

Analysis Input Menu

Items under this menu allow you analyze the results of a simulation after the system of equations is solved ([Solve](#)).

Hydrograph Points

These allow you to track head (or drawdown) vs. time at specific points during a transient simulation. They may be used to check the simulated water level at some other point such as an observation well location. After solving, you can make hydrographs with the model-simulated head (or drawdown) and observed values read from a file; this is done with [Analysis / Graph Head Hydrographs](#) or [Analysis / Graph Drawdown Hydrographs](#). This output is also available in [automated runs](#).

- **Label** is a text label that helps you keep track of multiple hydrograph points.
- **Level** is the model level of the hydrograph point.
- **X,Y** are the coordinates of the hydrograph point. The coordinates are two real numbers delimited by a comma. You may digitize the coordinates with the [Digitize/Point](#) selection in the plot view menu.
- **Show** is a checkbox that allows you to selectively include or exclude this particular hydrograph point in written or graphed outputs.

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, and the location is determined by the cursor position.

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.

Transient Line Conditions

This allows you to track head or extraction along a line during both a steady and a transient simulation. Tracking extraction along a line is useful for checking the accuracy of the governing equation approximation in transient simulations. Tracking head along a line allows you to see how head profiles evolve during a transient simulation. After solving, use these menu selections to

make graphs: [Analysis/Transient Line Conditions](#). This output is also available in [automated runs](#).

- Label is a text label that helps you keep track of multiple transient line conditions.
- Level is the model level of the line.
- Coordinates are the coordinates of the end points of the line. Click the Edit button to edit coordinates. The coordinates are two lines, 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.
- Points_on_line is the number of points on the line where head and extraction are computed. More points yield smoother, more accurate profiles at the cost of more computation.

These [lines can be displayed on plots](#). These lines 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.

Calibration Targets

Here is where you input information about targets for calibration. Targets may be heads or head differences between two locations.

Head

- Label is a text label that helps you keep track of head calibration targets.
- Level is the model level of the target.
- X,Y are the coordinates of the target. The coordinates are two real numbers delimited by a comma. You may digitize the coordinates with the [Digitize/Point](#) selection in the plot view menu.
- Observed_Head is the target head.
- Time is the time of the observation for a transient model. This time is measured from time = 0 at the start of the simulation. In a steady model, this parameter is ignored. In a transient model, you may input head targets at the same location but different times. The [Head Difference](#) calibration feature may be used to calibrate to temporal variations in head (e.g. rate of drawdown) during a transient simulation, in addition to spatial head differences.
- Show - if this is checked, this particular calibration point data will display when a plot is made with [What to Plot / Head Calibration](#) is checked. This allows you to make a plot with just certain calibration points posted (e.g. those in a one domain, or one model level, or at a certain time in a transient run).

Head calibration targets may be imported from spreadsheets like Excel. Start by creating a spreadsheet table with the same columns as Anaqsim's head calibration table. Then highlight and copy the block of data you want to paste, like is shown in the first image below for an Excel spreadsheet. Then right-click over the Anaqsim head calibration table and select

[Paste New Rows](#). The new rows will appear as shown in the second image below. If your data has X and Y in separate columns, you can use Excel's CONCATENATE function to combine the X and Y columns with a "," between them into one column.

| | A | B | C | D | E | F |
|---|--------------|--------------|------------|----------------------|-------------|-------------|
| 1 | Label | Level | X,Y | Observed Head | Time | Show |
| 2 | MW-1 | 1 | 1080, 8743 | 245.87 | 0.00 | TRUE |
| 3 | MW-2 | 2 | 876, 7659 | 243.91 | 0.00 | TRUE |
| 4 | MW-3 | 1 | 1173, 6236 | 238.89 | 0.00 | TRUE |
| 5 | MW-4 | 2 | 914, 7847 | 231.34 | 0.00 | TRUE |
| 6 | MW-5 | 3 | 1006, 7572 | 147.02 | 0.00 | TRUE |
| 7 | | | | | | |

Data: calibration heads

| Label | Level | X,Y | Observed_Head | Time | Show |
|-------|-------|------------|---------------|------|-------------------------------------|
| MW-1 | 1 | 1080, 8743 | 245.87 | 0.0 | <input checked="" type="checkbox"/> |
| MW-2 | 2 | 876, 7659 | 243.91 | 0.0 | <input checked="" type="checkbox"/> |
| MW-3 | 1 | 1173, 6236 | 238.89 | 0.0 | <input checked="" type="checkbox"/> |
| MW-4 | 2 | 914, 7847 | 231.34 | 0.0 | <input checked="" type="checkbox"/> |
| MW-5 | 3 | 1006, 7572 | 147.02 | 0.0 | <input checked="" type="checkbox"/> |
| | | | | | <input type="checkbox"/> |

Head Difference

This allows comparison of model-calculated head differences to target head differences. The difference is computed between two head targets listed in the [Analysis Input/Calibration Targets/Head](#) table. This may be used to calibrate to temporal variations in head at one location during a transient simulation (e.g. rate of drawdown), or to spatial head differences (e.g. head gradients). The labels are selected from drop-down lists generated from the [head calibration targets data](#).

- Label1 is the text label for the first head calibration target.
- Label2 is the text label for the second head calibration target.

Velocity

- Label is a text label that helps you keep track of velocity calibration targets.
- Level is the model level of the target.
- X,Y are the coordinates of the target. The coordinates are two real numbers delimited by a comma. You may digitize the coordinates with the [Digitize/Point](#) selection in the plot view menu.
- Observed_Horizontal_Velocity is the target magnitude of the horizontal component of the average linear velocity.

- Observed_Horizontal_Direction is the target direction of the horizontal component of the average linear velocity. This is measured in degrees from the x axis, positive in the counter-clockwise direction.
- Observed_Vertical_Velocity is the target vertical component of the average linear velocity. Positive is upward, negative is downward.
- Time is the time of the observation for a transient model. This time is measured from time = 0 at the start of the simulation. In a steady model, this parameter is ignored. In a transient model, you may input head targets at the same location but different times. The [Head Difference](#) calibration feature may be used to calibrate to temporal variations in head (e.g. rate of drawdown) during a transient simulation, in addition to spatial head differences.
- Show - if this is checked, this particular calibration point data will display when you [write calibration data to the run log](#) and will display on the plot when a plot is made with [What to Plot / Head Calibration](#). This allows you to make a plot with just certain calibration points posted (e.g. those in a one domain, or one model level, or at a certain time in a transient run).

This option is useful for calibrating a model to a known discharge direction indicated by an observed chemical plume. You can specify the direction of the observed horizontal velocity parallel to the observed plume and compare this to the modeled horizontal velocity direction. If you also know the magnitude of velocity in the plume, you can compare modeled to observed horizontal and vertical velocity components.

Head-specified Well Discharges

Here you select one or more head-specified wells to compare their simulated total discharge to some target or observed value of total discharge. This could be used to calibrate to an observed total discharge from a group of flowing artesian wells, for example.

- Label is the label give to this well or group of wells.
- Wells_Selected shows you a list of the head-specified wells from which you select the one(s) that you would like to calibrate to.
- Observed_Discharge is the observed or target discharge of this well or group of wells.
- Time is the time in a transient simulation (not used in a steady simulation).

Internal Line Boundary Discharges

Here you select one or more internal discharging line boundaries ([head-specified internal](#) or [river](#)) to compare their simulated total discharge to some target or observed value of total discharge. This could be used to calibrate to an observed discharge at a stream gauge or discharge from a pit dewatering system, for example.

- Label is the label give to this well or group of wells.

- `LineBoundaries_Selected` shows you a list of the internal head-specified line boundaries and river line boundaries from which you select the one(s) that you would like to calibrate to.
- `Observed_Discharge` is the observed or target discharge of this line boundary or group of line boundaries.
- `Time` is the time in a transient simulation (not used in a steady simulation).

Vertical Leakage Over Polygon Area

This feature allows numerical computation of vertical leakage discharges integrated over a polygon area. The area may be defined as the area covered by a specified domain or the area covered by a user-defined polygon. The discharges are computed by evaluating the vertical leakage rate $[L/T]$ at a large number of regularly-spaced points, and numerically integrating contributions over the small increments of area each point represents. The total leakage discharge $[L^3/T]$ over the area is reported. This tool will examine vertical leakages for each level that exists within the domain or polygon specified. For example, if levels 1,2,3, and 5 exist in the area of the polygon, the output will include:

- downward leakage into the topmost level of the model, regardless of whether the topmost level is level 1, 2, etc.
- downward leakage out of level 1
- downward leakage out of level 2
- downward leakage out of level 3
- downward leakage out of level 5

After solving, use [Analysis / Write Vertical Leakages Over Polygon Areas to Run Log](#) to view computed total leakage discharges. This output is also available in [automated runs](#).

Input parameters for this feature are:

- `Label` is a text label that helps you keep track of multiple polygon areas.
- `Polygon_Source` selects whether your polygon will be defined as the extent of a model Domain or by the Co-ordinates of a user-defined polygon, which could cross domain boundaries.
- `Points_Evaluated` is the approximate number of points evaluated (if your polygon is a rectangle parallel to the x-y axes, this number is the actual number of points, but if your polygon is irregularly-shaped, the actual number of points will be less than what is specified here. The algorithm sets up a spacing of points so that the specified number of points fills a rectangle that encompasses the entire polygon. More points yields more accurate estimates of integrated leakage discharge at the cost of more computation.
- `Domain` sets the domain that defines the polygon area, if you choose `Domain` as the `Polygon_Source`. If you choose `Coordinates` as the polygon source, this input will not be used and may be left blank.

- Coordinates are the coordinates of the polygon area, if you choose Co-ordinates as the Polygon_Source. If you choose Domain as the polygon source, this input will not be used and may be left blank. Click the Edit button to edit coordinates. The coordinates are multiple lines, each with two real numbers delimited by a comma (X, Y) to represent the vertex of the polygon. You may digitize the coordinates with the [Digitize/Polyline](#) selection in the plot view menu and then paste them in here. The coordinates may define a closed polygon (last point = first point), or it may omit the closing point (Anaqsim will close the polygon for you).

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.

Revision #7

Created 1 July 2023 17:35:38 by Mary Jeddere-Fisher

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