

Frequently Asked Questions

Why does GFLOW not have a Save command? GFLOW is a database program. This means that data management does not occur in RAM to be saved to disk by an explicit user action (clicking on *Save*), but any data modifications in GFLOW are immediately implemented in a database, which resides on the disk. This is similar to the way the database program works. Consequently, GFLOW has no *Save* option, but you will find a *Make Duplicate Database* option on the *Project* menu. If you want to make changes in a model and not lose the current model, you must make a duplicate database under a new name before you make your modifications. The database on disk is also referred to as the *project file* and has the extension **.gfl**.

The graphical user interface (GUI) uses various additional files to communicate with the Solver. For each project (file) you should also define a unique *Base Filename*, which is done on the *Project Settings* dialog box (*Project>Project Settings*).

Why are my base maps not lining up? After defining a set of background map files on the project menu, two post stamp size maps occur on two sides of the map window in GFLOW (after a zoom to extent). This is most likely due to the fact that the files represent maps in two different UTM zones. To fix this you should use a GIS software product to reproject these maps into a single consistent coordinate system.

How do I apply areal recharge due to precipitation? Areal recharge due to precipitation may vary over the model domain. You should think in terms of a background recharge rate, applied over the entire model area, and local adjustments to it. After the [near field](#) and [far field](#) line-sinks have been entered you will have a good idea of the extent of the model area. Place a big (e.g. rectangular) inhomogeneity over the entire model area with the background recharge rate and the (effective) porosity as the only input parameters. The porosity will default to the one specified on the *Aquifer* tab (*Models>Settings>Aquifer*). You can use four big line elements to define the area. Make sure you include all line-sinks (you cannot make the area too large). Next, add or subtract recharge in areas where you want a different recharge rate. You do this by defining inhomogeneity domains with the recharge set to what you want to add or subtract.

Note 1: When entering inhomogeneity domains with a different hydraulic conductivity or aquifer base elevation than default, you can add or subtract recharge at the same time.

Note 2: The recharge specified for inhomogeneity domains are cumulative; in nested domains these recharge rates will be added.

How do I represent wide streams, lakes or wetlands with line-sinks? This may depend on where the surface water feature occurs. In the near-field, line-sinks should be placed along the perimeter of the surface water feature, whether a lake or a wide stream. In the far-field, streams and small lakes or wetlands may be represented with line-sinks at their centers, while larger lakes and wetlands may still require a few line-sinks around their perimeter. The idea for the near-field lakes or wetlands is that groundwater will enter the lake (wetland) near its boundary, thus that is where the line-sinks should be. In the event that the lake, wetland or stream bottom offers resistance to groundwater inflow or outflow, line-sinks with resistance should be used. The width parameter is related to the "effective leakage zone" underneath the surface water. You may calculate this zone manually or (recommended) enter the actual stream width and select as a line-sink location on the *Linesink String Properties* dialog that it occurs along the center line of a stream or along its boundary. In the latter case the Solver will

calculate the "effective leakage zone." A more detailed discussion on this matter is found in the pdf document: "Dealing with Resistance to Flow into Surface Waters.pdf" in the documents folder.

What does "Treat as "far-field"" mean on the Linesink String Properties dialog? The area of interest in your model must be surrounded by surface water features (line-sinks) that define the "far-field" hydrology in the model. These line-sinks on the perimeter of your model area perform a similar function as the conditions specified on the perimeter of a finite difference grid. Line-sinks in the far-field receive water or infiltrate water based on arbitrary conditions outside the model domain. Consequently, these line-sinks do not realistically represent surface water and groundwater interactions and should not be given a bottom resistance, width and depth parameter. Instead, they should simply maintain the specified head, regardless of the amount of water that must be extracted or infiltrated. By checking the box "Treat as "far-field" on the line-sink dialog box all fields, except for the starting and ending heads are disabled. In summary, all line-sinks on the outside of the model domain must be checked as "Treat as "far-field"". For further reading on the matter see "Analytic Element Modeling of Groundwater Flow" by H. M. Haitjema, Academic Press, 1995, Section 5.1.1 page 207.

How can GFLOW produce three-dimensional pathlines? While GFLOW is a so-called Dupuit-Forchheimer model, which uses two-dimensional flow equations, the third (vertical) component of flow can be estimated from water balance considerations. In case the aquifer has a constant transmissivity and a constant recharge rate the vertical component of flow in GFLOW appears to be the same as found from a truly three-dimensional model. In all other cases the vertical component of flow in GFLOW is approximate. For most cases of regional flow the approximation is very accurate. For further reading on the matter see "Analytic Element Modeling of Groundwater Flow" by H. M. Haitjema, Academic Press, 1995, Section 3.5 (page 140).

How do I make conjunctive surface water and groundwater solutions? Decide which streams (line-sink strings) you want to include in the surface water calculations. These streams or stream sections must be in the [near field](#). You can calculate stream flow in a single stream or in a stream network consisting of a main stream and tributaries. On the *Routing* tab of the line-sink dialog box select "*Use streamflow routing*". Make sure that the last line-sink string in a stream network has the "End Stream" box checked. The Solver will automatically link the line-sink strings into one or more stream networks, which end at the "End Stream" strings. Check the box "Conjunctive Surface Water - Groundwater Solution" on the *Solver* tab of the *Model>Settings*.

Note 1: Stream flow routing is performed under steady state conditions using groundwater inflows and outflows for the line-sinks. When specified "Overland Flow" will be added.

Note 2: The "End Inflow" option allows for the specification of a known existing stream flow at the start of a stream network in the model. This allows for including streams in a stream network that extend (on the upstream end) outside of the model domain.

Note 3: Stream flow routing should never be attempted in the [far field](#).

Note 4: The Solver connects line-sink strings that are defined as stream features ("Use Stream flow Routing" specified) to the nearest line-sink in a string (stream feature) that has a lower water level (head) specified. Consequently, it is important that the heads for the line-sink strings are accurately defined.

Note 5: Under unrealistic hydrological conditions in the model (e.g. insufficient water to supply wells) the conjunctive solution procedure may not converge.

Note 6: Linesink strings that are part of a stream network (have the *Use Streamflow Routing* box checked on the *Routing tab*) will be colored dark brown. After a solution is obtained you can visualize the stream linkages (to check the integrity of the stream network) using *View>Show Stream Linkages..* You should not see long red lines connecting the wrong tributary to the wrong stream.

For further reading on the matter see "Analytic Element Modeling of Groundwater Flow" by H. M. Haitjema, Academic Press, 1995, Section 5.2.4 page 239 - 241.

How do I extract a MODFLOW model from GFLOW? Before extracting a MODFLOW model you must have a calibrated regional GFLOW solution with all hydrologic features included. For instance, when modeling flow due to a well field, all wells with the correct discharge must be in the GFLOW model. Open the *Finite Difference Grid* dialog box (GRID menu) and define all relevant parameters. Note the dimensions, which are all in feet or all in meters. Once you have positioned the grid over the model domain, you may refine it using the *Edit* options. Finally, go to the *Processing* tab to perform three actions in succession: (1) apply the analytic elements, (2) extract GFLOW results, and (3) make MODFLOW files. These actions require the presence of a valid GFLOW solution. The MODFLOW data files can in principle be imported in any MODFLOW preprocessor. This has been tested extensively using the *GWVistas* code from EnviroSolutions, Inc. Using a MODFLOW preprocessor (e.g. *GWVistas*) you can further refine the model.

Note 1: Always first reproduce the GFLOW solution to make sure all data are correctly transferred. Be sure to set the porosity parameter in the MODFLOW preprocessor, as this parameter is not part of the MODFLOW data files, but will be used by particle tracing programs (e.g. MODPATH).

Note 2: If you add or remove water from the MODFLOW model, for instance, change recharge or change pumping rates, the conditions on the grid perimeter may not be accurate anymore. In case you specified flux conditions on the grid perimeter you may check to see that the heads at the perimeter remain approximately the same as in the GFLOW model. If not you must apply the same changes in recharge or pumping to the GFLOW model and extract updated perimeter conditions.

Note 3: The problem discussed in Note 2 can often be avoided by selecting the grid large enough to include some far-field conditions.

How can I perform automatic parameter estimation? GFLOW facilitates the definition of symbolic names for several model parameters: regional and local hydraulic conductivity, aquifer base, recharge rate, and resistance to flow between the aquifer and surface waters. You will find *Scenarios tabs* on the *Model>Settings* menu, *Linesink String Properties* dialog, and *Inhomogeneity Properties* dialog. Enter the symbolic names and check the *Include in scenarios* check box for those parameters you want to include in the automatic parameter optimization. GFLOW supports the automatic creation of PEST control files, with symbolic names and calibration targets automatically incorporated, see *Tools>PEST*.