

GFLOW Trouble Shooting Guide

[General pointers to avoid trouble](#)

[Problems with base maps](#)

[Problems with printing](#)

[Large errors in results table](#)

[Conjunctive Surface and Groundwater Solutions do not converge](#)

[Missing or Incomplete Potentiometric Contours](#)

[Ragged Contour Lines](#)

[Inaccurate contours or path lines](#)

[Missing or Incomplete Path Line Traces](#)

[Path lines in MODPATH are too long or too short](#)

[Inaccurate or erroneous solution at horizontal barrier](#)

[A no-flow barrier lets water through](#)

[Fatal error reported by the Solver](#)

General pointers to avoid trouble

Most problems can be avoided by a step-wise approach to designing a model. The analytic element method is particularly suitable for step-wise modeling since the modeler does not have to commit to a grid or mesh design at the start of model construction. Define global aquifer properties based on data in your area of interest (*Model>Settings>Aquifer*). Introduce the main streams and lakes in near field and far field with no more than say 100 line-sinks. Add one recharge inhomogeneity, for instance a big rectangle with four line elements, over the far field and near field. -----Now try to solve! -----Test the plotting of potentiometric contours and pathlines. Add test points (points with known heads). Once everything works as expected take the following steps. Gradually build up the model, adding near field and far field features where necessary. Solve again. Add major inhomogeneity domains, solve after each one you entered. Add resistance to some critical line-sinks in the near field only. Solve again. Constantly solve while you build up complexity. Keep the model as small as reasonable. Small models solve faster and are more stable. Avoid unnecessary resistance line-sinks and stream features. They make the solution less stable! Think of the following:

- Line-sinks with very small resistance, e.g. 0.1 days or less, that are used for modeling stream segments can often be replaced by line-sinks with zero resistance. This improves model stability. **Note:** Line-sinks used to model horizontal wells or collectors for a radial collector well may require a resistance less than 0.1 to represent the resistance to 3D flow near the collector, see the document "Accounting for Resistance to 3D Flow.pdf," which is available from the Help menu under PDF documents.

- Line-sinks with very large resistance, e.g. 100 days or higher, can often be eliminated. This keeps the model small, thus faster and more stable.

- Head waters of streams are often losing stream sections. In case the stream is a "stream feature" in the model, the solver will drop line-sinks representing these head waters from the solution, but if you do not enter them in the first place the model will solve faster and be more stable.

- Use stream features (with conjunctive solution) sparingly, only in the near field and only where really needed. The solver has to work hard to sort out the stream flow and the process of establishing the proper surface water - groundwater interactions may cause solution instabilities.

- Changes in hydraulic conductivity are easier to solve than changes in the aquifer base

elevation. Do you really need the change in base or do you only want a change in transmissivity? In the latter case change the conductivity rather than the base elevation.

Note: for pathline tracing the residence time is influenced by the aquifer thickness, which may require a change in aquifer base elevation.

Problems with Base Maps

A base map, included in the project file, does not appear.

Check to see all maps are referenced to the same coordinate system.

The map may be outside the window. *Zoom to Extents* to see if the map shows up.

Use the *Reproject BBM* option on the *Tools* menu to bring all BBMs in a single UTM zone or State Plane coordinate system.

Base maps derived from DXF or DWG files are not positioned properly, they appear to have been shifted or rescaled.

This most often results from cutting and pasting layers in a CAD program. If you want to select one or more layers from a large DXF or DWG file and save them in separate files you must take special precautions to preserve the correct global coordinates, both the origin and the scale. The following procedure may help you accomplish this. Make a copy of the original file and open it in a CAD program. *Lock* the layer you want to store in another file. Next on the *Edit* menu *Select All* and *Delete*. This will leave you with the desired layer. Save the file under an appropriate name. You may repeat this procedure for other layers. If you want to combine the layers in a single file use the *Insert* option and specify the filename of the layer to be inserted. We found this procedure to preserve the origin and scale of the layers.

Maps may also occur in the wrong coordinate system or UTM zone. Use the *Reproject Maps..* option on the *Tools* menu to bring all maps in a single UTM zone or State Plane coordinate system.

Problems with Printing

Print option is not available.

You are running the *Educational Version*, which does not support printing. You may copy the graphics screen to the *Clipboard* and *Paste* it as a bitmap in a document. Select *Edit>Copy to Clipboard* and open the document in which you want to paste the graphics.

Printer option does not respond to *Portrait* and *Landscape* setting or *Paper Size* setting.

The GFLOW printer options are ignored under Windows NT and 2000. When printing under Windows NT or 2000 you must select these options directly for the network printer. Select *Start>Settings>Printer* and right-click on the network printer in use to select *Properties*. Click on *Preferences* and make the desired selections.

Large Errors in Results Table

The results table shows calculated and specified [heads](#) at [line-sinks](#) that differ significantly.

Check that no specified heads (for line-sink strings) are lower than the [aquifer base](#).

This is particularly important when you have changed the aquifer base elevation.

Check that specified heads are correct. Particularly, check that successive line-sink strings do not have a very large difference in head due to an input error.

A very likely error is the introduction of data in the wrong units!

Make sure you are not looking at line-sinks with a specified resistance. These must show differences in the *SpecifiedHead* and *CalculatedHead*. To verify the accuracy of these line-sinks look in the *Error* column, which shows the difference in the actual extraction rate of the line-sink and the one based on the difference in head across the resistance layer.

The results table shows large errors in the error column of [resistance specified line-sinks](#).

Large (relative) errors in resistance specified line-sinks may be due to the fact that the line-sink has a very low sink density (discharge), in which case the error is inconsequential.

Make sure that you do not have resistance specified line-sinks in the [far-field](#). All line-sinks in the far-field must have zero width, depth and resistance!

Your model may not have been solved completely. Select *Model>Continue Solve* for additional iterations.

The results table shows large errors in inhomogeneity domains.

Check that the heads near the domain are above the aquifer base.

Make sure you use sufficiently small line elements in areas where the head varies a lot. Make sure there are no sharp corners, particularly not in combination with large elements. Refine the domain by using smaller elements (adding vertices) and rounding off the corners.

Domains with a very low hydraulic conductivity tend to generate inaccurate solutions.

You may try to replace such a domain with a closed [Horizontal Barrier](#) with a zero hydraulic conductivity (no-flow boundary). In the latter case make sure to add a head specified well inside the barrier to ensure a well posed problem.

Conjunctive Surface Water and Groundwater Solution does not converge

Even after 10 or more iterations the stream flow solution (Solver messages in the Solver box and in the Run Time Message File) keeps reporting negative stream flows or under- or over-infiltrating line-sinks. Here are some things to consider.

A conjunctive surface water and groundwater solution requires a well- posed groundwater flow problem. For instance, if insufficient water is available to supply a well field, the aquifer near the well field will become dry (heads at the aquifer base) and an inaccurate surface water and groundwater solution may be the result. To check for this condition, uncheck *the Conjunctive Surface Water - Groundwater Solution* on the *Model>Setting>Solver* tab. Redo a solution (groundwater only) using 4 to 6 iterations and check for recharging stream sections in the [near field](#). If many stream sections recharge the aquifer, limiting that recharge to the available stream flow (during a conjunctive solution) may cause the aquifer to dry up.

Stream features (line-sink strings with the box *Use Streamflow Routing* checked on the *Routing* tab of the line-sink dialog box) should only occur in the [near field](#). Stream features in the [far field](#) are meaningless and almost certainly to cause solution instabilities, preventing the conjunctive solution to converge.

The Solver creates stream networks by coupling line-sinks strings that are defined as stream features as follows. If the *End Stream* box on the *Routing* tab of the line-sink dialog box is not checked, the Solver will link the downstream end of the string (end with the lowest specified head) to the nearest line-sink with a lower head that occurs in a nearby line-sink string that is declared a stream feature. Two errors may occur: (1) the line-sink string may be intended to form the end of a stream or stream network, but the *End Stream* box is inadvertently unchecked and (2) the specified heads along the line-sinks strings may be inaccurate causing an incorrect link. Verify these conditions in your stream network, even if the solution does converge. **Note:** You can visualize the stream linkages by checking *View>Show Stream Linkages*.. Red lines show line-sink connections, while black dots show where stream networks end.

Under certain conditions one or more line-sinks in a stream network may alternately be included and excluded during successive iterations. This prevents a completely converged solution, but the residual errors (e.g. negative stream flow) may be sufficiently small to accept the solution. The small residual negative stream flow may then be interpreted as zero stream flow. This situation is very likely to occur in large extensive stream networks, see for instance the example project file **Columbus.gfl** in the directory **example2**. The problem can sometimes be avoided by refining the stream network in the problem area, for instance by adding some vertices and improving the head distribution.

Inhomogeneity domains with a jump in the aquifer base result in non-linear equations under unconfined conditions. The more equations that are non-linear (resistance line-sinks, stream features, base jump inhomogeneities) the less stable the solution procedure. The solution procedure may be improved by providing an estimate of the average head along the perimeter of each base jump inhomogeneity. This average head may be set on the Inhomogeneity dialog box, check the box in front of *Provide estimate of average head*.

For large complex stream networks it may be necessary to apply an "under-relaxation" factor to dampen the stream flow solution process. Refer help on the Model>Settings>Solver tab sheet by [clicking here](#).

Missing or Incomplete Potentiometric Contours

I just ran my model, but I see no [contour lines](#) !

Verify that the *Contours* option is checked on the *View* menu.

Verify that the *Compute Contours* box under *Contour Type* on the *Contouring* tab sheet of the *Model>Settings* is checked and the radio button *Heads* is on.

Verify that the *Minimum Contours*, *Maximum Contours*, and *Contour Interval* under *Contour Levels* on the *Contouring* tab sheet are within the proper range of the expected [potentiometric head](#) values.

Verify that heads are above the aquifer base using the *Instant Inspector* feature; place the mouse pointer where you want to look at the head and do a *Shift left-click* to open a data panel with local properties and model output.

Ragged Contour Lines

Occasionally ragged [contour lines](#) occur after a solution. These may be due to (1) a potentiometric head surface that drops below the aquifer base, or (2) the presence of a [Horizontal Barrier](#).

Ragged contour lines surrounding a well or several wells are usually due to over pumping. The [heads](#) are drawn down to the [aquifer base](#) in the area surrounding the well, causing a few ragged contour lines with levels close to that of the aquifer base elevation. Make sure that you used the proper data in the proper units for the [aquifer thickness](#), [hydraulic conductivity](#), and pumping rate of the well (check your unit conversions).

Ragged contour lines may also occur close to a *Horizontal Barrier*. If the bundled up contours near the barrier show a regular pattern then they are usually not of any concern. The barrier generates a jump in the head across itself. The contouring routine, however, tries to make a smooth potentiometric head surface across the barrier, which results in bundled up contour lines that may be ragged close to the barrier. Irregular contours near barriers may, however, indicate a poor solution, see next topic.

Inaccurate contours or path lines

Problem: Contours appear to miss known mounds or cones of depression and path lines appear to miss expected turns or unexpectedly cross over features (e.g. a no-flow boundary). This problem is particularly evident when contouring and tracing in zoomed out views.

Cause: The resolution of the contour grid and the step size during path line traces are defaulted to values derived from the window dimensions. This default grid spacing and step size may be too coarse for the problem at hand, causing inaccurate results in the contour patterns and particle traces.

Remedy: To improve the contouring results, select the radial button Detailed on

Model>Settings>Contouring. This will increase the grid resolution from 40 horizontal points to 80 horizontal points. The number of vertical points in the grid depends on the aspect ratio of the window. GFLOW uses the same grid spacing horizontally and vertically. To improve the particle tracing results, deselect Use Default Step Size of on the Model>Settings>Tracing tab sheet and type in a smaller step size (e.g. factor 10 lower than default). Note: zooming in will automatically reduce the default grid spacing and the default step size and improve the accuracy of the contour plot and the particle traces.

Missing or Incomplete Path Line Traces

Verify that the *Show Paths* under the *Path Lines* option on the *View* menu is checked.

Verify that the box for *Compute Particle Paths* on the *Tracing* tab of the *Model>Settings* option is checked.

Make sure that there is a finite [saturated thickness](#) left near the well. In other words, that the well is not pumped dry with [heads](#) at or near the [aquifer base](#). The well will be colored [green](#) in that event!

Make sure that the [path line](#) or lines do not end in an area where the head is at or near the aquifer base. The path line would end there prematurely.

Make sure that a proper *Maximum Travel Time* has been specified on *Model>Settings>Tracing*. Too small a travel time may plot path lines too short to notice.

Remember that GFLOW traces particles in three-dimensions. Make sure that when tracing backward in time you set the starting elevation of the particle [below](#) the saturated aquifer top. Under confined flow conditions this means underneath the aquifer top. Under unconfined flow conditions this means underneath the water table (head in the model). When tracing back in time a particle at the aquifer top wants to leave the aquifer, moving against the incoming recharge. This leads to an abort of the particle trace after the first step. To prevent the particle to reach the surface, set the particle starting elevation equal to the (local) aquifer base elevation.

Path lines in MODPATH are too long or too short

Problem: After extracting a MODFLOW model from GFLOW and using MODPATH to trace particle paths, the path lines for a given time of travel are either longer or shorter than those in the original GFLOW model.

Cause: MODFLOW is a groundwater flow model that does not use the porosity of the aquifer for its calculations. In view of this MODFLOW does not import porosity values for its cells. Depending on the default values in the MODFLOW/MODPATH graphical user interface the porosity values provided to MODPATH may be larger or smaller than what was specified in GFLOW. As a result the path line traces for a particular time of travel are shorter or longer, respectively, than those in GFLOW.

Remedy: Manually set the porosity in the MODFLOW/MODPATH graphical user interface to correspond with the value or values in GFLOW. Be aware that inhomogeneity domains in GFLOW may or may not have been given differing porosity values, which must be manually duplicated in the MODFLOW/MODPATH model.

Inaccurate or erroneous solution at horizontal barrier

Inaccurate solutions near barriers manifest themselves by groups of circular or elliptical potentiometric head contours near sections of the barrier. Streamlines may weave back and forth across the barrier. In some cases the aquifer falls dry at one or more places near the barrier. Do not mistake a regular pattern of bundled up potentiometric contours for a poor solution! Barriers do generate a jump in the head across the barrier, which is manifested by bundled up contours. A good indication of a poor solution is when the error for the *total flow across the barrier* varies a lot between iterations and does not converge on a single value (see the *Run Time Message* file by selecting it from the *View Model Run Files* option on the *Model* menu). These inaccurate solutions do not imply that the specified conditions along the barriers are not met. Consequently, the % errors reported for the successive iterations may be rather low.

Below follow some suggestions to avoid or fix poor solutions.

Make sure that open barriers have smaller elements near the ends. Refine the barrier by adding vertices near the ends.

Make sure that open or closed **leaky** barriers have sufficient elements, particularly when specifying a relatively high conductivity or when the barrier is partially penetrating. Refine the barrier by adding vertices and round off sharp corners.

Avoid crossing barriers with inhomogeneity domain boundaries. If you do need to make such a crossing, use progressively smaller elements near the crossing for both the barrier and the inhomogeneity.

A no-flow barrier lets water through

Make sure the barrier is fully penetrating. The default bottom elevation on the barrier dialog box is equal to the aquifer base as set on the *Aquifer* tab on *Model>Settings*. However, when the barrier occurs inside an inhomogeneity with a lower aquifer base the bottom of the barrier must be lowered accordingly to keep it fully penetrating.

Fatal error reported by the Solver

Check that inhomogeneity domains or closed horizontal barriers do not have overlapping section or coinciding vertices. This is often caused by not properly entering the domain. Closed domains are entered by left-clicking at locations where a vertex is to occur. The last vertex to be entered is the one from which a line element extends to the first vertex entered to close the domain. DO NOT MOVE THE CURSOR TO THE STARTING VERTEX AND LEFT_CLICK!! This will create a duplicate vertex. Instead act as follows, after having entered the last vertex with a left-click, right-click on the same vertex to close the domain. To search for duplicate vertices zoom in on that part of the domain boundary where the first vertex is and check that there are no unintended vertices that cause the domain to overlap itself.

Closed no-flow barriers create two independent flow domains: the inside domain and the outside domain. Both domains must have at least one head specified condition. Consequently, when creating a closed no-flow barrier you should make sure there is at least one head specified feature inside. Failure to do so will lead to a singular set of equations and a failed solution process. When using a closed no-flow barrier to model a (very) low permeable area, simply place a head specified well inside with the average expected head in the area of the barrier. This will lead to a well posed problem and only small jumps in the head across the barrier.