydrographs to CSV**, a dialog pops up to select an output file location. The head hydrographs for all hydrograph points are then written to that file location, providing period, step, time and head at each time step.

If you select **Graph Head Hydrographs**, a dialog pops up and asks if you want to read in observed hydrograph data from a file and plot those data as well as the modeled hydrograph. If you select Yes, you will then open a text file that has data that is either tab-delimited or comma-delimited. The file can have as many series and data values as you would like. An example of the file format for a tab-delimited file is shown below. Each row has 2*n columns for n series of data. The first row has the series label in columns 1, 3, 5,...

The second and subsequent rows have time, value pairs for each series. In this example, columns 1 and 2 have time, value pairs for well1, columns 3 and 4 have time, value pairs for well2, and columns 5 and 6 have time, value pairs for well3. Note there can be different numbers of records in different series (e.g. these three series have 11, 7, and 10 records). This tab-delimited format can be generated directly from Excel by saving as a tab-delimited text (.txt) file.

well 1		well2		well3	
1	101	1	201	1	301
2	102	2	202	2	302
3	103	3	203	3	303
4	104	4	204	4	304
5	105	5	205	5	305
6	106	6	206	6	306
7	107	7	207	7	307
8	108		8	308	
9	109		9	309	
10	110		10	310	
11	111				

The equivalent comma-delimited (.csv) file, a format that can also be saved from Excel, would look like:

well 1,,well2,,well3,
1,101,1,201,1,301
2,102,2,202,2,302
3,103,3,203,3,303
4,104,4,204,4,304
5,105,5,205,5,305
6,106,6,206,6,306
7,107,7,207,7,307
8,108,,,8,308
9,109,,,9,309
10,110,,,10,310
11,111,,,,

You may export a bitmap graphic of the graph or the underlying data (see the [Exporting X-Y Graphs](#) topic).

A similar functionality is available in the [plot context menu](#) (right-click over the plot), which allows you to create hydrographs of all model levels at any location in a transient model. However, hydrographs created with the plot context menu lack initial ($t=0$) heads.

The locations of [hydrograph points can be included in plots](#). These locations may also be edited graphically, like can be done with well locations: click on a hydrograph point, then drag the purple square to the new location.

This output may also be written in [automated runs](#).

Drawdown Hydrographs

This item does roughly the same thing as [Analysis/Head Hydrographs](#) except that it calculates data for initial head - head to give you drawdown vs. time instead of head vs. time. To allow log-log graphing of drawdown vs. time, the ($\text{time}=0$, $\text{drawdown}=0$) data point is omitted. See [Analysis/Head Hydrographs](#) for a detailed description of how to set up this sort of analysis and how to display observed drawdown in addition to modeled drawdown.

Three options of writing outputs to Log, CSV and providing graphs are available. This output may also be written in [automated runs](#).

Calibration Results

Items under this menu allow you to write or graph head calibration results. You can include all head calibration targets or only the selected ones.

This output may also be written in [automated runs](#).

Write All Targets to Run Log

This writes the results of calibration to the run log, including all head, head difference, and velocity calibration targets. All calibration targets are input under the [Analysis Input/Calibration Targets](#) menu.

Write Selected Targets to Run Log

This writes the results of calibration to the run log, including all head, head difference, and velocity calibration targets. All calibration targets are input under the [Analysis Input/Calibration Targets](#) menu.

Graph All Head Targets

This creates a and x-y graph of modeled heads vs. observed heads for all head calibration targets. Head and head difference calibration targets are input under the [Analysis Input/Calibration Targets](#) menu.

Graph Selected Head Targets

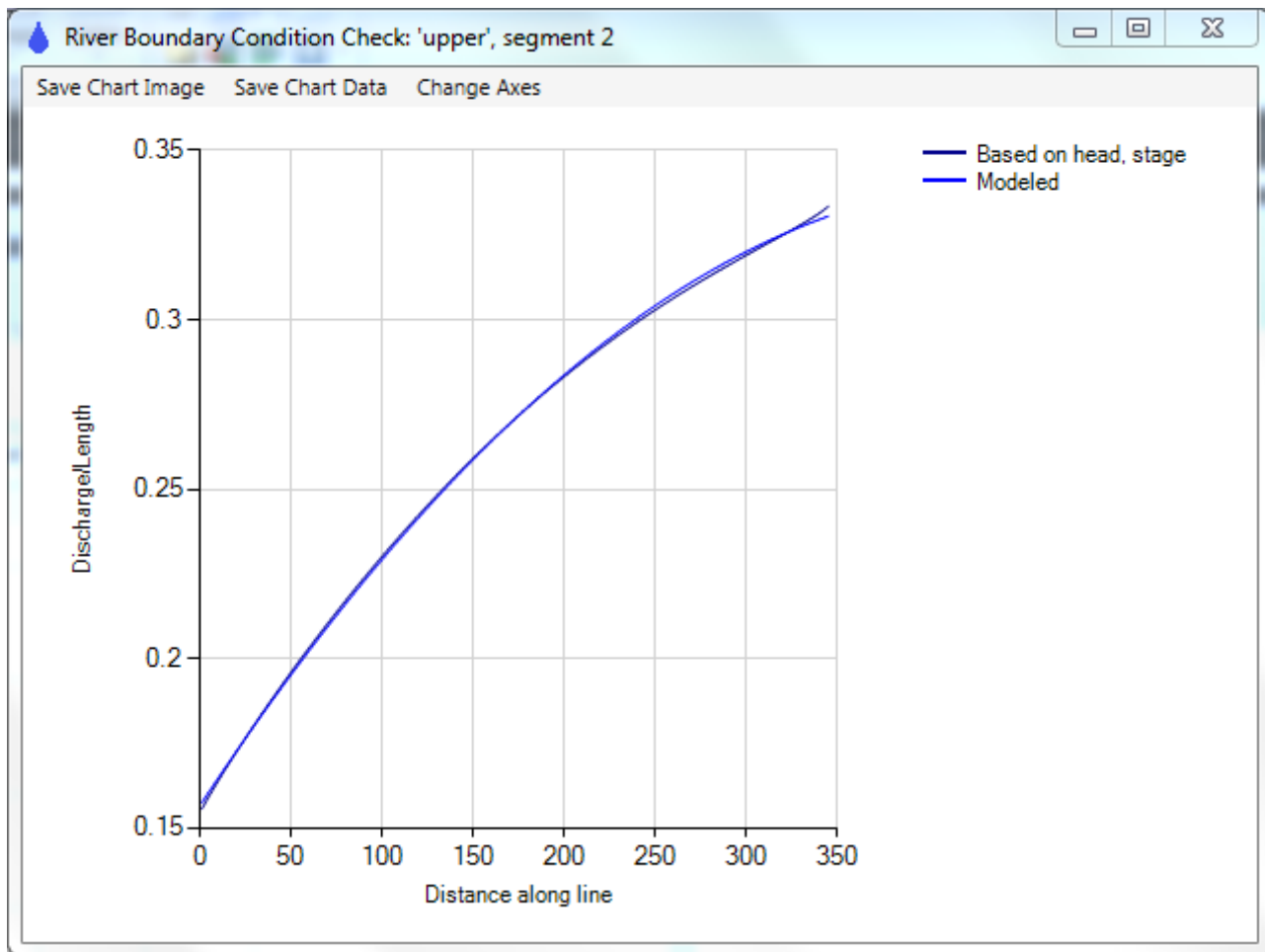
This creates a and x-y graph of modeled heads vs. observed heads for the selected head calibration targets. Head and head difference calibration targets are input under the [Analysis Input/Calibration Targets](#) menu.

X-Y Graphs: Exporting, Changing Axes

Several of the choices under the Analysis menu lead to X-Y graphs. All such graphs are displayed in a separate window like the one shown below, which has two menu items for exporting. Selecting Save Chart Image lets you export a bitmap graphic of the graph to a file in png, jpg, or bmp formats. Selecting Save Chart Data exports the underlying data of the graph to a text file in tab-delimited form so that it may be later imported into documents or spreadsheets like Excel.

You can customize the axes of X-Y graphs under the Change Axes menu item. You can make either the X or Y axis logarithmic, but this only is possible if all values on that axis are positive. You can also change the upper and lower limits and the label/grid interval of the X and Y axes.

When the cursor is moved over a curve on the plot, the label of that curve is displayed.



Revision #6

Created 1 July 2023 18:31:35 by Mary Jeddere-Fisher

Updated 12 January 2024 15:16:59 by Mary Jeddere-Fisher