Modeling with GFLOW

Beginning a Modeling Project
Model Calibration
Hypothesis Testing
Presenting Results

Beginning a Modeling Project

This is the most difficult moment of any groundwater modeling project: How to get started? On the table are some maps and perhaps a report of an earlier hydrogeologic study, but how do I use GFLOW to set up a model and create the required time of travel capture zones?

It will be helpful to break up the task in a few steps:

Step 1: Get a base map of the model area into GFLOW .
Step 2: Annotate the base map with water levels for the surface water features (streams and lakes) in the model area. This step is optional.
Step 3: Decide on a conceptual model for the aquifer (aquifer base, aquifer properties, transitions in aquifer thickness or hydraulic conductivity, aquifer recharge, etc.).
Step 4: Decide what part of the model area is near-field and what part you will include as far-field .
Step 5: Creating Line-sink in the Near-field and Far-field, crudely representing the streams in the far-field and more carefully representing them in the near-field.
Step 6: Enter estimated aquifer properties in GFLOW . When aquifer properties vary regionally (they will) start by entering those that apply to the near-field in the model. Later you can assign different aquifer properties to sub-domains of the model area.
Step 7: Run the model in GFLOW and display a potentiometric contour map that includes much of the far-field. Review the contour map to see that you have a reasonable groundwater flow solution consistent with the data you entered. At this time your solution does not yet have to accurately reflect what you know about the flow regime in the area.

After these initial steps you are up and running, you have accomplished the task of setting up a modeling project. The next phase is to use the model in conjunction with available field data to improve its realism, to represent the flow regime in the near-field as accurately as you can. Before discussing the procedure of modeling, however, let's elaborate on the steps above.

Menu Previous Next

Step 1: Get a Base map of the Model Area into GFLOW

You may use a variety of different map files that are either raster graphics files or vector files. Example file extensions are: DXF, SHP, tiff, jpg, dis, and .bmp. Each of these map image files must be **georeferenced** to work correctly in GFLOW. If working in the USA you may use USGS DLG files that have been converted to a proprietary format .bbm. These .bbm files may be obtained from the USEPA at

`https://www.epa.gov/exposure-assessment-models/whaem2000-bbm-files-us`

You can graphically locate the area for your bbm maps or click on a state and then quad name below the map of the US and state, respectively.
Select to download the map or maps to the download folder on your computer. The file that is being downloaded is a self-extracting zip file with the extension .exe. You will have to overrule the Windows security warnings and select "run anyway" when trying to run the executable.

IMPORTANT: EPA is in the process of converting old 16-bit self-extract files, which fail to run on 64-bit machines to newer 64-bit executables. In case the xxxx.exe fails to run (16-bit version), simply right-click the file and select Winzip or some other unzipping program on your computer to force the extract process.

You may use an existing map in DXF format, or create a DXF file by digitizing a map (e.g. using AUTOCAD). Similarly, you may use existing or create new Shape files in e.g. ArcView. The DXF and When creating DXF files it is useful to use different files for different map features. For instance, store roads in a separate file from the "hydrography" (all surface water features). You may create additional files with legal boundaries, geological information, etc. It is of course necessary for all maps to be referenced to the same coordinate system! We suggest you use UTM coordinates, which are found on most topographic maps. Note: if you digitize streams it will be useful to add tick-marks at stream locations where topographic contour lines cross the stream (see step 2).
Once you have the proper base maps in the working directory, start GFLOW, go to the *Project* menu, select *New Database* and type in a file name for your project. Next you will be prompted for the units to be used during modeling.
Note: You may select the base map units independently from the units used for defining heads, pumping rates of wells etc. The .bbm files are in UTM coordinates (meters). After opening your newly created project file you must add base map files to the project file. Select one or more of the base map files you created or downloaded.

Menu Previous Next

Step 2: Annotate the Base map with Water Levels

This step is optional. With water levels written next to streams (lakes) you can create Line-sinks (step 5) more easily, since you will be prompted for these water levels. In GFLOW, with the project file loaded, hence with the base map displayed, click on the *Edit* menu and click on *Add Text*. Move the cursor to a location on a stream where you know the surface water level, for instance, a location where a topographic contour line crosses the stream. If you digitized the map yourself, you may already have marked these locations with tick-marks (see step 1). Using a topographic map determine the water level and type it in the dialogue box (opens after clicking the left mouse button at the location where you want to put the water level). Make sure to select the *Hydrography Label* option in the *Text Label Properties* box. This will allow you later to select or deselect these water level markings. You will only want to see them on the map during the creation of line-sink strings. Otherwise your map may be overcrowded with labels. Continue to mark the map with water levels along all streams and lakes you intend to include in the model.

Menu Previous Next

Step 3: Deciding on a Conceptual Model

Now you must replace the real aquifer or aquifer system by a single horizontal aquifer, which may be locally confined or unconfined. This can be an agonizing task! Most geological formations don't resemble a homogeneous sandbox with a horizontal base, and representing them that way seems an absurd oversimplification. However, you should learn to think about the **hydraulic behavior** of the aquifer system, instead of its **geologic appearance**. Does it really matter if groundwater on its way to a stream moves in a homogeneous sandbox or is finding its way through a myriad of sand, gravel, and clay zones? Of course, locally it does matter, but on a sufficiently large scale only the average transmissivity of the aquifer and the arrangement of hydrologic boundaries (e.g. streams) determine the potentiometric head distribution and groundwater flow rates. You can include some of the variations in aquifer properties by defining inhomogeneities in the near-field. However, it is best to keep the model as simple as possible in this initial modeling step.

Note: Potentiometric heads and groundwater flow rates appear rather insensitive to the schematic aquifer representations used in most models. However, this is less true for groundwater velocities and travel times (residence times). Time-of-travel capture zones in GFLOW, therefore, must be interpreted as approximate.

Menu Previous Next

Step 4: Defining Near-Field and Far-Field

GFLOW, like any other analytic element code, models flow in a domain of infinite extent. Obviously, the area of interest is much smaller. Before you start to overlay streams with line-sink strings it is important to decide what streams are near-field features and what streams are far-field features. To explain this, assume you are trying to delineate a capture zone for a well or well field. Streams, lakes, wetlands, etc. that are close to the well or well field are most likely near-field features. You may distinguish between near-field and far-field features by use of the following thought experiment. Imagine you are standing at the well field and are able to see all surface water features in the model area (you can somehow see through all obstructions). For all possible directions, identify the nearest surface water feature; it is a near-field feature. All surface water features beyond these near-field surface waters are considered far-field features. The further away, the less critical they are for the purpose of our modeling: delineating the capture zone for the well or well field.

Note: For an accurate near-field representation it may often be necessary to include some of the features beyond the nearest surface waters as near-field as well. This is particularly true if the nearest surface waters are small and exert limited control on the groundwater flow regime.

Menu Previous Next

Step 5: Creating Line-sink in the Near-Field and Far-Field

The streams, ditches, lakes, and wetlands in the near-field are represented by line-sinks in much greater detail than the far-field features. In using more line-sinks for a particular stream reach, the geometry of the stream is represented more accurately, but equally important the variation in groundwater inflow (or outflow) along the stream (or lake boundary) is also represented more accurately. The far-field features, on the other hand, do not need to be modeled in great detail. Only major surface water features need to be included in the far-field, which you can represent with rather long line-sinks. Their purpose in the model is to create the proper heads on the outside of the area of interest. While we expect the near-field line-sinks to behave hydrologically in a realistic manner; extracting or infiltrating the correct amount of water, the far-field line-sinks may extract or infiltrate any amount of water, just as long as they control the potentiometric heads there. The far-field features play a similar role in our model as the grid boundary of a finite difference or finite element grid.

Important: Use only line-sinks with zero resistance, width and depth in the far-field. Never use stream flow routing in the far-field. In fact, the GFLOW graphical user interface forces you to explicitly define a line-sink string as a far-field feature or as a near-field feature. When you check the box *Treat as far-field* all attributes other then the *Starting Head* and *Ending Head* are disabled.

Note: Enter line-sink string vertices in down-gradient direction, hence starting at the headwaters and moving down the stream. Consequently, the *Starting Head* on the *General tab* of the *Linesink Properties* menu should be larger then the *Ending Head*.
Work in steps. Do not define too many line-sinks for your initial run. Leave out some less essential features and do not make your line-sinks too small. It is better to first create a simple stable solution and then refine and add features. For wide streams in the immediate area of interest you may consider using a double string of line-sinks, one string along each of the stream boundaries. Do not use stream flow routing in your initial model. It is better to work in steps and try conjunctive surface water and groundwater solutions only after you have successfully solved the groundwater problem by itself.

Adding Resistance to flow to line-sink strings.
In the near-field it may become important to account for the fact that a stream or lake is not fully connected to the underlying aquifer. A resistance layer may occur below the surface water feature formed by stream or lake sediments or a low permeable geologic formation. The *Resistance* parameter on the *General tab* of the *Linesink Properties* dialog is defined as the thickness of the resistance layer divided by its (vertical) hydraulic conductivity.
The *Width* parameter is related to an "effective leakage zone" of the stream or lake, which value must be selected such that the proper inflow rate into the surface water body is ensured. This concept of an effective leakage zone is discussed in the document *"Dealing with resistance to flow into surface waters.pdf*" available in the documents folder and also accessible from the *Help* menu. To review a summary of how to calculate this effective leakage zone click here. However, it is better to let the Solver automatically calculate this zone. If you place line-sinks along surface water boundaries (e.g. lake or both sides of a stream) select for *Location of Linesink* the radio button "*Along surface water boundary*." If you place line-sinks along the center of a stream select "*Along stream centerline*." In both these cases you should enter the actual stream or lake width and the Solver will automatically calculate the "effective leakage zone." If you select the radio button *Unknown*, you must manually calculate the "effective leakage zone" and enter it as the *width* parameter.
The *Depth* parameter is defined as the distance between the surface water elevation and the bottom elevation of the resistance layer. It is used by the Solver to determine when "percolating" conditions occur for recharging line-sinks and to calculate the effective leakage zone discussed above.

Inspecting the line-sink layout on screen.
Once you have created an initial model representing streams and lakes with line-sinks it is useful to zoom out and verify the proper definition of the line-sink strings. All near-field line-sinks show up as dark blue (assuming you did not yet define stream flow routing, which you should not at this initial step). All far-field line-sinks will show up as dark green. Make sure that the blue line-sinks (forming the near-field domain) are surrounded by green line-sinks (forming the far-field domain). It is OK to have some green line-sink in the near-field, which merely means that you consider the surface waters they represent fully connected to the aquifer (resistance, width and depth parameter set to zero). However, you should not have blue line-sinks near the perimeter of the model area (between or outside the green farfield line-sinks). If so you must either expand the farfield or check the box "Treat as Farfield" for these remote blue line-sinks to make them farfield features (resistance, width and depth set to zero).

Menu Previous Next

Assigning Aquifer Properties

The simple conceptual model of the real aquifer system forces us to make some difficult decisions on selecting aquifer properties. Most parameters, such as aquifer base, aquifer top, hydraulic conductivity, etc. will vary significantly over the model domain. Selecting a true average value for the whole model domain or for part of it may not be very meaningful (even if you know how to do such a thing). Initially you may select only one set of properties for the entire model domain and after some initial model runs define inhomogeneities to account for major variations in aquifer properties and recharge. This procedure may be unsatisfactory if a significant geological contrast occurs. For instance, a well field may be located inside the so-called "channel deposits" of a river, which means that the aquifer transmissivity reduces sharply at some distance away from the river, outside the channel deposits. In that case, you may have to define an inhomogeneity that represents the channel deposits right from the start. The properties outside the channel deposits are then entered under the *Aquifer* tab of the *Settings* option on the *Model* menu.

To define a regional areal recharge rate due to precipitation you must create an inhomogeneity domain that includes all of the model area, near-field and far-field. Zoom out so that all near-field and far-field features are displayed on the screen. Click on the inhomogeneity icon on the tool bar (or select *Inhomogeneity* from the *Element* menu). Only check the box *Change recharge from default* and enter the desired recharge in feet per day. For instance, 1 inch per year is 0.00022831 feet per day. Select a color and click *OK.* Create a rectangle around the domain using four large line elements. Right-click to close the domain when having entered the fourth vertex.

Note: At this point it is not fruitful to agonize too much over the precise choice of parameters. Just make sure that the aquifer bottom is not defined above water elevations in streams or lakes in the model area. This would, of course, lead to a conflict during the solution procedure. Your parameter choices are only a first estimate, you will later update the values during calibration or hypothesis testing.

Let's first get this model to run.

Menu Previous Next

Step 7: Run teh Groundwater Flow Model

We now entered enough data to look at some potentiometric contours in the area of interest. Select *Model>Settings>Contouring* and check the *Compute Contours* box, and click on the radio button for *Heads*. Under *Contour levels* select minimum, maximum levels, and increment in an appropriate range. Under *Grid resolution* check *Coarse.* Click on the calculator icon on the Tool bar. A black box appears with some progress indicators of the Solver (Solver runs in the DOS box). When done the base map is redrawn with on top of it potentiometric contours. It is a good idea to look at the *Results Table* at this time (click on *View Results Table* on the *Model* menu and click on *View* and *Line-sinks*). It lists all line-sinks with calculated heads next to specified heads at the line-sink centers. These heads should be nearly the same for a quality solution, except if a non-zero resistance parameter is set. The percent error is listed in the most right-hand column. If satisfied, close the *Results Table* box and look at the potentiometric contour plot. You should be looking at potentiometric contours that show some mounding in between major streams (line-sink strings). Verify that the contour levels are reasonable, for instance, that they do not exceed the terrain elevations. If you want to reduce or increase the number of contour lines change the contour increment on *Model>Settings>Contouring*.
If the solution is not accurate, increase the number of iterations on *Model>Settings>Solver*. Three to 5 iterations should be sufficient for most initial models.

Inspecting model input and output at a point in the domain.
During the modeling activities it is often useful to know one or more parameters at a particular point in the flow domain. For instance, you may want to know what the specified hydraulic conductivity or aquifer base elevation is at a point. Or you may want to know what the calculated head or groundwater velocity is at that point. GFLOW allows you quick access to these data as follows.
Move the cursor to the point in question and hold down the *Shift key* while clicking the *left mouse button*. The Solver will be launched to generate all relevant input and output data at that location, which will be displayed in a data box on the screen. You may repeat this procedure at any point in the domain. **Note**: The reported recharge rate is the sum of all infiltration and extraction rates specified at the point in question (e.g. due to nested inhomogeneity domains).

Menu Previous Next

Calibrating the Model

After having set up the initial model, adjustments in parameter choices and analytic element choices are needed to make the model as realistic as possible: often referred to as model calibration. At this point you should further detail the near-field. Any high capacity well (industrial wells or irrigation wells) in the model area should now also be included in GFLOW. Usually the only test of model realism is a comparison between known potentiometric heads and those calculated by the model. For instance, the model area may contain some piezometers installed by the USGS or consultants for the purpose of other groundwater flow studies. Less accurate, but more universally available are static water levels recorded at the time new wells (domestic or industrial) are drilled. You may also look at water levels in gravel pits or other small surface waters without a stream inlet or outlet. After you have defined some streams as part of a stream network, on the *Routing* tab of the *Linesink Properties* dialog box, you can also use average streamflow in the network as a calibration target.
Some topics relevant to the model calibration are:

Setting up points of observed heads (Test Points)
Adjusting for differences between modeled and observed heads
Interactions between surface water and groundwater

GFLOW also allows you to use scenario names for various model parameters, which can be used for manual or automatic model calibration. You may review the *Implement Scenarios* option on the *Model* menu. Also look at the *PEST* options on the *Tools* menu.

Menu Previous Next

Test Points

To facilitate a comparison between known (measured) heads and those calculated, you can introduce so-called "test points" in GFLOW. On the *Model* menu click on *Add Test Point*, move the cursor to the location of a piezometer, well, or gravel pit with known static water level and click the left mouse button. Select the radio button *Piezometer* and enter the known head and a label in the dialogue box and press *Enter*. A test point marker is plotted on the base map. When all available test points have been entered, click on *Solve* on the *Model* menu. When the base map is redrawn, triangles are plotted at the test point locations that point up or down for model heads that are too large or too small, respectively. In addition, the model head minus observed head is printed next to the test point. The display of test points can be modified on the *View* menu. See also "Heads too high or too low ".

Note: There are two additional Test Point types: *Gages* and *Lake Stages*. These are for use with conjunctive surface water - groundwater solutions and lake stage solutions, respectively.

Menu Previous Next

Heads to High or too Low

Calculated heads are expected to differ from observed heads for many reasons, most importantly because the model aquifer is merely an abstraction of the real aquifer system. A good model will show test point triangles that point both up and down, whereby the differences in head are not too large. A good model will also show a rather random distribution of up and down arrows, instead of having all up arrows clustered in one area and all down arrows in another.

In an ideal homogeneous single aquifer some generic rules apply for heads that are too high or too low. Heads that are consistently too low close to a high capacity well suggest too low a transmissivity in the model. Heads that are consistently too high in areas of groundwater mounding may also indicate too low a model transmissivity or too high an areal recharge rate (or both).
When the actual aquifer is significantly stratified or, in fact, consists of several vertically stacked interconnected aquifers, discrepancies between observed and calculated heads may also be explained based on the relative depth of the piezometers (wells) in the field. In most cases, shallow piezometers in a stratified aquifer tend to show higher heads than predicted in a homogeneous aquifer, while deep piezometers tend to show lower heads. Only close to discharge areas, such as streams or lakes, will this trend be reversed.

Heads that are too high or too low may also be the result of improper representation of surface waters, see Interactions between surface water and groundwater .

Menu Previous Next

Surface Water and Groundwater Interactions

One of the most difficult modeling issues is the interaction between surface water features and groundwater. In GFLOW the interaction is controlled by a resistance parameter in combination with a line-sink width and depth parameter. The depth parameter specifies the distance between the water level in the stream and the bottom of the resistance layer. The resistance is due to streambed deposits (silt) or lake bottom deposits which have a lower hydraulic conductivity than the aquifer material, or because the stream or lake occurs in a clay or silt layer that overlies the aquifer. Ideally, this resistance to flow should be determined (estimated) for all surface waters in the near-field and included in the model. In case a line-sink has a non-zero width parameter, the line-sink's infiltration rate (for a loosing stream) will be limited by the resistance to vertical flow. In case of a zero resistance the hydraulic conductivity of the aquifer forms the upper limit of the infiltration rate. Only when a line-sink has no resistance, no width and no depth will the line-sink be allowed to infiltrate unlimited amounts of water to keep the head in the aquifer equal to the water level in the stream. In the near-field this may not be acceptable, but in the far-field we want line-sinks to do just that: maintain the specified heads. Consequently, line-sinks in the far-field should be given a zero resistance, width and depth. Line-sinks in the far-field should, of course, never be part of a stream network with surface water routing. To ensure these conditions make sure that all line-sink strings in the far-field have the box *Treat as far-field* checked on the line-sink dialog box.

Menu Previous Next

Hypothesis Testing

Data uncertainty
Aquifer inhomogeneities
Horizontal Barriers
Transient flow: using bounding steady state solutions
Modeling beyond GFLOW: extracting a MODFLOW model

Menu Previous Next

Data Uncertainty

Field data such as hydraulic conductivity, recharge due to precipitation, etc. are not always accurately known. In fact, since in GFLOW we are introducing spatial and temporal averages for zones in the model domain, these data are by definition inaccurate. It will remain uncertain what choices for inhomogeneities and their properties will be "correct". At best, our field investigations and modeling will lead to some ranges within which we expect the optimum parameter values to occur. For instance, the representative hydraulic conductivity in some domain may be found to be most likely between 100 and 200 feet per day. And similarly the recharge rate due to precipitation may be found to be most likely between 10 and 15 inches per year. These ranges may have resulted from pumping tests, stream flow data and the observation that smaller or larger values, implemented in GFLOW, lead to unrealistic potentiometric contour patterns. Although it may seem reasonable to select median values for the "best fit" model, it would be unwise to ignore the uncertainty in the model parameters. An effective way to deal with these data uncertainties is to generate modeling results for combinations of extreme parameter choices within the established ranges. Assume that the modeling objective is to delineate a capture zone for a well. The variations (or lack there of) in capture zone configuration will provide information on the importance of uncertainty in various parameters. For instance, it may be found, depending on circumstances, that the uncertainty in hydraulic conductivity is of minor consequence for the time of travel capture zones, while variations in porosity lead to significantly different time of travel capture zones. In other cases, however, hydraulic conductivity differences may impact the capture zone shape or location. Always demonstrate the sensitivity of the model by showing the impact of parameter uncertainty on the relevant model output, that is the output that is the purpose of the modeling project. In other words, in the case of capture zone delineation it is not meaningful to show the effects of conductivity variations on the heads, when we are only interested in the robustness of the capture zone to uncertainty in the conductivity.

Menu Previous Next

Aquifer Inhomogeneities

To incorporate the heterogeneity of regional aquifers you may define domains with differing but constant aquifer properties and recharge rates due to precipitation. These inhomogeneity domains may be nested or abut each other, but may not overlap. An exception is an inhomogeneity domain that only adds to (or subtracts from) the areal recharge rate, leaving all other parameters unchanged. Such a "recharge inhomogeneity" may be placed everywhere and overlap any element or domain. Inside an inhomogeneity you may redefine the hydraulic conductivity, the elevation of the aquifer base and the aquifer porosity. A typical use of an inhomogeneity domain is to introduce channel deposits around a river in the near-field. While the actual channel deposits may extend into the far-field, it is sufficient to introduce an inhomogeneity domain that only represents that part of the channel deposits that occur in the near-field. In the case of the channel deposits it is likely that the hydraulic conductivity is (much) higher than the regional value, while the aquifer base may be lower (bed rock valley). The higher permeable and often flat channel deposit area also tends to exhibit a higher recharge rate due to precipitation. In GFLOW you can specify an "added recharge rate" for an inhomogeneity, hence the recharge is added to what may already have been specified at that point by another inhomogeneity.

To test the impact of aquifer heterogeneity you should make different choices for the geometry and properties of inhomogeneity domains and test their effect on the study results (e.g. a time of travel capture zone or flow direction of contaminated water). Keep the shape of the domains simple, unless you have clear evidence of some local detail. Usually, small differences in properties have little effect on the outcome of a study, but that may not be true under all circumstances. This is why you should experiment.

Menu Previous Next

Horizontal Barriers

Slurry walls or rock outcrops may form restrictions or barriers to (horizontal) flow. You may define one or more horizontal barriers to represent such features. GFLOW supports horizontal barriers that are open or closed strings of straight line elements (line doublets). A horizontal barrier may be given a width, depth, hydraulic conductivity and porosity. When the barrier is given a zero hydraulic conductivity and extends down to or below the aquifer base it will form a no-flow boundary.

It is sometimes easier to model a nearby very low permeable zone by placing a no-flow boundary along its perimeter than to define an inhomogeneity with a very low conductivity. In fact, it may be sufficient to only place a no-flow boundary (horizontal barrier with zero conductivity) along the near-field section of that boundary. After all we are not interested in the flow in the low permeable material, but only in its effect on the flow in the permeable near-field. If you have doubts about this, try it both ways; using a horizontal barrier or an inhomogeneity. Isolated rock outcrops, for instance, may be modeled by use of a closed domain horizontal barrier with zero conductivity. Slurry walls, used to block the migration of contaminants (open wall) or confine an area with contaminated water (closed wall) should be modeled by specifying the proper width, depth and conductivity of the slurry wall. In case the slurry wall partially penetrates the aquifer, the resistance to flow will be reduced due to the open zone underneath it. During particle tracking, a path line may go through the barrier or underneath it. The residence time of the particle that travels through the barrier is partly controlled by its porosity. **Important**: Horizontal barriers may require a rather high number of line elements (vertices). Open barriers should be given smaller line elements near their ends. Particularly for partially penetrating barriers, or barriers that are rather permeable, a rather large number of vertices may be needed. Too course a representation will manifest itself by clearly erroneous flow patterns near parts of the barrier. When you doubt the accuracy of the solution refine the barrier (add vertices) and resolve.

Menu Previous Next

Transient Flow: Using Bounding Steady State Solutions

One of the most troubling simplifications in our modeling is the representation of a groundwater flow regime that changes in time by a steady state model. For instance, recharge due to precipitation exhibits seasonal variations, and so do river stages in response. Depending on circumstances, aquifers may respond rather fast to these transient effects, exhibiting groundwater flow patterns in the summer and winter that seem consistent with the different recharge rates in summer and winter, respectively. If so, steady state solutions can be used to assess the range of flow patterns (and capture zones) that may occur during the seasons. On the other hand, if the aquifer responds rather slowly to transient effects, the actual groundwater flow patterns do not conform to either a "summer" or a "winter" steady state solution. The criteria for the use of bounding steady state solutions (summer and winter solutions) can be summarized by use of a characteristic factor, see Townley (1995) and Haitjema (1995). If the factor $SL^2/4TP$ is smaller than 1 the use of bounding steady state solutions is justified. The symbols are: S is the aquifer storage coefficient (close to the aquifer porosity for unconfined flow), L is the average distance between surface waters, T is the transmissivity, and P is the period of the transient forcing function, e.g. 365 days for seasonal variations. In practice, almost all confined aquifers and many highly permeable unconfined aquifers satisfy this condition, so that we can use bounding steady state solutions to represent extremes for transient seasonal flow.

Menu Previous Next

Modeling Beyond GFLOW

GFLOW is a single-aquifer steady state flow model, which may be too limited for accurately representing some detail in the near-field. For instance, there may be more than one aquifer in which case a multi-layer Dupuit-Forchheimer model, also referred to as a "quasi-three-dimensional model", is the appropriate modeling tool, not GFLOW. There may be local transient effects to be investigated. While GFLOW can model the effect of one or more wells that pump intermittently, it is not a generic transient flow model. Local three-dimensional effects may be important. While GFLOW can model three-dimensional flow near one or more partially penetrating wells, it is not a generic three-dimensional flow model either.

To facilitate the modeling of more local detail, the GFLOW graphical user interface allows the user to define a model grid on part of the GFLOW model domain. All internal boundary conditions (streams, lakes, wells, etc.) as well as all aquifer properties in GFLOW are automatically assigned to the **MODFLOW** grid cells. The cells on the grid boundary are assigned heads or fluxes obtained from the GFLOW solution. The data may be written to disk as a set of MODFLOW input data files, which may be entered directly into MODFLOW. In most cases you may want to use a preprocessor and postprocessor program to further detail the MODFLOW model and present its output. In this manner, GFLOW may serve as a screening model for a MODFLOW model, with calculated conditions on the MODFLOW grid boundary, rather than assumed conditions as is normally the case.

For further details on the use of the MODFLOW extract feature see the Grid menu.

Menu Previous Next

Presenting Results

Documenting Effects of Data Uncertainty
Generating DXF and Shapefile Maps
Generating SURFER Output
Generating Input Data Tables

Menu Previous

Documenting Effects of Data Uncertainty

It is important to present more than one potentiometric contour map, or time of travel capture zone, or whatever output the modeling objective dictates. The impact of the most sensitive parameters on the model output should be illustrated by presenting the differences and clearly explaining the different parameter choices that led to them. It is much more convincing to show that, for instance, a fifty feet per day difference in hydraulic conductivity does not noticeably change the capture zone than argue at length why the particular value chosen for the model is the right one. You may annotate the graphics (potentiometric contour maps or capture zones) directly on the screen by use of the *Add Text* option on the *Edit* menu. The maps may be printed using the *Printer setup* and *Print* options on the *Project* menu. You may also export the graphics in DXF format or as Shapefiles. In addition, you may export the potentiometric contour and path line data in SURFER format for further post processing, see *Export* option on *Tool* menu.

You may also perform parameter optimization on GFLOW models by use of PEST. Currently, GFLOW supports the automatic creation of PEST control files. To use this feature you must select and include symbolic names for the various parameters for which you want to optimize. You can define these symbolic names on the *Scenarios tab* of the following dialogs: *Model>Settings, Linesink String Properties,* and *Inhomogeneity Properties.*

Menu Previous Next

Generating DXF or Shapefile Maps

Instead of printing the maps directly to the printer, DXF files or Shapefiles containing the maps with contours and/or path lines may be created using the *Export* option on the *Tools* menu. You may also write the envelope of time of travel capture zones created with the *Draw Capture Zone* option on the *Edit* menu to a DXF file or Shapefile. DXF files may be imported in CAD programs for further editing and formatting. Shapefiles may be imported in ArcView or ArcInfo. Of course, you can also add exported contours or particle traces in the form of shapefiles or DXF files to the base maps on the *Project Settings* menu. This may be handy for the purpose of comparing different solutions to each other.

Menu Previous Next

Generating Surfer Output

You may output the grid of heads, velocities, and flow rates generated by the solver to a file in SURFER format, creating a **`filename.grd`** file. You may also output the contour lines or path lines in SURFER format, creating a **`filename.bln`** file. To perform these output functions select *Export* on the *Tools* menu.

Menu Previous Next

Generating Input Data Tables

Most input data can be presented in the form of a base map with superimposed the line-sink

strings, wells, inhomogeneities, etc.. When the base map has been annotated with water levels at points along the stream where surface elevation contours intersect the stream (Hydrography labels), a good overview of the applied boundary conditions is provided. The user may further annotate the basemap by use of text labels, see the *Add TextÖ* option on the *Edit* menu. The few aquifer parameters specified under the *Aquifer* tab of the S*ettings* option on the *Model* menu are easily summarized by hand.
The tables in the Results Table (*Model>View Results Table*) may be copied to the clipboard (*Edit* option) and pasted into a spreadsheet, e.g. in Microsoft Excel.

Menu Previous