

Tutorial

- [Setting Up a Project](#)
- [Selecting Base map Files](#)
- [Annotating the Base map](#)
- [Entering Aquifer Properties](#)
- [Entering Recharge](#)
- [Creating Analytic Elements](#)
- [Running the Model](#)
- [Checking the Solution Integrity](#)
- [Displaying Results](#)
- [Editing the Model](#)
- [Extracting a MODFLOW model](#)
- [Printing Graphics](#)
- [Additional Examples](#)

Note 1: For many of the menu options there is a short cut on the tool bar. At some point, after having loaded a project file, you may slowly move the mouse cursor over the various tool bar buttons and observe their functions (popup text). You can use these short cuts as alternatives to the menu options described in the tutorial or elsewhere in the Help system.

Note 2: The procedures outlined in the tutorial are implemented in the project file **example.gfl**, which may be inspected for further clarification.

Note 3: This tutorial illustrates only the most fundamental features of GFLOW. The user is encouraged to consult the on-line *Menus* section and read the documents (in PDF format) provided in the <user> application data folder *on the C: drive* in the subfolder *GFLOW*. Note: A precise path cannot be indicated as it differs per Windows version. These PDF documents can also be accessed from the *Help* menu.

Setting Up a Project

For the purpose of this tutorial you are provided with eight example vector graphics maps in the area of Vincennes, Indiana*. The vector graphics maps have the extension "bbm" and contain various layers of information in four areas surrounding the city of Vincennes. Maps with "hyf" in the name contain the "hydrography": streams, lakes, etc. Maps with "rdf" in the name contain the roads, with "rrf" in the name the rail roads and with "mtf" in the name some miscellaneous transportation features (power lines, pipelines, etc.).

The maps are to be found in the <user> documents folder on the *C: drive* in the subfolder **GFLOW projects**. Note: A precise path cannot be provided as it differs with Windows versions.

As a rule, when setting up a project, you must organize the relevant map files in your working folder. This is any user accessible folder in which you like to organize the files related to your modeling project. I recommend that you make this a subfolder of the *GFLOW projects* folder mentioned above. All these maps must be referenced to the same coordinate system, e.g. Universal Transverse Mercator (UTM) or State Plane, and must be in the same units - either meters or feet. Users in the USA may obtain digital versions of the USGS DLG files by downloading these from the EPA website:

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

These maps are downloaded as self-extracting zip files (xxxxx.exe), which may require the "run anyway" option to overrule Windows' security block. NOTE: EPA is in the process of converting

the 16-bit self-extracting files into 64-bit ones. If your xxxxx.exe fails to run, then simply right-click on it and select WinZip to force an extract of the maps.

You should also have completed a literature survey to obtain estimates for the elevation of the [aquifer bottom](#), elevation of the [aquifer top](#), [hydraulic conductivity](#), [effective aquifer porosity](#), and areal recharge rate due to precipitation. For the purpose of this tutorial these parameters will be given to you when discussing how to enter them into GFLOW.

*Note: You can also use alternative vector format files or raster format files, as indicated in the manual under *Menus*, *Project Menu*. We will not illustrate the use of these files in this tutorial, however.

[Main Menu](#) [Previous](#) [Next](#)

Selecting Base Map Files

Start GFLOW, click on the left most icon (blank sheet) or select *New Database* from the *Project* menu. In the dialog box for the project file name, navigate to your working directory (for this tutorial `\\GFLOW projects\\example`) and enter the filename `demo.gfl`, click *Open*.

The "New Database Wizard" will prompt you for a project description and the units associated with the input data. GFLOW requires consistent units, these are either meters and days or feet and days. For instance, if you select meters and days, the [hydraulic conductivity](#) must be entered in meters per day and a well pumping rate must be entered in cubic meters per day. However, the distance units associated with your base maps are specified independently from the "*Units for Computations*". In this way, you can accept base maps in meters (e.g. UTM coordinates), while using feet and days for all other input data or vice versa. The GUI will convert the base map units to the computational units when forwarding data to the solver, which works in consistent units throughout. We must select meters for the "*Distance Units in Base Map Files*", because the base maps provided with this tutorial are in UTM coordinates, while we will select feet and days as the "Units for Computations". The box "Project Description" allows you to keep track of some essential properties of this particular database, e.g. "Run with high recharge rate and low conductivity".

Important: You cannot select the "*Distance Units in Base Map Files*" freely; they must correspond to the units associated with your base map files!

After you made your selections click on *Create Database*.

The next window is the *Project Settings* dialog. The window "*Currently Used Base Maps*" will be empty at this point. Click on *Add Map* to display the available base map files. For this tutorial we will assume that `.bbm` files are used for base maps (contain USGS DLG maps). Users outside the USA may substitute the references to `.bbm` files elsewhere in this manual (not here in the tutorial) to the relevant file format for their project area (e.g. DXF, shapefile, tiff, etc.). Click on the first `*.bbm` file name and click on *Open*. Repeat this procedure for all `*.bbm` files you want to include. In most cases you have only copied the `.bbm` files to the working directory that need to be included. You may select them all at once by *left-clicking* on the first file and while holding down the *Shift* key *left-click* on the last file. After they all are highlighted click *Add Map*. All base maps should be displayed in the box "Currently Used Base Maps". Now click *OK*. The maps will be loaded and plotted in the graphics window.

Note: You can later select and deselect base map files for display on the *View* menu, selecting

Base Maps.. and *Vector Graphics* (or *Raster Graphics* if present). If you select *View>Base Map..>Select BBM Group*, then map boxes are drawn around each group of *.bbm files. A left-click inside a box brings up a **Group of BBM Maps** file management dialog box that let's you select, deselect or delete the group of files that make up the vector graphics image inside the selected map box. You can also select a single file or sub set of files by clicking on the others to deselect them (not highlighted). Terminate the BBM management mode by clicking on the button *Done Editing BBM Groups*.

Note: A *Help* file for the **Group of BBM Maps** dialog box can be obtained by pressing **F1**. The **F1** key will also display help files for other dialog boxes.

Note: You may find superimposed on the map a so-called editor grid. You may toggle this off or on by clicking on the *View* menu and deselecting or selecting *Show Editor Grid* under the *Base Map..* option.

[Main Menu](#) [Previous](#) [Next](#)

Annotating the Base Map

Zoom out using *Windows>Zoom Extents* or the *Zoom extents* button on the tool bar. The curved double blue line is the Wabash River in southwestern Indiana. The big city just below the center is Vincennes. Click on the button *Zoom to Window*, place the cursor to the upper left of the city and, while holding down the left mouse key, drag a box over the city and its surroundings.

It is useful to annotate your base map with estimated surface water elevations at those streams that you intend to include in the model (represent by [line-sinks](#)). This may be done by use of 7 1/2 minute USGS topographic maps as follows. (Refer to the second note below if you don't have these maps available).

1. Find the intersection of a surface elevation contour line (brown line) with the stream you want to annotate.
2. Estimate the location of that point on the map on your screen, click on *Add text* on the *Edit* menu, move the text cursor to the proper location on the stream and click the left mouse button.
3. Type the contour elevation in the dialog box, check the *Hydrography Label* option, and click *OK*.
4. The label appears on the map and the cursor reappears.
5. Repeat this procedure for all contour lines and all streams you intend to include in the model.

It is often sufficient to add these hydrography labels near the map boundaries and near confluences of streams. When creating line-sink strings you will only be prompted for a head at the start of the string and at the end of the string. The program will interpolate between these heads.

Note 1: It is important to enter all these water levels as *Hydrography Labels*, so that you can select them on the *View* menu as a map layer when creating line-sinks and make them disappear when done. This way the map on your screen is not cluttered with labels when modeling.

Note 2: An example annotated base map is provided in the project file **example.gfl**. You may open (*Open Database* on *Project* menu) this project file and activate the hydrography labels

(*View>Hydrography Labels*), and print the base map for guidance in marking up your map in **demo.gfl**. Prior to printing you may need to zoom out to include all of the labels. Hence, as an alternative to using the contour lines on USGS topographic maps, you may use the printout of hydrography labels from the **example.gfl** project as guidance to practice entering hydrography labels.

Note 3: The project file **example.gfl** contains a (crude) representation of the hydrogeology in the Vincennes, Indiana area. By no means does it represent a complete and realistic conceptual model of the aquifer system. It is provided here merely to provide some guidance in setting up a project. For a more complete description of groundwater flow modeling near Vincennes, Indiana, see Chapter 6, section 6.1 of *Analytic Element Modeling of Groundwater Flow*, H. M. Haitjema, Academic Press, 1995.

[Main Menu](#) [Previous](#) [Next](#)

Entering Aquifer Properties

Select *Settings* on the *Model* menu and click on the *Aquifer* tab. Enter the following data:

Base Elevation = 330

Thickness = 100

Hydraulic conductivity = 350

Porosity = 0.2

Click *OK* to enter these values in the database.

Note: We will not use the *Add Salt Water Interface* option for this tutorial, leave it unchecked.

[Main Menu](#) [Previous](#) [Next](#)

Adding Recharge

Recharge is introduced by use of an inhomogeneity domain. First zoom out so that the entire model area, including the [far-field](#), is depicted in the graphics window. Click on *Elements, New* and *Inhomogeneity*. Type "regional recharge element" in the *Label* box. Check the box "*Change Recharge from Default*" and enter 0.0032 in the box for *Added Recharge Rate*. Do not enter data in any other box and leave the *Change hydraulic conductivity from default*, *Change porosity from default* and *Change base elevation from default* boxes unchecked. Click *OK* and draw a rectangular box around the model area, including the far-field. Four large line elements are sufficient. Simply move the cursor to the corners of the desired box and click. After clicking on the last corner add a right-click and the box will be completed. **Important:** Do not move the cursor to the starting point and left-click, this would put two vertices on top of each other and cause computational problems.

If necessary, you can modify the recharge rate locally by defining additional inhomogeneities with only an *Added Recharge Rate* specified. Nested inhomogeneities with recharge are additive, the sum of the recharge rates are applied to the aquifer. At this point in our exercise we will work with only this one recharge rate for the entire model domain.

Note: Inhomogeneity domains may have common boundary sections, for instance to make a mosaic of different aquifer properties and recharge rates. Common boundary sections must have coinciding vertices, not just vertices that are close! To achieve this, you must hold down the CTRL key when placing a vertex on top of another vertex of another domain while left-

clicking the mouse. This will ensure that the vertices will truly coincide.

[Main Menu](#) [Previous](#) [Next](#)

Creating Analytic Elements

Line-sinks

To create [line-sinks](#), select *New* on the *Element* menu and click on *Linesink*. We will first enter line-sinks for Wabash River. Follow these steps:

1. Type "Wabash River west side" in the *Label* box. You should be on the tab sheet labeled *General* and the radial button for *Head-Specified* should be checked.
2. Enter the *Starting Head*. This is the water level in the stream at the start of the line-sink string you intend to create. In our case we will start our string just above the city of Vincennes, the head is 400 feet according to our hydrography labels. Type 400 in the box.
3. For now, enter 0 for *Resistance*, 500 for *Width* and 10 for *Depth* on the Linesink String properties dialog. We will later revisit these values.
4. Under *Linesink Location* check the radio button "*Along surface water boundary*".
5. Click *OK*.
6. Move the cursor to the point on the stream where the 400 label is located. Click the left mouse button to enter the first vertex.
7. Move the cursor following the river downstream along the western river boundary to the south and click the left mouse button again, creating the next vertex. Continue this procedure until you want to end the string.
8. End the string with a left click, followed by a right click, where the 393.7 ft label occurs, in a meander south west of Vincennes. The dialog box reappears.
9. Type the *Ending Head*, which is 393.7 ft. Click *OK* to complete the definition of this line-sink string.
10. Repeat this procedure, but now position the line-sinks along the eastern river boundary. Use "Wabash River east side" as the label.

At the center of each line-sink in the string GFLOW will assign a [head](#), which results from a linear interpolation of the starting head and ending head along the line-sink string.

Important: You should always **enter line-sink strings from the up gradient end of the stream section to the down gradient end**. This provides for consistency when querying the model results along a string.

Continue to create line-sink strings, using small line-sinks in the [near-field](#) and larger ones in the [far-field](#). None of the remaining streams require a double linesink string as done for Wabash River. Try to keep the total amount of line-sinks below 200 for this initial run.

For linesink strings in the nearfield leave the box *Treat as Farfield* unchecked and select the radio button *Along stream centerline*. Select 0 for the resistance and 5 for the depth parameter. Enter an estimate for the stream width for the *Width* parameter, say between 5 and 50 feet..

For linesink strings in the farfield check the box *Treat as Farfield*, which will set the *Resistance*, *Width* and *Depth* parameters to zero. Consequently, a line-sink string should either be entirely in the far-field or entirely in the near-field. Wabash River, therefore, should be modeled with at least four line-sink strings: a far-field string to the south-west of Vincennes (placed at the center of Wabash River), two near-field strings near Vincennes (which we introduced in the step by step procedure above), and a far-field string to the north of Vincennes, again along the center of the river.

Note: The map you just printed earlier (see Note 2 of [Annotating Base Map](#)) from the project file **example.gfl** has line-sinks on them: the solid black line strings (dark green, blue or pink on a

color print) with solid squares at the line-sink string starting points (vertices) in case the string is selected (clicked on with left mouse button). You may use this line-sink layout as some guidance for your own line-sink creation. The line-sink strings in **example.gfl** are only a first rough setup of the model. You may refine the model at a later time. Notice in **example.gfl** that the line-sink vertices along Wabash River are closer together opposite the well. This has been done to allow the groundwater inflow or outflow along Wabash River to vary due to the effect of the well. Remember that each line-sink has a constant sink density, hence multiple line-sinks are necessary to approximate any infiltration or exfiltration variations along Wabash River opposite the well by a step function of sink densities. When looking on the screen, also observe that the line-sinks near the well field (nearfield) are dark blue, while the line-sinks in the farfield are dark green. Always inspect your model to make sure no dark blue line-sinks occur in the farfield (dark green line-sinks are OK in the nearfield).

Wells

To create a well, point at *New* on the *Element* menu and click on *Well*. Follow these steps:

1. Move the cursor to the location of the well and click the left mouse button. You may place the well to the west of Vincennes near the Wabash River, see the printout of the map of **example.gfl**. A marker for the location of the well is added to the base map and a dialog box appears.
2. Type "wellfield" in the *Label* box. You should be on the tab sheet labeled *General* and the radial button *Discharge-Specified* should be checked. The well should be at UTM coordinates $x=452650$, $y=4280665$. You may adjust the coordinates by typing over them.
3. Type the well pumping rate 350000 in the *Discharge* box and the well radius 4 in the *Radius* box.
4. Since we intend to create [capture zones](#) for this well we should define some particle tracking starting points. Select the tab sheet labeled *Other*.
5. Check the box for *Trace particles from well*.
6. Type the desired number of path lines, e.g. 16, in the *Number of particles* box.
7. Type the aquifer base elevation 330 in the *Starting elevation* box.
8. You may also enter a *Setback Radius*, e.g. 100 feet. A dashed circle will be drawn around the well that may be a sanitary setback. Or, you may enter a "fixed radius wellhead protection zone," for instance 3000 feet.
9. Click *OK* to add the well to the database.

[Main Menu](#) [Previous](#) [Next](#)

Running the Model

Before running the groundwater model you may set some model output options. These include the generation of [potentiometric contours](#), [path lines](#) from wells, path lines from isolated points, and [heads](#) at [test point](#). For our initial model run we will only specify the generation of potentiometric contours.

Select *Model>Settings>Contouring* and check the box *Compute Contours*. Make sure that the option *Heads* is checked (default). You may keep the grid resolution (grid of points where heads will be calculated) at the default *Coarse* for now. Define the *Minimum Contour* level (330), the *Maximum Contour* level (450), and the *Contour Interval* (2) and click *OK* to leave the *Model* menu. Now click on *Model>Solve*, a DOS box appears with some progress indicators for the solution process. When the solver is done the base map reappears with an overlay of [potentiometric contours](#) (blue lines) that are labeled with their [potentiometric head](#) elevations. On top of the map is a text file *Message.log* that contains a runtime log of the Solver. Make sure

all % errors in boundary conditions are at least smaller than 1% and close the file to view the graphical model results.

Note: The heads in the far-field may exceed 450 feet, but will not be plotted above that level since we limited the maximum contour level to 450 (to avoid clutter). You may want to experiment with different contour settings.

[Main Menu](#) [Previous](#) [Next](#)

Check the Solution Integrity

There are 3 different ways in which you can check the accuracy of the groundwater flow solution. These are:

1. Select *Model>View Model Run Files* > *Runtime Message File*, which will open the *message.log* file from the Solver. This is the file that was displayed on top of the graphics at the end of the Solver run and contains a report on the maximum errors detected at boundary conditions. For a description of these errors [click here](#).
2. Select *Model>View Results Table*, click on *View* and on *Line-sinks*. Look in the table to ensure that the *SpecifiedHead* (specified head from water levels) and *CalculatedHead* (head calculated by model) for all [line-sinks](#) are nearly the same. You may also check the *Error* column to identify unacceptable differences. This ensures a solution that meets the boundary conditions, at least at the line-sink centers. Also in the table is a column with the *Discharge* for each line-sink, which is the extraction rate of the line-sink (or sink density) per unit length in square feet per day or square meters per day.
3. There is also a quick graphical way to verify the accuracy with which the boundary conditions at the line-sink centers are met. Select *View>Results Overlay* and check the box in front of *Display Graphical Results Information*. Next select the *Percent Error* from the options menu. Note that the minimum and maximum errors are displayed and used as a scaling parameter. The *Percent Error* will be displayed as a line thickness, whereby the thickness of the line is proportional to the magnitude of the error. Also note the other options on the menu, which you may like to use later during your modeling activities.

The check on boundary conditions (comparing specified and modeled heads) is only relevant after the solver has been run with new aquifer parameters or [analytic element](#) data. You do not have to perform this check, for instance, when regenerating contours or path lines based on the same solution.

[Main Menu](#) [Previous](#) [Next](#)

Displaying Results

Different map overlays may be specified on the *View* menu: e.g. *Elements*, *Contours*, and *Path lines*. If contours are not displayed at this time, check the *Contours* option on the *View* menu and verify that the [contour](#) levels match the specified [heads](#) at streams, at least approximately.

Testpoints

In order to compare the calculated heads with observed heads it is convenient to specify "[Test points](#)" at which you specify observed heads, for instance, heads measured at piezometers or static water levels reported for domestic and industrial wells. Water levels in gravel pits may also be used as test points.

Test points are added as follows:

1. Select *Add Test Point* on the *Model* menu.

2. Move the cursor to the estimated position of the test point on the screen and click the left mouse button.
3. Select the radio button for *Piezometers* (default) and type a label and the observed head (e.g. static water elevation on a well log) in the dialog box. Note: the other two options *Gages* and *Lake Stages* are for more advanced use and should be ignored for now.
4. Click *OK* to add this to the database.
5. Repeat the procedure for other test points.

For the purpose of this tutorial you may make up test points by selecting some arbitrary locations and estimating the heads by use of the potentiometric contours. To simulate reality select some heads a little too high and others a little too low.

Click on the calculator icon on the tool bar or press *F9* again to create a solution with *Test Points*. You may be notified that a valid solution exists and asked if you want to resolve. Click on *No* for using that solution to calculate new heads at the test points.

Note: At test point locations where the modeled head exceeds the observed head a green triangle points upward, its size proportional to the head difference. The head difference may also be posted next to the triangle. Similarly, when the modeled head is lower than the observed head a brown triangle points down. You can modify the scale of the triangle, display numbers only, or display triangles only. To do so point at *Test Points* on the *View* menu and check *As Text* or pointing at *As Symbols* check *+/-5*, *+/-10*, or *+/-20* for making the maximum triangle size correspond to 5, 10, or 20 feet, respectively.

Pathlines

Follow these steps to display pathlines toward the well:

1. Select *Model>Settings>Tracing* and check the *Compute Particle Paths* box.
2. Set the *Maximum Travel Time* to 3650 (10 years).
3. Make sure the *Use Default Step Size of* box is checked.
4. Click *OK*.
5. Click on the calculator icon to rerun the solver. If you are notified that a valid solution exists, go ahead and use it by clicking on *No*.

The base map has now an extra overlay: 16 red path lines that converge on the well. Water from the end points of these path lines will travel on average 10 years (3650 days) to arrive at the well.

To add time markers to the pathlines select *View>Pathlines>Time-of-Travel Tics*, and click on *Every 5 years*.

The map will be redrawn with markers on the path lines from which the water takes 5 years or 10 years to arrive at the well. You may have to zoom in a little to get a good picture. Use the *Zoom to Window* option on the *Window* menu and drag a window box with the path lines at its center.

Note 1: The accuracy of the path lines and time tics depends on the step size (in feet or meters) during the tracing procedure. The default step size depends in turn on the size of the window. If you reduce the window, a new path line trace will generate more accurate results. Reducing the step size below the default will further increase accuracy, but will also slow down the tracing procedure.

Note 2: You may find that some path lines cross the line-sinks along Wabash River. What happens is this. The well forces some of the line-sinks opposite the well to infiltrate water into the aquifer, while elsewhere Wabash River is receiving water. Particle traces end at such

loosing river sections, while elsewhere they cross the river. Additional model refinement may be necessary by adding resistance to the line-sinks, see also [Editing the Model](#) or [Surface water and groundwater Interaction](#)

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 1:** 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). **Note 2:** The default elevation for which the data is being reported is at the saturated aquifer top. You may select another elevation by clicking on the *Instant Inspector Settings* option on the *Tools* menu.

Creating an instant path line.

You may also create an instant path line from any point in the domain without creating a new particle and recalculating all contours and path lines. Just point at the desired starting point, hold down the *Ctrl-Shift* keys and *left-click* the mouse. The path line will have the same time of travel setting as all other traces. The starting elevation and direction of the instant trace can be set under the *Instant Inspector Setting* option on the *Tools* menu. In using this feature you have not created a new particle, thus the trace will not be repeated in subsequent contouring and particle tracing.

[Main Menu](#) [Previous](#) [Next](#)

Editing the Model

You may change any or all input data items. Aquifer properties can be changed under the *Aquifer* tab on the *Model* menu by typing over the data in the boxes. If you open the *Aquifer* tab and decide not to make a change, use the *Cancel* option to leave the *Model* menu, otherwise the program will assume you made a change and will prompt you to create a new solution. You may also change aquifer properties locally by defining inhomogeneity domains with differing hydraulic conductivity, aquifer base elevation, porosity and areal recharge rate. Note: the Inhomogeneity dialog box shows an *Added Recharge Rate* box, which indicates that the recharge rates defined for nested inhomogeneities are cumulative. To add about 1 inch per year of recharge you should enter 0.000228 (ft/day) in that box.

Note: Inhomogeneity domains with a change in the base elevation under unconfined conditions result in a non-linear set of equations. You must select at least 3 to 5 iterations on *Model>Settings>Solver* before clicking on the calculator icon to solve the model.

You may change the location or properties of wells and [line-sinks](#) as follows.

1. Click on the well or a line-sink vertex in the string you want to edit. The well or line-sink string will become highlighted (red).
2. Click on the *Edit* menu and select *Properties*, *Move*, *Delete*, or *Refine*.
3. For line-sinks, when you click on *Delete*, you will be asked whether you want to delete only the selected vertex or the entire string.


4. When selecting *Refine*, a vertex is added midway in the line-sink on either side of the vertex that was selected.

If all you want to do is change *Properties*, you may access the *Properties* dialogue box directly by double clicking on the well or a line-sink string vertex.

Refining line-sink strings

Let's refine the line-sink distribution opposite the well.


1. Click on a vertex of the line-sink string for Wabash River close to the well.
2. Click on *Refine* on the *Edit* menu. Notice the two added vertices (dots) on either side of the vertex you clicked on.
3. Repeat this for one or two other vertices of the Wabash River line-sink string near the well.
4. Rerun the solver by clicking on the calculator icon. You may not see much difference, depending on the quality of your original line-sink representation of Wabash River.

Note: Instead of going to the *Edit* menu, you may also go to the toolbar and click on the *Refine string* icon .

Moving vertices

You may move one or more vertices on line-sink strings to improve the distribution or change the shape of the polyline.

1. Click on the vertex you want to move. It will become red.
2. Click on *Edit>Move* and place the plus sign on the new vertex location.
3. Left click and the line-sink string will be adjusted.

Note: Instead of going to the *Edit* menu, you may also go to the toolbar and click on the *Move Vertex* icon .

Adding resistance to line-sinks

Another way to improve on the realism of representing Wabash River (and other streams in the near-field) is to include an appropriate resistance to flow from the aquifer into the river (or vice versa). To do so follow these steps:

1. Left click on a vertex of the line-sink string that is to be given resistance.
2. If needed uncheck the box *Treat as "far-field"* on the line-sink dialog box.
3. Enter the *Resistance*, which is defined as the thickness of the resistance layer (silt on the river bottom or clay layer underneath the river) divided by the (vertical) conductivity of the resistance layer.
4. We have earlier entered an estimate of the actual width of Wabash River: 500 feet. We entered line-sinks on both sides of the river. Consequently, check under *Linesink Location* the radio button *"Along surface water boundary,"* if not already done. The Solver will automatically calculate an "effective leakage zone" and replace the width parameter by that zone during the solution process. To understand this concept of an "effective leakage zone" read the document "Dealing with resistance to flow into surface waters.pdf" in the document folder or for a summary [click here](#).
5. The *Depth* parameter specifies the approximate distance between the surface water elevation in the stream and the bottom of the resistance layer underneath the stream. This will be an estimated average value for the entire line-sink string.

Note 1: Line-sinks with resistance under unconfined flow conditions result in a non-linear set of equations. You must select at least 3 to 5 iterations on *Model>Settings>Solve*.

Note 2: Line-sinks with resistance will correctly show a different *CalculatedHead* and *SpecifiedHead* in the *Results Table* on the *Model* menu. To verify the accuracy of the solution

for these line-sinks refer to the *Error* column (last column), which reports the difference in the actual line-sink extraction rate and the calculated rate based on applying Darcy's law across the resistance layer.

Note 3: It is **important** to never add *Resistance*, *Width* and *Depth* to line-sinks in the [far-field](#). This will lead to unstable and inaccurate solutions. To avoid this, always check the box *Treat as "far-field"* when a line-sink string occurs in the far-field.

Note 4: When all three parameters *Resistance*, *Width* and *Depth* are specified as zero, the line-sink string must get the *Treat as "far-field"* box checked.

Note 5: When assigning a zero *Resistance*, but a finite *Width* (and optionally *Depth*) the line-sink source density (when it is a losing line-sink) will be limited to the aquifer hydraulic conductivity. This is consistent with the case of incipient ponding in the stream.

Note 6: The *width* parameter in the *Model Results* table or on the *Line-sink Information* panel (which comes up after selecting a line-sink and right-clicking) reflects the "effective leakage zone" calculated by the Solver (if resistance is larger than zero). It differs, therefore, from the value shown on the *Linesink String Properties* dialog that is the actual surface water width.

Conjunctive surface water and groundwater modeling

GFLOW also supports stream flow routing (at least in a simplified fashion). To do so decide which strings of line-sinks you want to assemble in a stream network. One of these strings must be the *End Stream*, which is the most downgradient line-sink string where the network ends. To declare a line-sink string as part of a stream network you must first declare that line-sink string is a near-field feature by unchecking the box *Treat as "far-field"* on the line-sink dialog box (enter the dialog box by right-clicking on a vertex in the string). Make sure you only uncheck that box when the line-sink string actually occurs in the near-field. Next select the *Routing* tab sheet of the *Line-sink String Properties* menu. Check the box for *Use streamflow routing*. Only when this line-sink string is the most down-stream string in the stream network (or the only string) check the box *End Stream*.

For now leave the *Overland Flow* and *End Inflow* values at zero. Repeat this for all strings in the network.

Note: You must have at least one *End Stream*.

The solver will automatically assemble the line-sink strings into a stream network and keep track of base flow in the streams by adding groundwater inflow. In case a stream section tends to lose more water than available as base flow, the infiltration rate is limited to the available amount of water. In case the stream is dry, the line-sinks are removed from the groundwater flow problem.

You can refine this surface water solution by allowing for an average overland inflow. The *Overland Flow* on the *Routing* tab of *Linesink String Properties* is defined as a total volume of water (*cubic feet per day* or *cubic meters per*), which will be evenly distributed over the line-sink string. Note: Overland flow does not have to be Hortonian overland flow, but may represent seepage into the stream from saturated or partially saturated formations above the regional aquifer. You can also provide the stream with a starting stream flow, for instance from an up gradient stream section that is not included in the model. This is done using the *End Inflow* box with units of *cubic feet per day* or *cubic meters per day*. The *End Inflow* is added to the base flow of the up gradient line-sink in the line-sink string.

Keep in mind that the conjunctive surface water groundwater solutions only deal with average stream flow and groundwater flow (steady state model). Also, the river stage remains the same at all times (equal to specified water levels for the line-sink strings), regardless of the stream flow. These conjunctive solutions are most valuable for improving the realism of the groundwater flow solution, although the average base flows in the model can be helpful in

calibration, particularly to estimate the areal recharge rate due to precipitation.

Note 1: Conjunctive surface water and groundwater problems result in a non-linear set of equations. You must select at least 6 to 8 iterations on *Model>Settings>Solver*.

Note 2: Line-sink strings in stream networks should be assigned some appropriate *Resistance*. Without any resistance (resistance equal to zero) these line-sinks may show excessive changes in infiltration or extraction rates between iterations. This may prevent convergence of the solution procedure.

Note 3: Never create stream networks that extend into the far-field! This will lead to unstable and inaccurate solutions.

Note 4: The conjunctive groundwater and surface water solution procedure has only converged when there are no error reports for the surface water solution and sufficiently small errors (usually less than a percent) at boundary conditions for the groundwater solution. These errors are reported in the DOS box during the solver operation and may be reviewed by opening the *Runtime Message File* under the *View Model Run Files* option on the *Model* menu.

Note 5: Under some circumstances it may be necessary to accept small residual negative stream flows when a conjunctive solution fails to converge, even after many iterations. Under these circumstances some line-sinks with resistance are also likely to show relatively large errors.

Note 6: Large errors in line-sinks with resistance may be caused by a few line-sinks with very small extraction or recharge rates. These errors may be ignored.

Note 7: You place a *Test Point* of type *Gage* next to a line-sink in the stream network where the average stream flow or base flow is known, for instance at a USGS stream gage.

Note 7: For large stream networks an under-relaxation parameter may be needed to achieve convergence, see the *Advanced* button on the *Model>Settings>Solver* tab behind the options *Conjunctive Surface Water - Groundwater Solution*.

[Main Menu](#) [Previous](#) [Next](#)

Extracting a MODFLOW model

GFLOW facilitates the extraction of a MODFLOW model covering all or part of the model area in GFLOW. This may be done as follows.

1. Zoom out to the area you want to include in the MODFLOW model and click on *Linesink Vertices* on the *View* menu.
2. Click on *GRID* and click on *Show Grid Dialog*.
3. Click on *File* on the *Finite Difference Grid* dialog box.
4. Click on *New Grid*.
5. Specify the number of rows, columns, grid spacing, etc. as indicated on the dialog box.
Note: the *Lower Left X* and *Lower Left Y* will be automatically defined when you create the grid on the graphics screen, leave the 0.0 unchanged. Also note that the *Row Spacing* and *Column Spacing* are in feet if your *Computational Units* are in feet.
6. The *Perimeter* box is either *Head Specified* or *Discharge Specified*, which refers to the conditions defined on the grid perimeter. These conditions (heads or discharges) will be extracted from the GFLOW solution in a later step. If you select *Discharge Specified* the heads in the MODFLOW model along the grid perimeter must remain close to the current heads in the GFLOW model, or the (modified) MODFLOW model is not compatible anymore with the original GFLOW model. This may be the case when changing pumping rates or recharge rates in the MODFLOW model. In that event a new MODFLOW model must be extracted with the modified data implemented in the GFLOW model.
7. The *Rotation* parameter defines the rotation of the grid counter-clock wise in degrees.
8. Once you click on *Create Grid* the grid perimeter will show up with the lower left corner at the cursor. Move the grid in place and click. The parameter boxes on the *Grid* tab will

now be disabled.

9. To change the defaults on the grid layers, click on the *Layers* tab. Remember that MODFLOW numbers layers from the top down.
10. For further changes to the grid parameters select the *Edit* option on the *Finite Difference Grid* dialog box.
11. To verify the grid data select the *Tracking* tab and move the cursor to the desired grid cells for a review of the data. **Note:** At this point make sure that no line-sinks straddle the grid perimeter. This is why you turned on the *Linesink Vertices* on the *View* menu. If line-sinks do straddle the grid perimeter, move the nearest vertex to the grid boundary or add vertices (using the refine option) and then move the nearest vertex to the grid boundary.
12. Next click on *Processing* and click on *Apply Analytic Elements* to transfer the GFLOW boundary conditions to the MODFLOW grid.
13. Make sure a valid solution exists (Solve at this point if necessary).
14. Select *Extract GFLOW Results*, to transfer the aquifer data and calculated heads and discharges to the MODFLOW grid.
15. Finally click on *Make MODFLOW files* to write a set of MODFLOW input files.

You can import the MODFLOW data files in any MODFLOW preprocessor for further manipulation and for running MODFLOW.

Note: The MODFLOW files do not include the porosity data of GFLOW (MODFLOW does not use these data). However, when conducting particle tracking, for instance with MODPATH, the porosity values are needed. Make sure that the proper porosity values are set in your MODFLOW/MODPATH preprocessor for accurate travel time calculations during particle tracking. This may require you to manually set the porosity in different zones. GFLOW cannot do this for you because, as said, MODFLOW data files do not contain the porosity.

[Main Menu](#) [Previous](#) [Next](#)

Printing Graphics

A hard copy of the graphics box may be obtained by sending it to the printer. Open the *Project* menu and select *Print Setup*. Here you may set margins and select the box "Draw a box at picture perimeter". Click *OK* or *Cancel* and click on *Print* for a hard copy of the graphics box. If you want to use the default print settings, you may also click on the printer icon on the toolbar.

[Main Menu](#) [Previous](#) [Next](#)

Additional Examples

The directory **C:\GFLOW projects \example** contains a few more project files to illustrate the use of GFLOW. A brief description follows:

uniflow.gfl is an example of the use of uniform flow for the Vincennes capture zone delineation modeling represented by the example.gfl project file. You may experiment with changing the properties of the uniform flow by selecting *Uniform Flow* from the *Element* menu and specifying new values (these will replace the existing values).

barrier.gfl is an example of the use of both an open barrier and a closed barrier in the same Vincennes model area as used in the tutorial. The analytic element arrangements in **barrier.gfl** differ from the one in **example.gfl** and the inhomogeneities and barriers are fictitious; they do not represent any real geological features in the area. You may experiment with the properties

on the barriers (double click a vertex to open the barrier properties dialog box) and with the resolution (click on a vertex) and add or delete vertices using the refinement button on the tool bar or the delete key, respectively.

two lakes example.gfl is an example of the use of line-sink based lake features, see the document "Modeling lake-groundwater interactions in GFLOW.pdf" accessible from the *Help* menu.

highlake.gfl is an example of the use of an inhomogeneity domain to model a lake with a priori unknown lake stage, see the document "Modeling lake-groundwater interactions in GFLOW.pdf" accessible from the *Help* menu.

The directory **C:\GFLOW projects \example2** contains a project file with a complex groundwater flow model.

columbus.gfl is an example of a large regional model with several detailed near-field areas. The model is being used for capture zone delineation for the well fields of Columbus, Indiana. The model features an extensive stream network that is used to assess the impact of current and projected groundwater withdrawals on the stream flow in the Columbus area. The **Columbus.gfl** project file is included as a courtesy by WHPA, Inc. and should not be commercially used, referenced or published in any form.

columbus-dry.gfl is similar to ***columbus.gfl***, but several wells near a tributary of the East Fork of White River have been doubled in pumping rate, which causes sections of that tributary to fall dry.

The directory **C:\GFLOW projects \validations** contains some project files used for validating GFLOW.

Uniwell.gfl is a validation problem to test the accuracy of streamline tracing as described in Section 1.1 of the document GFLOW1 validations. The exact residence time for a particle along a horizontal line through the well, parallel to the regional uniform flow, from a point 200 meters up-gradient of the well to a distance of 1 meter from the well axis is 347.95 days.

reisbarrier.gfl is a validation of the horizontal barrier feature in GFLOW. A barrier with resistance should be used only when representing a thin wall. The barrier is compared to three different discrete representations of the wall, using the inhomogeneity feature in GFLOW. The inhomogeneities in the files ***resisbarrierdiscrete.gfl***, ***resisbarrierdiscrete2.gfl*** and ***resisbarrierdiscete3.gfl*** represent walls that are 80 meters long and 10, 5 and 2 meters thick respectively. Each of these walls, however, has been given the same resistance of 100 days by adjusting the hydraulic conductivity of each wall. The thinner the wall the better the barrier feature represents it. When comparing heads and discharges in the domain only the resistance of the wall matters. When comparing the residence time across the wall, the wall thickness of the barrier must correspond to the wall thickness of the discrete inhomogeneity. The total flux through the horizontal barrier in ***resisbarrier.gfl*** is 4.7 cubic meters per day. The total flow through the corresponding inhomogeneity domains are 6.4, 5.6 and 5.1 for the 10, 5 and 2 meter thick walls, respectively. The residence times through the horizontal barrier (along a streamline through its center) is 21.6 % , 11% and 4.9% higher than through the 10, 5, and 2 meter wide inhomogeneities, respectively. As is seen from these data, the horizontal barrier becomes more accurate the thinner the barrier.

circular interface.gfl and ***coastal interface with well.gfl*** are validation problems for interface flow in GFLOW, see the document "Fresh water and salt water interface flow in GFLOW.pdf"

The folder **sectional models** contains several cross-sectional models in GFLOW as described in the document "Modeling flow in the vertical plane in GFLOW.pdf"

[Main Menu](#)

[Previous](#)