Menus

Project Menu

The *Project* menu facilitates the selection and composition of project files and printing. Each project file may have one or more base maps associated with it. The base maps must be georeferenced vector or raster graphics files supported by GFLOW. An existing project file is selected using *Open Database*. A new project file is created using *New DatabaseÖ* . The base maps associated with a project file are added or removed on the *Project Settings...* dialog window.

New Database

Clicking the *New DatabaseÖ* option on the *Project* menu prompts for a project file name. After the project file name is entered you are prompted for your choice of units. First select the units associated with your base map (meters or feet). Next select either "meters and days" or "feet and days" for all input data, the "Units for Computations".

IMPORTANT**:** These unit choices require some explanation. GFLOW maintains the original units associated with the base map files. For instance, when using .bbm files (converted from DLG files) these units are METERS. Check the radial button for meters under "Distance units in Basemap files". It will display these units on the right-hand side of the status bar. Independent from the base map units you are free to use either meters or feet to specify heads. If you select feet, you must also enter pumping rates in cubic feet per day and recharge rates in feet per day, etc. You must tell GFLOW what you will use by specifying the "Units for Computations".

> Incidentally, the GUI converts the map units to the same units as used for computations before submitting data to the GFLOW solver (GFLOW1.EXE), hence the solver works in consistent units. When the solver hands data back to the GUI, the distance units are converted back to the "map units" in use by the GUI.

Next you enter the *Project Settings* dialog window. By clicking on *Add Map* a file dialog window is opened. To view a list of all supported file types, click on "All Supported Types". Click on the file name to be added to the project file and click *OK*. Repeat this procedure for the next base

map, etc.
You may add several maps at once in the usual manner (left-click on first file and shift left-click on last file of the group of files to be added). Next click *OK*.

The GUI has created a default *Base Filename* that is unique for your project. This *Base Filename* will be used to transfer data between the GUI and the Solver. The default filename is rather cryptic and may be replaced by a more meaningful name. However, it must be a unique and valid DOS name of no more than 8 characters. Like MODFLOW, the solver GFLOW1 is a DOS program that cannot read the *Project Filename* (filename.gfl), which may be longer than 8 characters and contain spaces. Note: You can also select the working directory where GFLOW will store all its intermediate files. It is recommended to keep all project files for a particular project in that same directory.
Warning: The *Change Units* button allows you to rename the map units or computational units in case they were incorrectly set. You are only renaming the units, however. No unit conversions take place when changing the units here!
Click *OK* to leave the *Project Settings* dialog window and wait for the maps to be drawn on the screen.

Open Database

Clicking the *Open Database* option on the *Project* menu prompts for a project file name. An existing project file name may be selected from a list displayed in the browser window.

Make Duplicate Database

Clicking *Make Duplicate DatabaseÖ* allows you to duplicate the project file under a different name. This is useful when you want to make changes, but retain the original model.
Note: You may also want to change the name of the *Base Filename* on the *Project Settings* dialog box. Make sure it is unique. If not, GFLOW may refer to the wrong solution!

Close Database

Clicking the *Close Database* option on the *Project* menu clears the current project file. This is done automatically when opening a new project file.

Project Settings

Clicking the *Project SettingsÖ* option on the *Project* menu displays a description of the current project (under *Description*) and associated base maps (under *Currently Used Base Maps*). You may add base maps by clicking on *Add Map..* and clicking on the base map name in the browser window to highlight a base map name. Click on *OK* to add it to the project. You may select more than one file at the time using the *Shift left-click* sequence. You may remove a base map from the project file by clicking on the base map in the *"Currently Used Base Maps*" window and clicking on the *Remove Map..*

The *Change Units* option is a rescue function in case the *Distance units in Basemap Files* or the *Units for Computations* have been entered incorrectly in the *New Database Wizard,* but the units of the maps and data entered are correct. For instance, if the coordinates of the basemaps are UTM, but the *Distance units in Basemap Files* have been inadvertently entered as feet, you can use the *Change Units* option to change that in meters.

Page Setup

Clicking the *Page SetupÖ* option on the *Project* menu will allow you to format the printer page.

Print

Clicking the *Print* option on the *Project* menu will open the printer dialog for your printer. Here you can control printer options (depending on your printer and printer drivers) and print the current graphics screen.

Exit

Clicking the *Exit* option on the *Project* menu will terminate the execution of GFLOW and remove the program from the desktop.

Edit Menu

Select with Window -
Properties-
Move-
Delete-
Refine-
Delete All-
Delete Particle Mode-
Add Text-
Draw Wellhead Protection Area-
Copy to Clipboard-

The *Edit* menu facilitates modifications in model input data.
You can move, delete, or edit the various analytic elements, delete particles (for particle tracing), add text to the graphics screen and copy the graphics screen to the clipboard for including it in a Word document, for instance.

Select with Window

Allows the selection of a group of wells, particles or test points for editing or deletion. After selecting the element type on the Select Area dialog and clciking OK you can drag the cursor to define a window.

Properties

Will display the properties dialog window for the analytic element selected (with a left-click on the cursor), e.g. well, line-sink string, inhomogeneity, horizontal barrier, etc. Alternatively you may *double click* on the vertex to bring up the properties dialog window.

Move

Will allow you to move the vertex selected (with a left-click on the cursor). The mouse pointer will change into a "plus" sign that should be positioned at the new vertex location. Left-click the mouse to move the vertex. Alternatively you may press *CTRL M* to invoke this move feature.

Delete

Will delete the vertex selected (with a left-click on the cursor). Alternatively you may press the "delete" key on the key board.

Refine

Will add a vertex on either side of the vertex selected (with a left-click on the cursor). The new vertices will be placed at the center of the line-elements that are adjacent to the highlighted vertex. Alternatively you may press *CTRL R* to invoke this feature.

Delete all

Select **Hydrography Labels** to remove all hydrography labels from the model.
Select **Test Points** to remove all Test Points (piezometers, gages, and lake stages) from the model.
Select **Particles** to remove all particles used for pathline tracing from the model.
Select **Text Labels** to remove all text labels (permanent and hydrography) from the model.
Select **Discharge Specified Wells** to remove all of them from the model.
Select **Head Specified Wells** to remove all of them from the model.

Delete Particle Mode

Allows the user to delete multiple particles (for pathline traces) by simply clicking on them. Intended for fast delete of particles entered with the *Add Particle Grid* option on the *Model* menu.

Text Label Properties

The *Add Text* option may be used to annotate the base map. Simply click on *Add Text* and move the cursor to the desired starting of the text and click the left mouse button. Type the text in the dialog box and press *Enter*. The text will appear on the base map and is now part of the database.
The *Text Label Properties* dialog box has two radial buttons:

Permanent Labels The labels will be permanently displayed. These labels are used to add names to towns and roads, etc.

Hydrography Labels The labels can be activated or deactivated (hidden) by checking or unchecking *Hydrography Labels* on the *View* menu. This option is used for indicating surface water levels at points along streams. When creating line-sink strings the hydrography labels can be made visible to facilitate the definition of starting and ending heads on line-sink strings. When not in the process of entering line-sink strings the hydrography labels may be turned off to avoid clutter on the graphics screen.

Draw Wellhead Protection Area

The option *Draw Wellhead Protection Area*Ö may be used to draw an envelope around the particle traces from the well or wells. This is also a useful drawing tool for other base map annotations.

Copy to Clipboard

The option *Copy to Clipboard* saves a bitmap of the graphics box on the clipboard for easy inclusion in reports. This option is particularly useful for users of the educational version of GFLOW, who do not have a *Print* option.

View Menu

The *View* menu controls what will be displayed on the graphics screen. The analytic elements, various base maps, Contours, Path lines, Test points, *Results Overlay, Wellhead Protection Areas*, Hydrography *Labels*, etc. can all be shown together or combined in any way desired. Each of these features is seen as a map layer. The various options will only produce results if the underlying data layers have been created. For instance, for contours to be displayed, a solution with contours must have been generated. To do so, check the *Show Contours* box *Model>Settings>Contouring*, etc.

Elements

Check to show a layout of the analytic elements (line-sinks, line doublets, wells, etc.). This option is almost always checked.

Contours

Check to show contour lines, which may be potentiometric contours or any other contour type as selected on the *Model>Settings>Contour* tab.

Pathlines

When pointing to *Path Lines* a menu appears from which you may select:

Show Paths When checked the pathlines will be shown on the screen.

Time of Travel Tics When pointing at *Time of Travel Tics* a menu appears from which you may select:

- **Show Tic Marks** If checked, small circles are placed on the pathlines at predefined travel time intervals. These travel time intervals are selected on the same *Time of Travel Tics* menu, see below.
- **Every 1 year** Sets the travel time interval to 365 days.
- **Every 2 years** Sets the travel time interval to 730 days.
- **Every 5 years** Sets the travel time interval to 1825 days.
- **Custom Ö** Enter the travel time interval for time tics in days.

Pathline Elevation Tics

When pointing at *Pathline Elevation Tics* you are prompted *to*

- **Show Tic Marks** If checked, small **green** circles will be placed on pathlines to for every rise of a predefined interval and blue circles will be placed on the pathlines for every drop of a predefined interval. The rise or drop intervals are set using the *Enter Spacing* option, see below.

Enter Spacing. If, for example you enter 5, circles are plotted on top of the path line at elevations that are multiples of five, e.g. 535, 540, 545, etc. If the path line goes downward in the aquifer the circles are **green**, if the path line goes upward the circles are blue. To identify the actual elevations you must count the circles on the screen starting at the known starting elevation of the path line as specified for the particle. Double (left-)click the particle to display its properties or double click the well and select the *Other* tab to find the starting elevation of particle traced backward from the well.

Note: *Three-dimensional pathlines* may be displayed outside the GFLOW GUI, for instance by exporting the pathlines as a BLN file using the *Export* option on the *Tools* menu. These BLN files may be imported in SURFER.

Test Points

When pointing to Test Points a menu appears from which you can select:

Show Test Points If checked will display at piezometer locations the difference between measured heads and modeled heads on the screen as scaled symbols. Similarly the difference between modeled streamflow and measured streamflow will be displayed at gage locations. The symbol is a triangle centered at the test point location that points upward if the model head is larger than the measured head and downward if the reverse is true. The triangle is scaled as indicated below.

Numerical Values If checked the difference ?h between the modeled heads and the measured heads will be printed next to each piezometer symbol: ?h=model head - measured head. The difference ?Q between modeled stream flow and measured stream flow will be printed as a percentage of the measured streamflow: ?Q=((model Q - measured Q)/measured Q)*100%

Scaled Symbols (Piezometers) When pointed at the following option appear:

+/- 5 If checked the maximum size up and down arrow will indicate a ?h of 5 feet/meters or larger.

+/- 10 If checked the maximum size up and down arrow will indicate a ?h of 10 feet/meters or larger.

+/- 20 If checked the maximum size up and down arrow will indicate a ?h of 20 feet/meters or larger.

Scaled Symbols (Gages) When pointed at the following option appear:

+/- 10% If checked the maximum size up and down arrow will indicate a ?Q of 10% of the measured stream flow.

+/- 50% If checked the maximum size up and down arrow will indicate a ?Q of 50% of the measured stream flow.

+/- 100% If checked the maximum size up and down arrow will indicate a ?Q of 100% of the measured stream flow.

Scaled Symbols (Lake Stages) When pointed at the following option appear:

+/- 5 If checked the maximum size up and down arrow will indicate a ?h of 5 feet/meters or larger.

+/- 10 If checked the maximum size up and down arrow will indicate a ?h of 10 feet/meters or larger.

+/- 20 If checked the maximum size up and down arrow will indicate a ?h of 20 feet/meters or larger.

Results Overlay

When checking the *Results Overlay* a dialog box appears with *Results Overlay Selections*:

By checking the box *Display Graphical Results Information* the layout of line-sinks will exhibit line thicknesses proportional to what is checked under *What to Display*:

Percent Error Is the difference between calculated and specified head at the center of line-sinks with a zero resistance, or is the difference between the line-sink discharge (sink density) calculated from Darcy's law applied to the resistance layer and the actual discharge calculated by the model. For the case of a head difference the percent error is relative to the saturated thickness underneath the line-sink. For the case of the resistance line-sinks the percent error is relative to the average discharge for all line-sinks.
Strength Is the discharge or sink-density of the line-sink in square feet or square meters per day. It will be displayed as a blue line when positive and a pink line when negative.
Baseflow Only functional when a conjunctive surface water and groundwater flow solution is available. When the stream section has no flow it will be displayed by a gray line.
Streamflow Only functional when a conjunctive surface water and groundwater flow solution is available. When the stream section has no flow it will be displayed by a gray line.
Highlight Recharging Colors all recharging line-sinks (negative discharge) light green.
Highlight Percolating Colors all line-sinks that "percolate" (head in aquifer below the resistance layer) as light green.

Note: In order to force the default scaling values, based on the minimum and maximum values in the model, you may have to select another option and than select the desired option again.

Wellhead Protection Areas

Check to show the wellhead protection boundaries drawn with the wellhead protection drawing tool.

View Hydrography Labels

Check to show hydrography labels on the basemap. Hydrography labels are used to determine beginning and end heads for line-sink strings. After having created the necessary line-sink strings the *Hydrography Labels* may be turned off to avoid clutter on the screen. See also *Add TextÖ* on the *Edit* menu.

Toolbar

Check to display a toolbar with shortcuts to menu options.

Refresh

The *Refresh* option redraws the map, which is used in conjunction with the *Manual Redraw* option discussed below.

Show Vertices

Produces a submenu with the following choices:

Line-sink Vertices, when checked, makes the vertices on all line-sink strings visible. This is important when entering inhomogeneity domains to ensure that a line-sink is either entirely inside or outside a domain.

Inhomogeneity Vertices. When checked makes all inhomogeneity vertices visible.

Domain and Boundary Vertices, when checked makes vertices on HUC domain boundaries and HUC farfield boundaries visible. See also "Managing Hydrologic Units in GFLOW.pdf" in the documents folder.

Show Stream Linkages

The *Show Stream Linkages* is available only when a solution is present (else it is grayed out). The downgradient point of the last line-sink in a string is connected by a red line to the upgradient end of the line-sink to which it is connected (to which it gives its streamflow). When zoomed out correct linkages will show up as red dots. The end of an "*Endstream*" linesink string is marked with a black dot. **Note:** Lake features (*Lake* selected on the *General tab* of the *Linesink Properties* menu) will have a black dot somewhere on the lake boundary, since they are defined as *Endstreams* even though they may be inside a network (have one or more inlet streams and outlet streams. Line-sink strings connected to line-sink lakes that form inlet streams should show a connection (red dot). If not they may have been checked as *Inlet stream*, which is reserved for high-k lakes (see comment on *Routing tab* of *Linesink Properties* menu). Line-sink strings connected to lakes that form outlet streams, should be checked *Outlet stream* on the *Routing tab*, but do not show a connection (no red dot).

Hydrologic Unit

When pointing at *Hydrologic Unit* a menu appears with the following options:

Hydrologic Unit Boundary Check to display the boundaries of the hydrological unit domains.

Farfield Boundary Check to display the farfield boundaries associated with the hydrological unit domains.

Model Boundary Check to display the model boundaries associated with the hydrological unit domains.

See also "Managing Hydrologic Units in GFLOW.pdf" in the documents folder.

Manual Redraw

The *Manual Redraw* option, when checked, prevents the maps from automatically being redrawn after data manipulation or solutions. This is useful when working with many maps on a slow computer. The user can redraw the maps at will by clicking on *Refresh* (*View* menu or *Window* menu).

Base Map

When pointing at *Base Map* a menu appears with the following options:

Show Editor Grid When checked grid lines are drawn with a spacing defined by *Editor Grid Settings*, see below.

Snap to Grid When checked the locations of vertices of line-sinks or other analytic elements are rounded to the nearest grid point.

Editor Grid Settings Opens the Change background grid spacing dialog box. Enter the desired grid spacing in current map units.

Vector Graphics Opens a dialog to select and unselect vector graphics files for display. When unselected, the files will remain associated with the project. They may be permanently deleted under *Project>Project Settings*.

Click the *Shapefile Settings* button to edit Shapefile attributes, click here for details.
Raster Graphics Opens a dialog to select and unselect raster graphics files for display. When unselected, the files will remain associated with the project. They may be permanently deleted under *Project>Project Settings*.
Select BBM group brings up a box around each group of BBM maps and enters a BBM map editing mode. Click inside a box to select the associated BBM maps. A dialog comes up that shows the file names of the BBM maps. You can select one or more of these files and *Hide Maps*, *Show Maps* or *Remove Maps* them (from the model). Click *Apply* and then *OK* to go to another box or click *Done Editing BBM Groups* to exit the BBM editing mode.

Shapefile Properties

Shapefiles do not have standardized attributes, like CAD files have. The GFLOW GUI will scan for color attributes and try to display these colors, but colors may or may not be in the file, or not in a form recognized by the GUI.
The *Shapefile Properties* dialog provides an opportunity to display attributes of a selected Shapefile as text labels and to edit attribute colors or assign a single color for the entire map, in case color information is missing. There are two tables of attributes displayed on the dialog: one with all attributes that could potentially be displayed as text labels, and one with all attributes that could potentially be refer to color information.

Displaying Text Labels.

Simply highlight the fields in the table that you like to be displayed as text labels on the graphics screen.

Displaying the map in color.

You may overrule color selections for the color types listed in the table. Simply replace the color code (integer) by a different code.
If the table does not show color information the entire map will be displayed in a default color: black. To overrule this default, select the radio button "*Pick a fixed color*" and click on *Choose Color*. The entire map will be drawn in this newly selected color.

These attribute modifications will be saved in the database when clicking *Apply Map Changes*. Repeat this procedure for all Shapefiles in the project database.

Haze Background

Checking this option will place a haze over the background map. This feature is useful when the background map is too busy to clearly see potentiometric contours and other modeling results. You can select hazing properties (intensity and color) under *Tools>GUI Options*.

Element Menu

To create a new analytic element click on the *Element* menu and point at *New*. The following options appear.

New

- Well
- Linesink
- Horizontal Barrier
- Inhomogeneity

The links lead to the Help screens for the Properties dialog windows. Below follows a general description of the use of these analytic elements.

Uniform Flow

A *Uniform FlowÖ* option on the *Element* menu allows for conceptual modeling of a well in an ambient (uniform) flow regime. Such a solution may provide a first approximation to a capture zone for that same well in a flow regime generated by recharge and controlled by surface waters which serve as boundaries to the flow field. The *Uniform FlowÖ* option is **not** an analytic element. As a rule it should not be used when solving regional flow subject to hydrologic boundary conditions. This issue is presented in a text box that appears after clicking on *Uniform Flow*. To enter *Uniform Flow,* click on *Yes* and place the cursor (+ sign) on a location where you want to enter a reference head (location of known head). Enter the reference head (head at the location you placed the + sign), regional hydraulic gradient and its orientation. The orientation of the uniform flow is in degrees measured counter clockwise from the positive x-axis. For instance, if the flow is due North enter 90, while if the flow is due South enter -90. Flow that is due East may be entered as 180 or -180.

Wells

To create a new well click on *Element*, point at *New*, and click on *Well.* Move the cursor to the point on the map where the well is to be placed and click the left mouse button. The well is drawn on the map and a dialog box appears.

There are four tab sheets. Most of the time you will need to select the *General* tab sheet. There you can select whether the well is *Head-Specified* or *Discharge-Specified*. The coordinates are from the map, but if you know exact coordinates you may overwrite them.

If you intend to use the MODFLOW extract feature and want the well in a particular layer, or if you want to specify a partially penetrating well (three-dimensional solution) in GFLOW, you should refer to the *3-D Geometry* tab sheet. As a rule, you do not need to *Use 3-D Partially Penetrating Function*. It can only be used in GFLOW in areas of confined flow and is computationally intensive. However, if you do specify the *Top of Screen* and *Bottom of Screen* the MODFLOW extract feature will place the well automatically in the proper layer, even though GFLOW treats the well as fully penetrating.

The *Transient Well* tab sheet is particularly useful when simulating pumping tests. However, when the transient effects of the well (e.g. drawdowns) reach a stream or other type of boundary condition the conditions at that boundary section will be violated; GFLOW is not a fully transient model.

The *Other* tab sheet allows for the drawing of a circle around the well, e.g. a sanitary setback radius or a fixed radius capture zone (wellhead protection zone). You may also specify how many particles should be traced (backward in time) from the well. The starting elevation should be below the aquifer top or water table to avoid a trace to prematurely end because the streamline entered the aquifer too close to the well. Remember that, although GFLOW is a Dupuit-Forchheimer model (2D equations) it does trace streamlines in three-dimensions.

For further details on the well dialog see Well

Line-sinks

To create a new line-sink string click on *Element,* point at *New,* and click on *Line-sink*. A dialog box will appear to let you specify a *label* for the line-sinks (line-sink string). Below it are three tab sheets.

The *General* tab sheet

This is where you select the type of line-sink string you want to create and provide its parameters.

Head-Specified line-sinks

Usually you will select *Head-Specified* to represent streams or lakes. You must give a *Starting Head*, which is the water level in the stream at the start of the string you are about to create. The *Ending-Head*, of course, is the water level at the end of the string. You may enter it now or after having completed the string, when the dialog box reappears. After clicking on *OK* move the cursor to the location for the first vertex of the line-sink string and click the left mouse button. Continue to move the cursor to subsequent vertices and click the left mouse button to enter these vertices. Click on the right mouse button after having entered the last vertex of the line-sink string. At this point the dialog box reappears and now you must specify the head (surface water level) at the end of the string, if you have not already done so. The heads for each of the line-sinks entered will be added to those line-sinks by linear interpolation between the starting and ending heads you defined in the dialog box. Note: if you marked up the streams with water levels (Hydrography labels) you may activate these labels on the *View* menu prior to creating line-sinks.

Line-sinks can be a near field or a far field feature. If a line-sink is entered in the far field you must check the box *Treat as "far-field",* which will prevent you from specifying a *resistance*, *width*, and *depth* parameter. Only if a string occurs in the near field should you uncheck the box *Treat as "far-field"* to activate these attributes. Once you unchecked the *Treat as "far-field"* box you may specify a *Resistance*, *Width* and *Depth* for the line-sink.

Important: Line-sinks in the far field should have zero resistance, zero width and zero depth. In other words, they must have the box *Treat as "far-field"* checked.

Note: Line-sink strings that are used for modeling surface waters with a bottom resistance must be given an appropriate width parameter to accurately model the rate of groundwater inflow or surface water outflow. You may select the line-sink location as "*Along surface water boundary*" or "*Along stream center line*" to have the Solver calculate this effective leakage zone automatically. For these cases the width parameter is the actual surface water width at the location of the line-sink string.

Note: When the *Resistance, Width and Depth* parameters are all set to zero, the line-sink string must have the box *Treat as "far-field"* checked. You may opt to give a line-sink string in the near-field at least a width parameter other than zero, in which case the maximum infiltration rate of the line-sink (when it represent a loosing stream) will be limited to the aquifer conductivity.

Discharge-Specified line-sinks

When you select *Discharge Specified* you will be prompted for the *Sink Density* (line-sink strength) at the start of the string and at the end of the string. The Sink Densities for each of the line-sinks will be determined by linearly interpolating between these values. Note: Each line-sink has one constant sink density, which is the total inflow into the line-sink (cubic feet per day) divided by the line-sink length. Hence, the Sink Density is defined in *square feet per day* or *square meters per day*. When you check *Discharge Specified* all other options are turned off, e.g. you cannot use these line-sinks as part of stream networks.

Drain

When Drain is checked the head specified line-sinks will be prevented from recharging the aquifer. This option is similar to the drain cells in MODFLOW.

Gallery
When the Gallery option is checked the line-sink string becomes a horizontal well (gallery) with a specified total discharge (pumping rate). The head inside the gallery will be constant along the gallery and depend on the pumping rate and hydrogeology of the aquifer.

Lake
When the Lake option is checked the line-sink string is assumed to define the perimeter of a lake. The user must supply lake level estimates and the solver will iterate toward a lake level that provides water balance for the lake. For this to occur you must check the box *Apply outside loop iterations* and specify a number of iterations (>3).

For further details, see *General Tab* under Linesink

The *Routing* tab sheet

The *Routing* tab sheet is used to make the string you entered part of a stream network in GFLOW. The *Routing* tab sheet is only active when the *Treat as Far-Field* box is checked off, in other words the line-sink strings must be in the near-field. In such a network base flow accumulation (routing) will be implemented and line-sinks will not be allowed to lose (infiltrate) more water then available as stream flow (base flow plus a specified *Overland Flow*. The *End Stream* box must be checked if the string is the last section of the stream network, hence if it does not connect to any other string. Each stream network must contain one *End Stream*! For a further explanation of the options see *Routing Tab* under Linesink

Note: To implement the stream flow routing you must check the box *Conjunctive Surface Water - Groundwater Solution* on the *Solver* tab of the *Model* menu. GFLOW will automatically make the stream network connections for the strings. You need more iterations for conjunctive solutions than for groundwater flow solutions alone (depending on the complexity of the problem 6 iterations or more). When you uncheck the box *Conjunctive Surface Water - Groundwater Solution* the routing settings in the GUI are preserved, but only a groundwater flow solution will be generated (no stream flow solutions).

Important: Line-sinks in a stream network (when *Use streamflow routing* is specified) should have an appropriate *Resistance*, *Width* and *Depth*. If these parameters are zero the solution may not converge. Line-sink strings that are part of a stream network should only occur in the near field. In other words do not declare a line-sink string a near-field feature (*Treat as Far-Field* turned off) when it actually occurs in the far field.
More details about the theory and implementation of stream flow in GFLOW may be found in "Modeling Steady State Conjunctive Groundwater and Surface Water Flow with Analytic Elements", Sherry Mittchell-Bruker and Hendrik M. Haitjema, Water Resources Research, Vol. 32, No. 9, pages 2725 - 2732.

For further details, see *Routing tab* under Linesink

The *Scenarios* tab sheet

The *Scenarios* tab sheet facilitates the assignment of a symbolic name to the *Resistance* of the line-sink string. This symbolic name is included in a template file that can be exported for use with the automatic calibration tool PEST, see *PEST* on the *Tools* menu.
Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _
Warning: The *width* parameter on the *Line-sink Properties* dialog, when properly entered, depends on the "characteristic leakage length." For more details click here. When PEST replaces the scenario name with resistance values, this *width* parameter will not be updated. To overcome this problem, the *width* parameter in the *.tpl file will be set to *w=1.0* whenever a scenario name is specified. The PEST user must specify the range for the resistance values to be used by PEST as the range for *c*=c/w*, where *c** is an effective resistance, where *c* is the actual resistance, and where *w* is determined in accordance to the "characteristic leakage length," which depends on both the actual bottom resistance and the aquifer transmissivity in use.

Horizontal Barriers

To create horizontal barriers click on *Element* , point at *New*, and click on *Horizontal Barrier*. You will be prompted for the name of the barrier. If you want to model a closed slurry wall or a rock outcrop, or a closed aquifer basin, check the box for *Barrier bounds a closed domain*. You have the option to prevent the plotting of contours in the area inside a rock outcrop, to do so check *Inside Boundary* for the *Ignore Region* option. If you want to prevent contours outside a closed aquifer basin check the box before *Outside Boundary*. This will lead to cleaner graphics and will also instruct MODFLOW to create inactive cells in case you use the MODFLOW extract option (see *Grid* menu). Next enter the hydraulic conductivity of the barrier material, its thickness and the depth of the barrier. GFLOW will calculate a conductance for the barrier that is based on the barrier properties and its degree of penetration. Groundwater flow will occur through the barrier as well as underneath it, unless the barrier extends down to or below the aquifer base.
After clicking on *OK* move the cursor to the location for the first vertex of the barrier and click the left mouse button. Continue to move the cursor to subsequent vertices and click the left mouse button to enter these vertices. Click on the right mouse button after having entered the last vertex of the barrier.
Note 1: A horizontal barrier that is a closed no-flow boundary divides the aquifer into two independent flow domains. Consequently, at least one head specified boundary condition must be specified on the inside of a closed no-flow boundary!
Note 2: A horizontal barrier creates a jump in the potentiometric head across that boundary. Consequently, potentiometric contour lines are bundled up and often ragged next to these barriers. This is an artifact of the contouring routine trying to make a smooth potentiometric head surface out of the jump. It does not in any way influence the validity of the groundwater flow solution.
Note 3: Horizontal barriers may require a rather high resolution (many vertices). This is particularly true for barriers with a rather low resistance, such as partially penetrating barriers. Also, open barriers should have rather small line segments near their ends (increased vertex density near the ends).
Note 4: When entering a new horizontal barrier, the bottom of the barrier on the dialog box

defaults to the aquifer base elevation (fully penetrating) as specified on the *Aquifer* tab of *Model>Settings*. If the barrier occurs inside an inhomogeneity area with a different aquifer base elevation you should not forget to adjust the bottom elevation of the barrier to keep it fully penetrating.

For further details see Horizontal Barrier

Inhomogeneity

To create areas with different hydraulic conductivity, aquifer base elevation, porosity and recharge due to precipitation, inhomogeneity areas may be used. Before introducing an inhomogeneity area it is advised to check the *Line-sink Vertices* option on the *View* menu. This will make all line-sink vertices visible on the graphics screen, which is important to ensure that a line-sink is either entirely inside or outside an inhomogeneity. To enter an inhomogeneity domain open the inhomogeneity dialog box by clicking on *Inhomogeneity* on the *Elements* menu.

The *General* tab sheet

When only changing the *Recharge*, leave all other check boxes unchecked and enter the recharge rate. The recharge rate will be added to what may have been specified in the area with another inhomogeneity. Areas that only define a change in recharge may be placed anywhere and may overlap any other analytic element or inhomogeneity domain. This is not true for domains that redefine other parameters, see below.

To change the hydraulic conductivity check the box *Change hydraulic conductivity from default* and enter the proper value in the box *Conductivity*. This value will replace whatever hydraulic conductivity was defined earlier on the *Aquifer* tab sheet or by use of another inhomogeneity domain. Domains with a different *Conductivity* may not overlap each other, but they may be nested or abutted, and should intersect line-sink strings at a vertex (select *Line-sink Vertices* option on *View* menu to display these vertices). Avoid sharp corners in inhomogeneity domains with differing hydraulic conductivity. When the hydraulic conductivity inside an inhomogeneity is more than one or two orders lower than the surrounding value a rather large number of vertices may be needed along the inhomogeneity boundary.

To change the porosity check the box *Change porosity from default* and enter the new porosity. This value will replace whatever porosity was defined earlier on the *Aquifer tab* sheet or by use of another inhomogeneity. Domains with differing porosity may not overlap each other (they may be nested or abutted). Domains with only the porosity defined (all other boxes left unchecked) can have any shape and any element sizes (can have few vertices). They do not add equations to the solution procedure.

To change the base elevation check the *Change base elevation from default* box. Enter the new base elevation in the *Base Elevation* box.
Under **unconfined flow conditions**, the Solver needs the head distribution along the inhomogeneity boundary to arrive at a correct solution. This head distribution is, of course, not a priori known. The Solver will use an iterative solution process starting at iteration 1 by using the average specified head in the model domain as the head along the inhomogeneity boundary. During successive iterations the newly calculated heads along the boundary will be used. You

can optionally provide an estimate of the *Average Head* along the boundary of the domain to improve on this iterative solution procedure. This is useful when you encounter solution stability problems. Check the box *Provide average head for the domain* and enter an estimate of the average head in the *Average Head* box. **Warning**: The default value of the *Average Head* is arbitrarily set equal to the aquifer top, which may be a very poor estimate of the average head along the boundary of the domain. Make sure to provide a more reasonable estimate. Under **confined flow conditions** the head along the inhomogeneity boundary is not being used, hence no Average Head needs to be provided.

You may provide the inhomogeneity domain boundary with a unique color using the *Color* selection option on the Inhomogeneity dialog box.
Click *OK* to start entering vertices for the inhomogeneity domain. When you entered the last vertex right-click the mouse and the domain will automatically be closed. Remember when entering domains where the box *Change hydraulic conductivity from default or Change base elevation from default* is checked that you should intersect line-sink strings only at vertices, and should avoid making sharp corners.
Note 1: Make sure the new base elevation does not exceed the anticipated groundwater table or the aquifer top!
Note 2: Make sure that other elements are adjusted to the new base elevation. For instance, the elevation of the top and bottom of a well screen and the bottom elevation of a horizontal barrier should be set at or above the new aquifer base if they occur inside the inhomogeneity.
Note 3: The default value of the *Average Head* is arbitrarily set equal to the aquifer top, which may be a very poor estimate of the average head along the boundary of the domain. Make sure to provide a more reasonable estimate.
Note 4: Providing an average head estimate will add three extra iterations to the number specified on the *Model>Settings>Solver* tab.

For further details see *General tab* under Inhomogeneity

The *Scenarios* tab sheet

The *Scenarios* tab sheet facilitates the assignment of a symbolic name to the *Conductivity* of the domain or *Added Recharge* of the domain. This symbolic name is included in a template file that can be exported for use with automatic calibration tools such as PEST, see *PEST* on the *Tools* menu.
Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Well Properties Dialog

The ***Label*** is a user definable label for the well, which must be specified.

The *General* tab sheet:

Head-Specified should be selected if the head at the well screen (water level in the well) is known and the discharge is to be calculated by the model. This type of well will add 1 equation to the solution procedure.

Discharge Specified should be selected if the pumping rate of the well is known. This type of well will not add an equation to the solution procedure.

X coordinate and ***Y coordinate*** define the location of the well in map units and are obtained from the cursor position on the map and may be altered.

Discharge is the average rate at which the well pumps. It must be entered in cubic feet per day or cubic meters per day, see units displayed! Some useful conversion:

Multiply	by	to obtain
GPM	192.528	cubic feet per day
GPM	5.452393	cubic meters per day
GPD	0.1337	cubic feet per day
GPD	0.0037864	cubic meters per day

Radius of the well must be set sufficiently large to ensure that the head remains above the aquifer base. Most high capacity wells have a radius of 0.5 feet or more (0.2 meters or more).

The *3-D Geometry* tab sheet

Check the ***Use 3-D Partially Penetrating Function*** box to use a complete three-dimensional function for the partially penetrating well. This type of well adds 2 or 3 equations to the solution procedure, depending on the well being discharge or head specified, respectively.
Note: For most applications this feature is *unnecessary*. Three-dimensional effects occur only in the immediate vicinity of the well (about within two aquifer thicknesses). For discharge specified wells the flow regime outside that small domain is the same for a fully or a partially penetrating well.
Warning: This 3-D function (partially penetrating well) can only be used in a *confined* flow zone. This means that inside a radius of about two times the aquifer thickness the head must be above the aquifer top. When applying the 3-D functions for a discharge specified well in an unconfined aquifer zone an incorrect solution will occur near the well (not remote from the well).

The ***Top of Screen*** and ***Bottom of Screen*** must occur within the confined aquifer near the well.
Important: This option must be set regardless whether the *Use 3-D Partially Penetrating Function* option has been checked if a MODFLOW model will be extracted. The well screen settings will place the well in the proper MODFLOW grid layers.

The *Transient Well* tab sheet

Use Theis transient well function box may be checked to create a discharge specified well that starts pumping at some specified time (larger than zero).

Cycling pumping box may be checked to create a well that alternatingly pumps and is off for a specified time interval. Cyclic pumping always starts at time zero. For further details on how cyclic pumping is implemented refer to Haitjema (1995) section 3.6.3

Storage Coefficient is the dimensionless storage parameter *S* obtained by multiplying the "specific storage coefficient" S_s by the saturated aquifer thickness, see Haitjema (1995) equation 3.329 on page 152. In an unconfined sandy aquifer, for instance, the *Storage Coefficient* is a little less than the effective porosity. In a confined aquifer the *Storage Coefficient* is the elastic specific storage S_s (Haitjema, 1995 equation 3.331 on page 152) times the saturated aquifer thickness.

Starting head is the (approximate) head at the well prior to pumping. It is used to define the saturated aquifer thickness used in the storage term of the Theis equation.

Starting time is the time, larger than or equal to zero, when the well starts pumping. This option is replaced by *On/Off time period* when the *Cyclic pumping* option has been checked.

On/Off time period is the time that the well pumps and the time that the well is off.

Number of cycles is the number of times that the well is turned on and off. For instance, with *On/Off time* = 1 (days) and *Number of cycles* = 3 the well will be on from time 0 to 1 (during first day), off from 1 to 2, on from 2 to 3, off from 3 to 4, on from 4 to 5 and off after 5 days.

Note: To implement the transient well you must first make a new solution (click the calculator icon), which will occur for a default time of zero (no transient effects). Next set the time using *Model>Time*. After that the program will regenerate the contour plot for the specified time. The transient well can only be used for a limited pumping time (see warning below).

Warning: Transient wells (Theis function) are isolated features that are added to the steady state groundwater flow solution. The effects of the transient well are not incorporated in the sink densities or strengths of line-sinks or inhomogeneities, respectively. Consequently, adding transient wells is only valid so long as their impact (drawdowns) does not extend to other analytic elements. The effect of a nearby stream on the transient well solution may be approximated by use of an image well, see Haitjema (1995) page 159 and figures 3.66 and 3.67.

The *Other* tab sheet:

Setback radius defines a circle around the well that may serve as a "sanitary setback" or which may reflect a "calculated fixed radius capture zone". The latter calculated fixed radius *R* may be obtained from:

$$R = \sqrt{\frac{Qt}{N\pi t + n\pi H}}$$

where *Q* is the pumping rate of the well, *t* the desired travel time for the fixed radius capture zone, *N* the areal recharge rate, *n* the effective aquifer porosity, and *H* the saturated aquifer thickness. The parameters should, of course, be entered in consistent units. Make sure *R* is in meters or feet as indicated on the dialogue box. A conservative (large) value for R is obtained by ignoring recharge; N=0.

Trace particles from well may be checked to create time of travel capture zones for the well. Particles will be traced back in time starting at the well and stopping at a travel time defined as *Maximum Travel Time* on the *Model>Settings>Tracing* tab sheet.

Number of particles define the number of path line traces that will occur. Fifteen path lines are suitable for single wells. For more complicated situations, like several nearby wells, a higher path line resolution may be needed, e.g. 20 to 25 path lines. The particles will be distributed evenly around the well.

Starting elevation is the elevation at which the particles will be distributed around the well. GFLOW performs 3D path line tracing, hence the particle starting elevation may influence the extent of the path line trace. For instance, if the path line starts close to the saturated aquifer top, the path line may stop at the aquifer top before the *Maximum Travel Time* has been obtained. Conversely, if the *Starting elevation* is set at the aquifer bottom, the path line will remain at the aquifer bottom, unless it enters a stream (line-sink).

Linesink String Dialog

Label is a user definable label for the line-sink string, which must be specified.

General tab
Routing tab
Scenarios tab

The *General* tab sheet

Treat as "far-field" is checked for line-sink strings that represent far field features. When checked no *Resistance*, *Width* and *Depth* parameters can be set and the *Routing* tab sheet is not accessible.
Important: All line-sink strings in the far field must have the box *Treat as "far-field"* checked and thus should not have a *resistance*, *width* or *depth* defined nor should they be used as part of a stream network.

The 5 radial buttons on the left define 5 different applications for line-sink strings. Depending on the selection, the data fields on the general tab will change. Consequently, these data fields will be discussed as a subtopic for each selection.

The 3 radio buttons on the right define the location of the line-sinks when used to model flow into or out of surface waters and control the treatment of the width parameter as explained below.

Head-Specified should be selected if the line-sink string is used to represent a stream section or lake boundary with a given water table.

Starting Head Water level in the stream at the starting point of the line-sink string.
Ending Head Water level in the stream at the end point of the line-sink string.
Resistance is the thickness of the resistance layer between the surface water feature and the aquifer divided by the average vertical hydraulic conductivity of the resistance layer. See Haitjema (1995) section 5.2.3 pages 234 - 239.
Width is a parameter related to an "effective leakage zone" underneath the surface water body in case it has a bottom resistance (*Resistance* parameter larger than zero). This effective leakage zone depends on the bottom resistance and the aquifer transmissivity and may be calculated as indicated in the document "*Dealing with resistance to flow into surface waters.pdf*" For a summary of calculating the width parameter click here. You must do this calculation by hand if you check the radio button "*Unkown*" for the line-sink location. If you check the radio button "*Along surface water boundary*" or "*Along stream center line*" the Solver will automatically do the calculations. For those cases the *width* parameter on the *Linesink String Properties* dialog must be set to the actual stream or lake width. For PEST applications the option "*Unkown*" should not be used, since the Solver must be allowed to recalculate the effective leakage zone based on the changing resistance and transmissivity values. The option "*Unkown*" is included for compatibility with older project files where the *width* parameter has already been set equal to the effective leakage zone or when the resistance parameter is not used for surface water bottom resistances, but, for instance, to represent resistance to three-dimensional flow, see "*Accounting for resistance to 3D flow.pdf*"

Depth is the distance between the surface water elevation and the bottom of the resistance layer (like for river cells in MODFLOW). The *Depth* parameter is used to determine the aquifer thickness underneath the surface water (used in calculating the effective leakage zone, see *width* above). The depth parameter is also used to determine when a line-sink represents a percolating surface water (unsaturated zone between the stream bottom and the groundwater table).

Note: It is recommended to enter line-sink strings along streams going down-stream (into the direction of the stream flow). This is particularly important when querying line-sinks that are part of stream networks, as it will assure that the stream flow reported actually occurs at the vertex that is highlighted.

Discharge-Specified should be checked is the extraction rate of the line sinks in the string is known.

Starting Sink Density Groundwater inflow rate into the stream at the starting point of the line-sink string.
Ending Sink Density Groundwater inflow rate into the stream at the end point of the line-sink string. The line-sinks in the string have a constant inflow rate (sink density) which is calculated at line-sink centers by linearly interpolating the sink densities along the line-sink string between the *Starting Sink Density* and *Ending Sink Density*. Note: if the string consists of only one line-sink its sink density will be the average of the *Starting Sink Density* and the *Ending Sink Density*.

Drain should be checked to create a line-sink string that will extract groundwater, but will not infiltrate water (like the drain cells in MODFLOW*). Drain elements* can only be *Head Specified*. In the near-field drain elements can be given a *Resistance*, *Width* and *Depth* parameter, see the discussion under *Head Specified* line-sinks above. To make the drain element a near-field feature you must uncheck the box *Treat as "far-field"*.

> **Starting Head** Water level in the stream at the starting point of the line-sink string.
> **Ending Head** Water level in the stream at the end point of the line-sink string. The heads at line-sink centers are obtained by linearly interpolating the heads along the line-sink string between the *Starting Head* and *Ending Head*.
> **Resistance** is the thickness of the resistance layer between the surface water feature and the aquifer divided by the average vertical hydraulic conductivity of the resistance layer. See Haitjema (1995) section 5.2.3 pages 234 - 239.
> **Width** is a parameter related to an "effective leakage zone" underneath the surface water body (e.g. ditch) in case it has a bottom resistance (*Resistance* parameter larger than zero). This effective leakage zone depends on the bottom resistance and the aquifer transmissivity and may be calculated as indicated in the document "*Dealing with resistance to flow into surface waters.pdf*" For a summary of calculating the width parameter click here**.** You must do this calculation by hand if you check the radio button "*Unkown*" for the line-sink location. If you check the radio button "*Along surface water boundary*" or "*Along stream center line*" the Solver will automatically do the calculations. For those cases the *width* parameter on the *Linesink String Properties* dialog must be set to the actual stream or lake width. For PEST applications the option "*Unkown*" should not be used, since the Solver must be allowed to recalculate the effective leakage zone based on the changing resistance and transmissivity values. The option "*Unkown*" is included for compatibility with older project files where the *width* parameter has already been set equal to the effective leakage zone or when the resistance parameter is not used for surface water bottom resistances, but, for instance, to represent resistance to three-dimensional flow, see "*Accounting for resistance to 3D flow.pdf*"

Gallery should be checked to create a line-sink string with a given total discharge (pumping rate) and a constant, but a priory unknown head along the line-sink string.

> **Total Discharge** is the total "pumping rate" of the gallery.
> **Starting Head (Minimum)** Lowest possible water level in the gallery at the starting point of the line-sink string.
> **Ending Head (Minimum)** Lowest possible water level in the gallery at the end point of the line-sink string. The heads at line-sink centers are obtained by linearly interpolating the heads along the line-sink string between the *Starting Head* and *Ending Head*.
> **Resistance** is the thickness of the resistance layer between the surface water feature and the aquifer divided by the average vertical hydraulic conductivity of the resistance layer. See Haitjema (1995) section 5.2.3 pages 234 - 239.
> **Width** is a parameter related to an "effective leakage zone" underneath the surface water body (e.g. horizontal well or gallery) in case it has a bottom resistance (*Resistance* parameter larger than zero). This effective leakage zone depends on the bottom resistance and the aquifer transmissivity and may be calculated as indicated in the document "*Dealing with resistance to flow into surface waters.pdf*" For a summary of calculating the width parameter click here. You must do this calculation by hand if you check the radio button "*Unkown*" for

the line-sink location. If you check the radio button "*Along stream center line*" the Solver will automatically do the calculations. For those cases the *width* parameter on the *Linesink String Properties* dialog must be set to the actual stream or lake width. For PEST applications the option "*Unkown*" should not be used, since the Solver must be allowed to recalculate the effective leakage zone based on the changing resistance and transmissivity values. The option "*Unkown*" is included for compatibility with older project files where the *width* parameter has already been set equal to the effective leakage zone or when the resistance parameter is not used for surface water bottom resistances, but, for instance, to represent resistance to three-dimensional flow, see "*Accounting for resistance to 3D flow.pdf*"

Lake should be checked to introduce a lake with an a priory unknown lake stage. The lake stage will be determined during the solution procedure requiring water balance. For a detailed description of this special lake feature, read the document "Lake-Groundwater interactions in GFLOW" accessible on the *Help* menu via the *PDF Help Files* link or at www.haitjema.com
IMPORTANT: When the lake stage is known simply represnet a lake with a string of line-sinks with the radio button *Head-Specified* checked.

Lake Bottom is the lowest lake stage that can be obtained.
Estimated Level 1 and **Estimated Level 2** are two different estimates (a high and a low, or vice versa) for the lake stage.
Resistance is the thickness of the resistance layer between the surface water feature and the aquifer divided by the average vertical hydraulic conductivity of the resistance layer. See Haitjema (1995) section 5.2.3 pages 234 - 239.
Width is a parameter related to an "effective leakage zone" underneath the lake in case it has a bottom resistance (*Resistance* parameter larger than zero). This effective leakage zone depends on the bottom resistance and the aquifer transmissivity and may be calculated as indicated in the document "*Dealing with resistance to flow into surface waters.pdf*" For a summary of calculating the width parameter click here. You must do this calculation by hand if you check the radio button "*Unkown*" for the line-sink location.
If you check the radio button "*Along surface water boundary*" (recommended) the Solver will automatically do the calculations. For those cases the *width* parameter on the *Linesink String Properties* dialog must be set to the actual lake width. For PEST applications the option "*Unkown*" should not be used, since the Solver must be allowed to recalculate the effective leakage zone based on the changing resistance and transmissivity values. The option "*Unkown*" is included for compatibility with older project files where the *width* parameter has already been set equal to the effective leakage zone or when the resistance parameter is not used for surface water bottom resistances, but, for instance, to represent resistance to three-dimensional flow, see "*Accounting for resistance to 3D flow.pdf*"
Depth is the distance between the surface water elevation and the bottom of the resistance layer (like for river cells in MODFLOW). The *Depth* parameter is used to determine when a line-sink represents a percolating surface water feature (unsaturated zone between the stream bottom and the groundwater table).

Selecting the line-sink width parameter

It is recommended to use the *Linesink Location* options *Along stream centerline* or *Along surface water boundary* and enter the actual stream width or lake width as the *Width* parameter. However, when selecting *Unknown* you must select a *Width* parameter as explained in the document "*Dealing with Resistance to Flow into Surface Waters.pdf*", which is available from the *Help* menu in GFLOW (*View PDF Files..)*or found in the user folder *Application Data/GFLOW*. The rules for determining the proper value for the *Width* parameter, as discussed in that document, are summarized below.

Wide streams with line-sinks along both boundaries:
In the following rules *B* is the actual stream width, *w* is the *Width* parameter to be entered on the *General* tab of the *Linesink Properties* dialog and {λ is defined as:

$$\lambda = \sqrt{kHc}$$

where *k* [m/day or ft/day] is the hydraulic conductivity and *H* [m or ft] the saturated aquifer thickness below the stream. The parameter *c* [days] is the *Resistance* as entered on the *Linesink Properties* dialog and equals the thickness of the resistance layer under the stream divided by the vertical conductivity of the resistance layer.
The width *w* follows from:

$$w = \lambda \qquad \lambda \leq 0.1B$$

$$w = \tanh(B/2\lambda) \qquad 0.1B \leq w \leq 2B$$

$$w = B/2 \qquad \lambda \geq 2B$$

Streams with a single line-sink string along the center line:
The same rules apply as above, except that *B/2* should be replaced by *B*.

Line-sinks along a lake boundary:
The same rules apply as for wide streams, whereby the width *B* must be interpreted as the distance between opposite lake boundaries.

> **Note:** The option *Unkown* for *Linesink Location* may also be used to represent resistance to three-dimensional flow, see "*Accounting for resistance to 3D flow.pdf*"

Routing Tab

Use streamflow routing should be checked when the line-sink string is part of a stream network.

End Stream should be checked if the line-sink string forms the down stream part of a stream network, hence if the last line-sink in the string should not be connected to another line-sink string. Important: Each stream network must at least have one string with the *End Stream* box checked!

Inlet Stream should be checked only if the line-sink string is an inlet stream for a "high-k lake", see the document "Modeling Lake-Groundwater interactions in GFLOW" available in the install CD or on www.haitjema.com **Warning:** Do not check this option when the line-sink string forms an inlet stream for a line-sink lake (a lake formed by a line-sink string with the "*Lake"* option checked on the *General* tab).

Outlet Stream should be checked if the line-sink string forms an outlet stream for either a "high-k" lake or a line-sink lake.

Overland Flow represents the total volume of water per unit time that enters the line-sink string other than as groundwater inflow. For instance, it may represent seepage out of shallow water baring formations above the aquifer represented in the model. The *Overland Flow* will be evenly distributed along the line-sink string.

End Inflow represents a total stream flow rate into the first line-sink in the string. It is used to model stream flow in streams which headwaters are not included in the stream network (or model for that matter). This data item is only available for *Head-Specified* line-sink strings.

Evapotranspiration represents the evapotranspiration rate for the lake per unit area. The total evapotranspiration is this rate times the lake area as defined in the stage table, see *Filename for "stage table"* below. This data item is only available for line-sink strings that are *Lake* features.

Precipitation represents the precipitation rate for the lake per unit area. The total precipitation is this rate times the lake area as defined in the stage table, see *Filename for "stage table"* below. This data item is only available for line-sink strings that are *Lake* features.

Filename for "stage table" represents the filename of an ASCII file with extension .stg that contains a table of lake stages and associated lake areas (for *Lake* features) or surface water flow from the lake into the outlet stream (for *Outlet Stream* features). The file must be user supplied and has the following format:

H1 A1
H2 A2
H3 A3
H4 A4
Ö .
Hn An
Quit

The first column contains lake stages (with the same datum as the potentiometric heads in the model) and the second column contains lake areas (for lake features) and outflow rates (volume per unit time) for outlet streams. This table is used by the solver to determine the lake area or outlet flow rate. For lake stages below *H1* the area or outflow rate is *A1*. For lake stages above *Hn* the area or outflow rate is *An*. The area or outflow rate for lake stages between *H1* and *Hn* are obtained through linear interpolation between the values in the table.

Scenarios Tab

By specifying a *symbolic name* for the *Resistance* on this tab sheet and checking the Include in Scenarios option, the line-sink resistance will be included in PEST runs, see *Tools>PEST.* These symbolic names may also be used for calibration or sensitivity analyses, see *Model>Implement Scenarios.*

Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Barrier Properties Dialog

Facilitates the introduction of no-flow boundaries or barriers with some resistance to flow, such as slurry walls. The barrier may be partially penetrating the aquifer, hence not reach all the way down to the aquifer base.

The ***Label*** is a user definable label for the domain or barrier, which must be specified.

The *General* tab sheet

Barrier bounds a closed domain should be checked if a closed domain barrier is to be introduced. When entering vertices, right click after the last vertex is entered to close the domain.

The *Ignore Region* option is only relevant for contour plotting. If none of the options is checked contours will be plotted both inside and outside the domain.
Inside Boundary should be checked, if you want no contours plotted inside the domain.
Outside Boundary should be checked if you want no contours plotted outside the domain.

The *Barrier Properties* have the following meaning:

Conductivity is the hydraulic conductivity of the barrier material.

Thickness is the actual thickness of the barrier.

Porosity is the effective porosity of the barrier. It is important for calculating the residence time of a particle that crosses the barrier.

Bottom Elevation is defaulted to the model aquifer bottom setting as entered on *Model>Settings>Aquifer*. If the barrier has a bottom elevation equal or lower than the aquifer bottom it is fully penetrating. Warning: The GUI does not keep track of changes in bottom elevation resulting from inhomogeneity domains. Make sure that the bottom elevation of the barrier is set in accordance with the local bottom elevation in the area of the barrier.

Click *OK* to enter the vertices of the barrier.

Inhomogeneity Properties Dialog

Facilitates the introduction of recharge areas and areas with differing aquifer properties.
Label is a user definable label for the domain, which must be specified.

The *General* tab sheet

There are two different types of inhomogeneity domains: Recharge areas and areas with differing aquifer properties. Recharge areas are defined as inhomogeneity domains with only an added recharge rate, while the aquifer properties are not changed. Areas of differing aquifer properties may have added recharge, but also have one or more aquifer properties changed from default.

Recharge areas.

Only check the box **Change recharge rate from Default**.
Enter the desired recharge rate in the box ***Added Recharge Rate***.
Note 1**:** Recharge only inhomogeneities may overlap each other or overlap other type of inhomogeneities with an *Added Recharge Rate*. The recharge rates are cumulative! This means that at a point that occurs in more than one inhomogeneity the sum of the specified *Added Recharge Rates* applies.
Note 2**:** Recharge only inhomogeneities may cross any other element at any location.
Note 3**:** Recharge only inhomogeneities may have arbitrary long line elements (line doublets) and be of any shape.
Note 4**:** Recharge only inhomogeneities are not adding equations to the solution procedure in the GFLOW Solver.

Areas with differing aquifer properties.

Before introducing an inhomogeneity with differing aquifer properties it is advised to check the *Line-sink Vertices* option on the *View* menu. This will make all line-sink vertices visible on the graphics screen, which is important to ensure that a line-sink is either entirely inside or outside an inhomogeneity.

Select the desired boxes for:
Change hydraulic conductivity from default,
Change recharge rate from default,
Change porosity from default, and
Change base elevation from default.
Note 1: Domains with a differing aquifer properties may not overlap each other (although they may be nested or abutted) and should intersect line-sink strings at a vertex (select *View>Show Vertices>Line-sink Vertices* to display these vertices).
Note 2: Avoid sharp corners in inhomogeneity domains with differing hydraulic conductivity or base elevation.
Note 3: When the hydraulic conductivity inside an inhomogeneity is more than one or two orders lower than the surrounding value a rather large number of vertices may be needed along the inhomogeneity boundary.

When selecting ***Change base elevation from default*** you can optionally provide an estimate of the average head along the boundary of the domain to improve the convergence of the solution procedure. This option is particularly useful for large models (many equations) that contain many resistance line-sinks and stream features (conjunctive solutions). Check the box ***Provide estimate of average head*** and enter an estimate of the average head in the ***Average Head*** box.
Warning: The default value of the *Average Head* is arbitrarily set equal to the aquifer top, which may be a very poor estimate of the average head along the boundary of the domain. Make sure to provide a more reasonable estimate.
Note 1: Make sure the new base elevation does not exceed the anticipated groundwater table or the aquifer top!
Note 2: Make sure that other elements are adjusted to the new base elevation. For instance, the elevation of the bottom of well screens and the bottom elevation of horizontal barriers must be at or above the newly defined aquifer bottom.
Note 3: The default value of the *Average Head* is arbitrarily set equal to the aquifer top, which may be a very poor estimate of the average head along the boundary of the

domain. Make sure to provide a more reasonable estimate.
Note 4: When you do not check the option *Provide estimate of average head*, the Solver will start the iterative solution procedure using the average of all specified heads in the model domain.

You may provide the inhomogeneity domain boundary with a unique color using the ***Color*** selection option on the Inhomogeneity dialog box.

Click *OK* to start entering vertices for the inhomogeneity domain.
When you entered the last vertex right-click the mouse and the domain will automatically be closed. Remember when entering domains where the box *Change hydraulic conductivity from default or Change base elevation from default* is checked that you should not overlap with other domains, should intersect line-sink strings only at vertices, and should avoid making sharp corners.
Warning: Do not enter a vertex on top of the starting vertex in the string. In other words, when closing the domain do *not left-click* on the first vertex. Instead, enter the vertex at the beginning of the last (closing) line-element and *right-click* the mouse to close the string.

IMPORTANT: **When abutting inhomogeneity domains you must make the vertices on common boundary segments coincide.** This is accomplished by moving the cursor to an existing vertex of the inhomogeneity with which a boundary segment is to be shared, holding down the CRTL key and left-clicking to enter the vertex. The vertex will now be forced to coincide with the existing one.

The *Scenarios* tab sheet

The *Scenarios* tab sheet facilitates the assignment of a symbolic name to the *Conductivity* of the domain and *Added Recharge* of the domain. When this option is selected these parameters will be included in PEST runs (parameter optimization runs), see *Tools>PEST.* These symbolic names may also be used for calibration or sensitivity analyses, see *Model>Implement Scenarios.*

Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Model Menu

The *Model* menu controls the operation of the analytic element solver and provides access to its results. The following options are available:

Settings
Solve
Continue Solve
Add Particle
Add Particle Grid
Add Test Point
Add Flux Inspection Line
Add a Hydrologic Unit Boundary-
Add a FarField Boundary-

Settings

Aquifer Tab

Aquifer Properties
Base elevation is the elevation with respect to mean sea level of a horizontal no-flow boundary that forms the aquifer base. In case the base varies throughout the flow domain areas with a different base elevation can be defined using *Inhomogeneities*, see *Element* menu. The *Base elevation* specified on this *Aquifer* tab is the elevation outside these inhomogeneity areas.

Thickness is the total thickness of the aquifer material, whether saturated or not! In case the aquifer is unconfined a sufficiently large thickness should be specified to ensure that the water table will never reach the aquifer top, which elevation is the *Base Elevation* plus the *Thickness*. The actual saturated thickness of the unconfined aquifer will depend on the groundwater flow solution. Make sure that the aquifer base plus the aquifer thickness is more than the highest aquifer bottom elevation set in inhomogeneities!

Hydraulic Conductivity specified here will be used throughout the model domain. In case the hydraulic conductivity in reality varies over the aquifer domain areas of different hydraulic conductivity can be defined using *Inhomogeneities*, see *Element* menu. The *Hydraulic Conductivity* specified on this *Aquifer* tab is the hydraulic conductivity outside these inhomogeneity areas.

Porosity in GFLOW is the effective porosity that is used during particle tracing for the calculation of travel times. The (effective) porosity should be entered as a fraction, NOT as a percentage! In case the porosity in reality varies over the aquifer domain areas of different porosity can be defined using *Inhomogeneities*, see *Element* menu. The *Porosity* specified on this *Aquifer* tab is the porosity outside these inhomogeneity areas.

Interface Flow
Check *Add a Saltwater Interface* to implement steady state interface flow in GFLOW. The salt water is at rest and the interface elevation is calculated using the Ghyben-Herzberg relationship. The Solver uses a discharge potential formulation as proposed by Strack

(1989) section 10, pages 96 - 112.

Fresh Water Specific Gravity [dimensionless] is the unit weight of the fresh water in the aquifer divided by the unit weight of pure water at 4 degrees Celsius.

Salt Water Specific Gravity [dimensionless] is the unit weight of the salt water in the aquifer divided by the unit weight of pure water at 4 degrees Celsius.

Average Sea Level is the local average sea level measured with respect to the same datum as the heads in the aquifer are measured, e.g. mean sea level (msl).

Note: There are some limitations to the use of interface flow in GFLOW. Read the PDF document "Freshwater and Salt Water Interface Flow in GFLOW," which is found in c:\Program Files\GFLOW\Documents before using this feature. You can access this document from the *Help* menu, you may also access the document at www.haitjema.com

Scenarios

You may define a symbolic name for the regional *hydraulic conductivity* and the *aquifer base* defined on the *Aquifer* tab. When this option is selected these parameters will be included in PEST runs, see *Tools>PEST.* These symbolic names may also be used for calibration or sensitivity analyses, see *Model>Implement Scenarios.*

Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Contouring Tab

Contour Type

Check the *Compute Contours* box to instruct the solver to generate contours of one of the following: Heads, Potentials, Discharge, Interface, Stream Function, or Drawdowns. The *Contour Type* is selected by checking the appropriate radio button.

Heads for a potentiometric contours (will jump across horizontal barriers).

Potentials for equipotentials (discharge potentials) (will jump across inhomogeneity domain boundaries and horizontal barriers)

Discharge for contours of equal absolute values of the discharge vector.

Interface for elevation contours of the fresh water - salt water interface.

Stream Function to create stream lines. The stream function is only valid in the absence of areal recharge! This feature is most useful in cross-sectional models, see "*Modeling flow in vertical plane.pdf*" in the documents folder. To minimize the thickness of the branch cut the grid resolution should be selected at 80 or above, see "Grid Resolution" below.
For further reading on the use of the stream function, see *Analytic Element Modeling of Groundwater Flow, H.M. Haitjema, Academic Press, 1995* page 40 through 57. You may also read the document "*Modeling flow in the vertical plane with GFLOW*," which can be accessed from the *Help* menu as a PDF file.

Drawdowns due to a well, for instance, can be shown as follows. First create a contour plot of heads without the well pumping. Next, open the *Model Settings Contouring* tab and select *Drawdowns*, but make sure not to change the grid resolution and not to change the window settings. Add the well and solve again. The current heads will be subtracted from the previously generated grid with heads and the difference (the drawdowns) will be contoured and plotted. Since two grids with head values will be subtracted it is essential that the domain (window setting) and grid resolution are the same.

Contour Levels
The *Minimum Contour* value is the lowest level that will be contoured.
The *Maximum Contour* value is the highest level that will be contoured.
The *Contour Interval* is the increment between successive contours.

Make sure that the values you enter are within the expected range of contour levels. For instance, heads near streams will be close to the water levels in the streams. In between streams there may be some mounding (rise in head) due to recharge, while near wells there may be some drawdown (lowering) of the heads.

Note: When the *Stream Function* option is selected the *Contour Interval* will be replaced by the *Number of stream lines* to be contoured.

Grid Resolution
The default setting is *Course*, which means that 40 grid points will be distributed in horizontal direction over the domain in the graphics window, while points will be distributed in vertical direction at the same spatial distance as in horizontal direction. The *Course* setting is adequate for inspecting most intermediate modeling results.
The *Detailed* setting is double the resolution of the *Course* setting (80 grid points) and useful in the presence of jumps in the head, e.g. across horizontal barriers.
The *Custom* setting allows the user to specify the resolution up to a maximum of 200 grid points (in horizontal direction). Such a high resolution allows for highly detailed contour plots. A very high resolution is desirable to reduce the branch cut thickness when contouring the *Stream Function*. For further reading on the use of the stream function, see *Analytic Element Modeling of Groundwater Flow, H.M. Haitjema, Academic Press, 1995* page 40 through 57. You may also read the document "*Modeling flow in the vertical plane with GFLOW*," which can be accessed from the *Help* menu as a PDF file.

Tracing Tab

Check *Compute Particle Paths* to instruct the solver to generate path lines of particles starting at the well or at specified particle locations as specified by *Add Particles* on the *Model* menu.

The *Maximum Travel Time* defines the extent of the particle trace up to this travel time. For instance, if five-year time of travel capture zones are called for you may set the *Maximum Travel Time* to 1825 days. This will cause the path lines starting at the well to stop when the average groundwater travel time toward the well is 5 years.

Check the *Use Default Step Size of* box for normal operation. In case the particle traces seem inaccurate a smaller step size may be entered in the dialog box after *Tracing Step*

Size, by first deselecting the *Use Default Step Size of*. The step size is the approximate distance in feet or meters that is traversed during every step along the particle path line.

Note: you may perform instant path line traces from the cursor location using a *Ctrl-Shift left-click*. To set the properties for an instant trace go to *Tools>Instant Inspector Settings*. Particles cause (new) traces after each new solution or change on the *Model>Settings>Contour* or *Model>Settings>Trace* tab sheet. Instant traces are not repeated (no particle location is being stored).

Solver Tab

Number of Iterations (inner loop)

These iterations are necessary to deal with non-linear equations that result from streams with resistance under unconfined flow conditions, inhomogeneity domains with changing bottom elevation under unconfined flow conditions, partially penetrating horizontal barriers under unconfined flow conditions, and stream flow solutions that change the line sink configuration between iterations.
For problems without any of these features a single iteration should suffice, although a slightly more accurate solution may result from GFLOW's iterative refinement strategy when selecting 2 or 3 iterations. Most non-linear problems will converge in less than 6 iterations, but conjunctive surface water and groundwater solutions may require 6 to 10 iterations or more.
The ***Advanced*** button behind the *Number of Iterations (inner loop)* allows the user to specify a convergence criterion for each of the analytic element types supported by GFLOW. If all criteria are met the solution procedure is completed, hence the specified number of iterations becomes the maximum number of iterations. This feature is particularly useful when performing many batch solver runs, for instance, during parameter optimization with PEST. The specification of the convergence criteria is realized by editing a file "converge.tab." Simply follow the directions in the comment statements of that file.
Note: The *minimum number of iterations* setting in "converge.tab" dictates the minimum number of inner loop iterations in a single outer loop iteration.

Apply outside loop iterations. Number of iterations**:** Check if one or more line-sink lake features are included in the model and specify the number of times the lake stage is to be updated. Each update requires the completion of the number of inner loop iterations. Warning: Use only when "line-sink lake" features are present ("lake" option on the line-sink dialog).

The ***Conjunctive Surface Water*** - ***Groundwater Solution*** box must be checked to implement conjunctive solutions. It may be unchecked, however, even if one or more stream networks are defined. In that case, GFLOW simply does not apply the surface water calculations and treats the stream networks as regular strings of line-sinks.
The ***Advanced*** button behind Conjunctive Surface Water - Groundwater Solution allows the user to specify an *under-relaxation* parameter to dampen stream flow variations during the iteration process. This may increase solution stability, but also increases the number of required iterations. The specification of these under-relaxation factors is realized by editing the file "relax.tab." Simply follow the directions in the comment statements of that file.

Store decomposed matrix on disk for faster solve Check to speed up the solution procedure by storing the decomposed matrix on disk. Under certain circumstances this option may lead to unstable (diverging) solutions when the problem includes resistance line-sinks and/or resistance barriers and /or inhomogeneity domains with a jump in the bottom elevation. The option is also not available when a fresh-salt water interface is present ("interface" option on the Aquifer tab of the Model>Settings dialog). The option is checked as default.

Suppress reporting for faster solve. Check to avoid error reports on groundwater flow boundary conditions and stream flow to speed up the solution process. An error report will still be provided after the last iteration. This option will only lead to noticeable speedup for large models (> 300 equations) and is specifically useful for batch runs (e.g. PEST applications).

View error log file after solve. If checked the *error.log* file that contains a solver error report will automatically be opened after the solver ends. If the file contains no errors, it will not be displayed, regardless of this setting.

View message.log file after solve. If checked the *message.log* file containing a solver report on the solution process will automatically be opened after the solver ends. This file contains a listing of the maximum errors detected at collocation points for the various analytic elements as well as responses related to the stream flow solution (for conjunctive solutions).

Note: If the above two options are unchecked, the files *error.log* and *message.log* can still be opened from *Model>View Model Run Files..*

Solver Convergence Criteria

In this file "converge.tab" you can set a convergence criterion for each of the analytic element types supported by GFLOW. If all criteria are met the solution procedure is completed, hence the specified number of iterations becomes the maximum number of iterations.
This feature is particularly useful when performing many batch solver runs, for instance, during parameter optimization with PEST.
The specification of the convergence criteria is realized by editing a file "converge.tab." This file must reside in the working directory where the Solver is also finding the input file "basename.dat."
An example file with default values is shown below:

```
* test file "converge.tab"
*
* Note: all lines below must be present in that order!
*
  minimum iterations    50
  reference point       0.0001
  linesinks dirichlet   0.0001
  linesinks resistance  0.1
  inhomogeneity domains 0.01
  barriers noflow       0.001
  barriers resistance   0.01
  wells 3D              0.01
  wells 2D              0.0001
  quit
*
```

Note, in this example, that line-sinks with a specified resistance are considered sufficiently accurate when the Cauchy boundary condition is met within 0.1%, while Dirichlet boundaries (head specified) have been given a much stricter convergence criterion; they must be met within 0.0001%.
When performing PEST runs, it is recommended that you perform a few GFLOW runs with bounding parameter values (use *Model>Implement Scenario* option) and observe the maximum errors in boundary conditions as reported in the "message.log" file. Use these experimental runs to set a maximum number of iterations and appropriate convergence criteria for the various analytic element types.

Surface Water Solution Relaxation Parameter

The file "relax.tab" should be in the working directory for the project. It controls the surface water calculations during conjunctive solutions. For relatively simple problems the conjunctive surface water and groundwater solution procedure will converge fine with no under relaxation for the surface water calculations (relaxation = 1). However, for some more complex problems, including extensive stream networks, under relaxation may improve solution stability. Under relaxation, however, will increase the number of iterations required to obtain convergence.

An example default file "relax.tab" is shown below:

```
* Format of the file "Relax.tab":
*
* * comment statement (* in first column)
* * iteration #     relaxation factor
*   1               0.5
*   5               0.8
*   8               0.9
* quit
*
*     The effect of this file is as follows: During the first 5 iterations a (under)relaxation factor
*     of 0.5 is used. During the next 3 iterations a relaxation factor of 0.8 is used, while after
*     that the relaxation factor is set to 0.9.
*     NOTE: The relaxation factor should NEVER be set larger than 1.0
*
1 1
quit
```

Note 1: The current setting, as seen at the bottom of the file, is that the relaxation factor is set to 1 starting at iteration 1, hence no under relaxation is being used.

Note 2: For some runs it may be necessary to keep sufficient under relaxation for all iterations, for instance never raise the relaxation factor above 0.75.

Solve

Click to start the solution procedure. The Solver (GFLOW1.EXE) will be scheduled and runs in the DOS box. The messages that scroll over the screen contain errors at boundary conditions and errors in stream network calculations. These messages are echoed to the file "message.log", which can be accessed by *Model>View Model Run Files>Runtime Message File*. After a solution has been obtained and after a redraw, you should check the *Runtime message File* to verify the accuracy of the solution.
In case a valid solution already exists (which means you did not alter critial data in the model), you are asked to *Resolve* or to use the current solution for new contours,

pathlines, or flux inspector results. These options do not require a resolve.

Continue Solve

Number of additional iterations (inner loop): defines the number of surface water (if conjunctive solutions are defined on *Model>Settings>Solver* tab) and groundwater flow solutions.

Apply outside loop iterations. Number of iterations: Check if one or more line-sink lake features are included in the model and specify the number of times the lake stage is to be updated. Each update requires the completion of the number of inner loop iterations.

Suppress reporting for faster solve Check to avoid error reports on groundwater flow boundary conditions and stream flow to speed up the solution process. An error report will be provided after the last iteration. This will only lead to noticeable speedup for large models (> 300 equations).

Store decomposed matrix on disk for faster solve Check to speed up the solution procedure by storing the decomposed matrix on disk. Under certain circumstances this option may lead to unstable (diverging) solutions when the problem includes resistance line-sinks and/or resistance barriers and /or inhomogeneity domains with a jump in the bottom elevation.

View error log file after solve. If checked the error.log file that contains a solver error report will automatically be opened after the solver ends.

View message.log file after solve. If checked the message.log file containing solver output on the solution process will automatically be opened after the solver ends. This file contains a listing of the maximum errors detected at collocation points for the various analytic elements as well as responses related to the stream flow solution (for conjunctive solutions).

Particle Properties Dialog

This box will appear after having defined a particle location using the *"Add Particle"* option on the *Model* menu. The particles will form the starting point of path lines. After that select *Forward in time* or *Backward in time* as tracing options. Backward in time will cause the particle to be traced in a direction opposite the groundwater flow direction. The *Starting Elevation* of the particle may be set at the water table or aquifer top by checking the box in front of "*Start at top of saturated thickness."* Unchecking that box allows the specification of a particle starting elevation.
Note 1: The *starting elevation* must be inside the aquifer (between the local aquifer bottom and aquifer top). In case the particle elevation is inside the aquifer, but above the water table, the Solver will drop the particle to the water table and start the trace.

Note 2: If the starting elevation is at the water table and the option Backward in time is selected, no path line may appear. This is due to the fact that the particle moves opposite the direction of flow and thus immediately leaves the aquifer (if recharge is present).

Note 3: You may perform instant path line traces from the cursor location using a *Ctrl-Shift left-click*. To set the properties for an instant trace go to *Tools>Instant Inspector Settings*. Particles cause (new) traces after each new solution or change on the *Model>Settings>Contour* or *Model>Settings>Trace* tab sheet. Instant traces are not repeated (no particle location is being stored).

Particle Properties Dialog

This box will appear after having defined a window in which particles are to be distributed using the *Add Particle Grid..* option on the *Model* menu. The particles will form the starting point of path lines. Specify the *Number of Particles* you want distributed in the window. After that select *Forward in time* or *Backward in time* as tracing options. Backward in time will cause the particle to be traced in a direction opposite the groundwater flow direction.
The *Starting Elevation* of the particle may be set at the water table or aquifer top by checking the box in front of "*Start at top of saturated thickness."* Unchecking that box allows the specification of a particle starting elevation.
Note 1: The *starting elevation* must be inside the aquifer (between the local aquifer bottom and aquifer top). In case the particle elevation is inside the aquifer, but above the water table, the Solver will drop the particle to the water table and start the trace.

Note 2: If the starting elevation is at the water table and the option Backward in time is selected, no path line may appear. This is due to the fact that the particle moves opposite the direction of flow and thus immediately leaves the aquifer (if recharge is present).

Note 3: You may perform instant path line traces from the cursor location using a *Ctrl-Shift left-click*. To set the properties for an instant trace go to *Tools>Instant Inspector Settings*. Particles cause (new) traces after each new solution or change on the *Model>Settings>Contour* or *Model>Settings>Trace* tab sheet. Instant traces are not repeated (no particle location is being stored).

Test Point Properties Dialog

Facilitates the definition of locations where the groundwater elevation or potentiometric head is known (*piezometers*) or where the stream flow in a stream is known (*gages*), or where a lake stage is known (*Lake Stage*). These test points are used during model calibration. They are also used as calibration targets during parameter optimization with PEST, see *Tools>PEST*.
Select the radio button for *Piezometer, Gage,* or *Lake Stage* to enter a test point with an observed head, observed stream flow, or observed lake stage, respectively.

When adding a *Piezometer* place it at the location of a well with a reported static water level or a gravel pit, etc.

When adding a *Gage*, place it next to the end vertex of the line-sink for which you want to specify an observed stream flow. Note: that line-sink must be part of a stream network and you must have specified a "conjunctive surface water - groundwater solution" on the *Model>Settings>Solver* tab to get a meaningful test point response.

When adding a *Lake Stage* place it near one of the line-sinks of a line-sink lake feature, see *Lake* option on the *Linesink String Properties* dialog. Note: Make sure that outer loop iterations are specified, see *Model>Settings>Solver* tab in order to get a meaningful test point response..

A *Label* may be specified to identify the test point on the map. The label will not be posted on the map, but can be requested by clicking on the test point marker and clicking on properties on the edit menu.
Note: The label should be limited to 20 characters and not contain any spaces

For *Piezometer* the known potentiometric head at the test point is entered in the box after *Observed head*.

For *Gage* the known stream flow in the stream section near the test point is entered in the box after *Observed Streamflow*.

For *Lake Stage* the known lake stage elevation for the lake in which the test point is placed is entered in the box after *Observed Stage*.

Flux Inspection Line Properties

Facilitates the definition of lines across which the groundwater flux will be calculated. The "flux" is calculated as the total flow (ft3/day or m3/day) over the aquifer height and along the poly- line.
The flux inspection poly-lines are plotted on the base map as a purple dashed line or poly-line.

Flux inspection lines have an orientation: the first point entered is the starting point, marked with a small diamond. A **positive flux** implies water flowing from left to right across the line, when viewed from the starting point!

The *Name* is a user definable label to identify the flux inspection line.

Hydrologic Unit Domains

Hydologic Unit domains are often referred to as HUCs (HUC: Hydrologic Unit Code). These domains are sub-watersheds that typically have only one inlet stream and one outlet stream, although this is not required for the implementation in GFLOW. A large model domain may be subdivided into HUCs and a subset of these HUCs may be read into GFLOW for a local model (see *Tools>Import>Hydologic Unit*). At least one HUC is read in as a "nearfield" domain, while the surrounding HUCs must be read in as "farfield" domains. Inhomogeneities that redefine aquifer properties and recharge rates are also read in from separate files. The HUCs reside on disk in ASCI format files.

Before using this feature the user should read the document "Managing HUCs in GFLOW" stored as *Managing HUCs.pdf* in the document folder *C:\Program Files\GFLOW\Documents*.

HUCs are defined in GFLOW by entering polygons as follows:

Add Hydrologic Unit Boundary.. Allows the user to enter a name (e.g. the Hydrologic Unit Code) and select a color for the boundary. Click OK and enter a polygon around the stream and its tributaries that define the sub-watershed that is the hydrologic unit.

Add FarField Boundary.. This option will only become available if a HUC boundary exists and is selected (highlighted). The label for the FarField boundary is predefined based on the label for the HUC boundary and cannot be altered. You can select a color. Enter a polygon outside the HUC boundary that includes all stream sections that you want treated as nearfield when modeling the current HUC as nearfield. Note: only the starting vertex of a stream reach has to be inside the FarField boundary for the entire stream reach to be treated as nearfield. All stream reaches that occur outside the HUC boundary, but inside the FarField boundary will be defined as "End Streams," which means that, in case they are loosing water, they may fall dry in the absence of stream flow. However they are not networked (do not contribute water to the stream reaches to which they connect).

Add Model Boundary.. This option will only become available if a HUC boundary exists and is selected (highlighted). The label for the Model boundary is predefined based on the label for the HUC boundary and cannot be altered. You can select a color. Enter a polygon outside the HUC boundary and outside the FarField boundary that includes all stream reaches that you want to keep in the model when the current HUC is modeled as a nearfield feature. This boundary is optional and intended to limit the model size (number of farfield line-sinks in the model) to increase computational efficiency.

For further reading refer to the document "Managing HUCs.pdf" in the document folder *C:\Program Files\GFLOW\Documents*.

Model Size

The total number of wells, line-sink strings, line-sinks, inhomogeneities, inhomogeneity elements, barriers, and barrier elements are reported. Each line-sink adds one equation, each barrier element adds two equations, and each inhomogeneity element adds two or three equations to the model. For fast interactive model response the number of equations should be limited to a few hundred or no more than a thousand. Models with more than a thousand elements are considered very large. A beginners mistake is to use too detailed a line-sink representation of streams, particularly in the far field. Only in the immediate area of interest is a more detailed representation warranted. It is best to start with relatively few detail (a few hundred equations) and add detail as needed.

View Results Table

Provide access to the some relevant data from both the GUI and the Solver, including computational results.

View
Well
Displays data for all wells. *GFLOWlabel* is the GUI generated label used by the Solver and reported as part of error messages from the Solver.

Line-sinks
Displays data for all line-sinks. *GFLOWlabel* is the GUI generated label used by the Solver and reported as part of error messages from the Solver.

Test Points
Displays data for all test points (piezometers). *VID* is the GUI vertex identification number, which may be ignored.

Flux Inspection Lines
Displays data for all flux inspection lines. *PID* is the GUI polyline identification number, which may be ignored. *TotalFlux* is the total flow (cubic meters per day or cubic feet per day) across the flux inspection line.

Inhomogeneities
Displays data for all inhomogeneity domains. Many of the data items are of little interest to the user, except for the total domain area in the 7th column.

Edit
Copy
Will copy all columns to the clipboard for pasting in a spreadsheet, for instance, in Microsoft Excel.

View Calibration Statistics

The *View Calibration Statistics* option on the *Model* menu provides access to some statistics on the comparison between:

modeled and observed heads (piezometers)
modeled and observed stream flow (gages)
modeled and observed lake stages (lakestages)

These statistics are provided on three different tab sheets and based on the three different *Test Point* types: piezometers, gages, and lake stages.

Caution is to be used in interpreting these results when only few *Test Points* are available, see *Number of Observations* box. For meaningful statistics it may be advisable to only use *Test Points* in the near field. These statistics are best used in conjunction with the graphical display of head differences, stream flow differences, and lake stage differences at *Test Points* to identify trends and to distinguish between areas of better and lesser fits between observed and measured values. See the *Test Points* option on the *View* menu.

The calibration results can be copied to the clipboard using the *Edit>Copy Chart* option. To view the numerical data simply *Paste* the clipboard contents to Word or Excel. To view the graph, you must use the option *Edit>Paste Special* in Word or Excel.

View Model Run Files

The following files can be accessed:

Input File: ASCII script file with instructions and data produced by the GUI and read by the solver. Filename: *basename.dat*

Error Log File: ASCII response file from the solver with error messages. The file will contain only an * on the first line if no errors were found. Filename: *error.log*

Contour File: ASCII file with vertices of contour lines produced by the solver and used by the GUI. Filename: *basename.ctr*

Trace File: ASCII file with vertices of path line traces produced by the solver and used by the GUI. Filename: *basename.pth*

Inquiry File: ASCII file with solver results requested by the GUI. Filename: *basename.xtr*

Runtime Message File: ASCII file with the responses from the solver that scroll over the screen during a solve operation. This file documents, among others, the progress of the solution during successive iterations. For further details see Runtime Message File
Filename: *message.log*

Command Echo File: ASCII file with a copy of all the command that were received by the solver. Filename: *echo.log*

Note: As a rule the user does only have to access the *Runtime Message File*.

Runtime Message File

The file is an echo of Solver messages generated during the solution process. For each iteration a solution progress report is given and errors at boundary conditions are provided. The errors are relative errors in % of the head or flow or other relevant property. The % errors have the following meaning:

Head specified line-sinks--- difference between the calculated and specified head as a percentage of the specified head measured with respect to the aquifer base.
Resistance specified line-sinks---- difference between the inflow into the line-sink calculated by Darcy's law in the vertical direction (flux across the resistance layer) and the actual sink density of the line-sink as a percentage of the average of the two.
Line doublets in inhomogeneity domains---- difference between the heads on either side of the line-doublet as a percentage of the head. This difference should be zero (no jump in the head).
Line doublets in horizontal leaky barriers---- difference between the actual flow across a barrier segment (approx. line doublet length) and the flow calculated from the line doublet strength as a percentage of the actual flow.
Line doublets in horizontal no-flow barriers---- total flow across a barrier segment (approx. line doublet length).

The errors reported are the largest found for the group of elements.

> In addition to these summary error reports, you may inspect the data (and errors) of individual elements. To do so, click the *View Results Table* on the *Model* menu. Select *Wells*, *Line-sinks, or Inhomogeneities* and pertinent data will be listed in a table. You may also right-click on a selected element to open a data box that includes the error. Finally, errors can be visualized using the *View>Results Overlay* option and selecting to plot line thicknesses proportional to the % errors.

For conjunctive surface water and groundwater flow solutions the stream flow solution also produces error messages, these are:

Negative streamflow in line-sink Ö the line-sink is part of a stream network and is found to infiltrate more water than the available stream flow. The infiltration rate is limited to the amount of available streamflow and a new solution is required.
Underinfiltration in line-sink Ö .the line-sink is part of a stream network and does not

infiltrate enough water based on Darcy's law in vertical direction. The line-sink was probably discharge specified in the previous iteration, but the groundwater table has dropped and the line-sink should become head specified again. A new iteration is needed.
Overinfiltration in line-sinkÖ . the line-sink is part of a stream network and is found to infiltrate more water than it should based on Darcy's law in vertical direction. The line-sink was probably discharge specified (limited in infiltration) in the previous iteration, but the new groundwater table is higher and the line-sink should become head specified again. A new iteration is needed.

For models with one or more line-sink lake features a water balance report for the lake is included in the *Runtime Message File*, for each inner loop.

In case the option *Suppress error reports for faster iterations* is checked on *Model>Settings>Solver* or *Model>Continue Solve*, the error reports will only occur at the end of the last iterations.

Time

The *Time* option allows the specification of a time for which a contour plot is to be generated in case transient wells are specified (use of Theis' solution), see the *Well* dialog box. If no transient wells are specified, the time setting is ignored.

Inplement Scenario

Facilitates the implementation of scenarios. This feature facilitates trial and error calibration procedures or sensitivity testing by temporarily modifying sets of related parameters as defined by symbolic names on the *Scenario* tab sheets.

Different scenarios of property values can be run by selecting alternative *Current* values. The *Default* values are retained. At the end of one or more scenario runs you may selectively replace *Default* values with *Current* values, replace all *Default* values with *Current* values, or return back to the initial values by replacing all *Current* values by the *Default* values.

All symbolic names for global aquifer properties, inhomogeneity properties, and line-sink resistances are listed together with their *Current* values and the associated *Default* values. The *Default* values are the values entered on the various *Properties* forms. The *Current* values are set equal to these defaults, but can be modified in the tables on this *Implement Scenario* dialog. When clicking on *Apply*, these *Current* values replace the values entered on the various *Properties* dialogs, however, the original values (*Defaults*) are retained for later use. A subsequent model run will use these new *Current* values and they will appear on the *Properties* dialogs.

When using this option for purposes of sensitivity testing, the *Current* values must be reset to the original *Default* values at the end of the sensitivity testing process. This may be done by clicking *Replace all Current values by Default values* on the *Implement Scenario* dialog.

When using this option for calibration purposes, you may want to make the entire set of *Current* values the new *Default* values after the calibration process is completed. This may be done by clicking *Replace all Default values by Current values* on the *Implement Scenario* dialog.

Note 1: Changes made on the *Implement Scenario* tables are only saved to the database (thus in effect) after clicking the *Apply* button.

Note 2: You may replace all *Current* values by the results of a PEST parameter optimization run using the *Import Optimized Parameters* button on the *Tools>PEST* control panel. You may then decide to make these parameters *Defaults* or not.

Note 3: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Grid Menu

The *Grid* menu facilitates the extraction of a MODFLOW model from the GFLOW model domain or the creation of a hybrid GFLOW - MODFLOW model by adding a grid of leakage values obtained from a multi-layer MODFLOW model at the bottom of the GFLOW model. These two options are accessed by the following submenus of the *Grid* menu, respectively.
MODFLOW Extract
Leakage from MODFLOW

MODFLOW Extract

Facilitates the extraction of a MODFLOW model by designing a MODFLOW grid covering part of the GFLOW model area. The grid may be scaled, rotated and shifted over the model domain. Gridlines may be shifted for local grid refinement. Aquifer properties and boundary conditions are assigned to the grid cells based on the existing GFLOW solution.
To create a grid, click on *Grid* on the menu bar and click on *MODFLOW Extract>Show Grid Dialog*. The *Finite Difference Grid* dialog contains four menus: *File*, *Edit*, *View* and *Processing*. To create a new grid the *New Grid* option on the *File* menu must be selected. A dialog box will open that allows you to select the grid geometry. After clicking on the *Create Grid* button the grid can be oriented on the map. Using the *Processing* menu the boundary conditions and aquifer properties are assigned and MODFLOW files are written.

There are four menu options on the *Finite Difference Grid* dialog of *MODFLOW Extract:*

File

Edit

View

Processing

File

Click on *New Grid* when you want to create a new grid rather than edit an existing grid. The *New Grid* option will open the *MODFLOW Extract* dialog box to select grid geometry properties. The *MODFLOW Extract* dialog box has three tabs: *Grid*, *Layers* and

Tracking.
Start with the *Grid* tab.
The lower left corner of the grid defaults at the origin of a local coordinate system. You do not have to modify the *Lower Left X* and *Lower left Y* boxes, which values will be automatically assigned when locating the grid on the map. Grid dimensions and resolution are determined by selecting *Row Spacing and Column Spacin*g and the number of *Rows* and *Columns* in the grid. *Rotation* is the orientation of the rows measured with respect to a horizontal line. For instance, a 30 degree rotation tilts the grid in counter clockwise direction over 30 degrees. Select the number of layers in the grid. *Layer Thickness* refers to the default thickness of a model layer. The anisotropy is the horizontal hydraulic conductivity (Kh) over the vertical hydraulic conductivity (Kv). The grid *Perimeter* may be either *Head Specified* or *Flux Specified*, using data obtained from the GFLOW solution. When selecting a *Flux Specified* condition the heads at the grid perimeter in MODFLOW must match those in GFLOW. If not, the MODFLOW grid may be too small to let differences in the near-field disappear at the grid boundaries. Alternatively, modifications in MODFLOW may have altered the far-field conditions and a new GFLOW solution may be necessary to adapt the grid perimeter conditions to these modifications. The grid location is determined graphically, after the *Create Grid* button is clicked, by shifting a grid outline on the basemap.

Select the *Layers* tab to change the top and bottom of each grid layer from its default. Remember that MODFLOW numbers layers from top to bottom. Note: Later, when *Applying GFLOW Results*, (see Processing menu.) the GFLOW aquifer bottom elevations are assigned to each grid layer, provided the elevation is larger than the bottom elevation specified on the *Layers* tab. To correctly implement all GFLOW bottom elevations, make sure that the bottom of the lowest layer is set at or below the lowest bottom elevation in GFLOW. *Applying GFLOW Results* can force bottom elevations above the top of a layer. This situation should be avoided by proper choices of top elevations on the *Layers* tab or it should be corrected in the MODFLOW graphical user interface (e.g. Groundwater Vistas), for instance by changing such cells into "inactive cells".

Select the *Tracking tab* to monitor the world and grid coordinates when rolling the mouse over the grid. The aquifer properties on the tracking tab will display *not available* at this time, because no GFLOW data has been applied to the grid yet, see the Processing menu.

Click on *Open Grid* when you want to edit an existing grid.
Click on *Save Grid* to save the currently selected grid to its existing name (overwrite with changes).
Click on *Save Grid As* if you want to save the currently selected grid under a new name (make a copy with or without changes).
Click on *Close Grid* if you want to start over creating or opening a grid.
Click on *Exit Grid Dialog* if you are done creating or editing a grid.

Edit

Click on *Cell Spacing* to redefine the grid properties. Move the cursor to a cell and click to change the row and column spacing for all cells that share the row and column with the selected cell. Note: this will shift all grid lines and thus change the overall grid dimensions. It is **important** to first refine the grid and then select the *Processing* menu to apply the GFLOW boundary conditions and GFLOW aquifer properties. If *Cell Spacing* is changed after processing the GFLOW data the latter procedure must be repeated, by again selecting the Processing menu. This is necessary to avoid shifting boundaries when changing grid configurations.

Check *MODFLOW Elements* to facilitate the inspection of boundary conditions in the MODFLOW grid, which is only meaningful after the *Processing* of GFLOW data has been completed. By clicking on a constant head cell, river cell, etc. the type of cell and the data applied to the cell are displayed in a message box.

View

You may select which type of cells should be visible:

Constant Head Cells, when checked, will be blue.
Well Cells, when checked, will be red.
River Cells, when checked, will be green.
Drains, when checked, will be yellow.

Processing

Select the *Processing* menu to extract data from the GFLOW model and apply it to the MODFLOW grid. You should be done editing (refining) the grid before using this menu. Also, before executing *Apply Analytic Elements* you should make sure that line-sinks are either entirely inside the grid or entirely outside the grid, see the "**Important**" note below.

Click *Apply Analytic Elements* to transfer the boundary conditions in GFLOW to the MODFLOW grid. Cells that coincide with line-sinks will be assigned the proper head and surface water properties (resistance, width, depth, etc.). In the MODFLOW grid the resistance of the line-sink will be converted into a cell conductance. Depending on the function of the line-sinks in GFLOW the cells in MODFLOW will become constant head cells, river cells or drains. A well will transform the nearest cell into a well cell. Inhomogeneities will alter the aquifer properties of the cells that coincide with them.
Important: Line-sinks should not cross the grid perimeter. Before executing this step you should move the nearest vertex in a line-sink string that intersects the grid perimeter to that perimeter. This way a line-sink is either entirely inside the grid or entirely outside the grid.

Click *Extract GFLOW Results* to assign heads to the interior MODFLOW grid cells and heads or fluxes to the perimeter cells, depending on what was selected on the *Grid tab*. These heads and fluxes, as computed by GFLOW, define the perimeter boundary conditions and precondition the MODFLOW model for fast convergence.
After these choices the aquifer properties on the *Tracking tab* of the *Finite Difference Grid* dialog box will show the cell by cell properties when rolling the mouse over the grid.
Important: While the *Tracking* feature shows a porosity value inside the grid, MODFLOW, because of its design, will not read porosity data. Consequently, when using MODPATH default porosity values of the MODFLOW/MODPATH graphical user interface will be used. The user may have to replace these default values in the MODFLOW/MODPATH graphical user interface in order to obtain correct time of travel particle traces in MODPATH.
Note: The grid layer(s) must be given a bottom and a top elevation on the *Layers* tab. Bottom elevations obtained from GFLOW will be implemented in these layers, except that the bottom of a layer will not be set lower than the elevation specified on the *Layers* tab. It is important, therefore, to specify the bottom of the lowest layer below or at the lowest GFLOW bottom elevation and let GFLOW set the actual elevations. When more

than one layer is defined, however, it is possible that the bottom elevation in a layer is set above the top elevation of that layer. This may be avoided by a proper selection of *Top Elevation* on the *Layers* tab or may be corrected in the MODFLOW graphical user interface (e.g. Groundwater Vistas) by changing cells with zero thickness into "inactive cells".

Complete the procedure by clicking on *Make MODFLOW Files* on the *Processing* menu of the *MODFLOW Extract* dialog box. Make sure to select the proper folder and desired filename for the set of MODFLOW input files.

You may save the completed grid using the *Save* option on the *File* menu of the *MODFLOW Extract* dialog box.

Leakage from MODFLOW

The Leakage from MODFLOW option allows the creation of a Hybrid GFLOW - MODFLOW model whereby leakage rates from an existing MODFLOW model are applied at the bottom of the GFLOW model. The idea is to conceptually replace the upper MODFLOW layer or layer(s) that are in direct contact with surface water features (streams, lakes, etc.) by a single layer GFLOW model. The leakages from the underlying MODFLOW model layers are then applied to the bottom of the GFLOW model by defining a MODFLOW grid of leakage elements and importing an array of leakage values from the MODFLOW model. Also imported are an array of recharge values that are defined at the top of the upper MODFLOW layer and will now be applied to the top of the GFLOW aquifer. To ensure that the recharge in the GFLOW model is fully compatible with the recharge of the original MODFLOW model no other recharge should be defined in GFLOW by use of its Inhomogeneity domains. Details on the use of this feature are found in *Help>PDF Help Files>Hybrid GFLOW - MODFLOW model.pdf.*

To create a leakage grid in GFLOW for the Hybrid GFLOW - MODFLOW feature click on *Grid* on the menu bar and then click on *Leakage from MODFLOW>Show Grid Dialog*. There are two tabs on the *Leakage from MODFLOW* dialog:
General
Tracking

General Tab

First go to the *File* menu on the *Leakage from MODFLOW* dialog and click on *Open Grid*. If you created a *basename.del* file that defines the grid it will show in the *Open MODFLOW Grid* dialog window. The "basename" is the *Base Filename (DOS)* defined on the *Project>Project Settings* dialog. The proper format for the *basename.del* file is illustrated below:

```
7      <number of rows>
5714   <height of row 1>
5714   <height of row 2>
5714
5714
5714
5714
5714
8     <number of columns>
5000  <width of column 1>
5000  <width of column 2>
5000
```

5000
5000
5000
5000
5000
2203883 <lower left x-world coordinate of grid>
15584505 <lower left y-world coordinate of grid>

The last two lines in the example *basename.del* file are optional, but when provided must be in the same coordinate system as used in the GFLOW model. If these lines are not present the grid must be georeferenced on the *General* tab by typing in the coordinates in the two data boxes and then click *Update* or by clicking select and the placing the + sign at the location where the lower-left corner of the grid must be and *left-click*. The latter procedure is OK for testing or demonstrations, but will not lead to an accurate correspondence between the grid locations in the MODFLOW and GFLOW models, of course.
When the *basename.del* file is being read by the GUI the leakage file *basename.vlk* will also be read. This is an ASCII file with rows of (real) numbers corresponding to the rows of MODFLOW cells representing the leakage values in these cells. Also present must be the recharge file *basename.rta* with cell-by-cell recharge values at the top of the aquifer. This file will not be read by the GUI, but will later be read by the Solver when creating a solution.

Tracking Tab

The *Tracking* tab allows the inspection of data related to the imported MODFLOW grid. In addition to the world coordinates of the position of the cursor its grid coordinates (relative to the lower-left corner of the grid) are also given. The cell-by-cell leakage values are given at the bottom of the table on the *Tracking* tab. Note that the recharge rate is not given. The recharge values are imported only into the Solver and will be available on the *Model Results* data box after a solution has been obtained. To see these data, place the cursor at the point in the model domain where you seek information and perform a *Shift Left-Click*.

Tools Menu

GUI Options-
Import
Export
GFLOW Database Viewer
Instant Inspector Settings..
Search
Solver Integrity Check-
PEST

The *Tools* menu provides access to GUI options, a utility program to convert various map files to base map (.bbm) files, import functions for text labels, wells and test points, export functions to DXF, SURFER format files and GFLOW input files, a viewer for the GFLOW database, the base map browser, and settings for the instant inspector feature. The *Solver Integrity Check* offers a check on the computational integrity of the solver, but this is not necessary as a routine procedure.
Finally, the *PEST* option brings up a control panel from which to setup and run parameter estimation runs.

GUI Options

This menu allows the user to change default GUI settings regarding the line *type* and

color of *contour* lines and *Path lines.*
Note: The line type option is only available for the smallest line width (1) due to a limitation of the graphics library used in VB6. For line width values larger than 1 the line type will default to a solid line.

The user may add a shadow to line-sinks. This may make line-sinks more easily visible when plotted on a raster graphics base map. The proper setting for the *Linesink Shadow Width* (number larger than 0) is depends on the base map and is best found through trial and error.

The user can also set the *Base Map Haze Level* for the *base map hazing* feature (see *View* menu). The proper setting depeds on the base map used and is best found though trial and error.

The option to *Suppress Coordinate Translation* is useful when performing conceptual modeling in a local coordinate system. When checked, the GUI and Solver use the exact same origin, hence the Solver reports the same coordinates as the GUI. Note: For regular modesl with base maps in UTM or State Plane coordinates this option leads to inaccurate spatial calculations in the Solver, because small distances in the field are obtained by subtracting very large numbers in the world coordinate system. This is why GFLOW uses a local coordinate system in the Solver with its origin at the lower left corner of the window in the GUI at the time of clicking the calculator icon or selecting *Solve* from the *Model* menu.

Import

To import a file containing **text labels** select *Import..* from the *Tools* menu and then *Text Label File...* The file must be an ASCII text file (filename.TX) in which each line has the following format:

x, y, h, label

where **x** and **y** are the (basemap) coordinates of the lower left corner of the text, where **h**=1 for hydrography labels (water levels in surface waters) and **h**=0 for any other text label. The **label** is the water level or text to be printed on the map. The text labels will be added to the current project database file.

To import a file containing **discharge specified wells** select *Import..* from the *Tools* menu and then *Discharge Specified Wells FileÖ* The file must be an ASCII text file (filename.WL) in which each line has the format:

x, y, Q, r, label, top, bottom

where **x** and **y** are the (base map) coordinates of the well, **Q** its pumping rate, **r** its radius, **label** a well identifier (combination of numbers and letters), **top** the elevation of the top of the well screen, and **bottom** the elevataion of the bottom of the well screen. These last two items (top and bottom) are optional. If not present the screen will be set from the local aquifer bottom to the aquifer top. Make sure that the *label* does *not contain spaces or commas* or unpredictable results will occur upon import of the file. Note: The screen top and bottom elevations are only relevant if a 3D well function is used (default is 2D) or if the wells are exported as part of the MODFLOW extract feature. In the latter case the screen setting will determine in which MODFLOW layer or layers the well will occur.
The wells will be added to the current project database file.

To import a file containing **head specified wells** select *Import..* from the *Tools* menu and then *Head Specified Wells FileÖ* The file must be an ASCII text file (filename.WL) in which each line has the format:

x, y, h, r, label, top, bottom

where **x** and **y** are the (base map) coordinates of the well, **h** is the specified head in the well bore, **r** its radius, **label** a well identifier (combination of numbers and letters), **top** the elevation of the top of the well screen, and **bottom** the elevataion of the bottom of the well screen. These last two items (top and bottom) are optional. If not present the screen will be set from the local aquifer bottom to the aquifer top. Make sure that the *label* does *not contain spaces or commas* or unpredictable results will occur upon import of the file. Note: The screen top and bottom elevations are only relevant if a 3D well function is used (default is 2D) or if the wells are exported as part of the MODFLOW extract feature. In the latter case the screen setting will determine in which MODFLOW layer or layers the well will occur.
The wells will be added to the current project database file.

To import a file containing **test points** select *Import..* from the *Tools* menu and then *Test Point FileÖ* The file must be an ASCII text file (filename.TP) in which each line has the format:

x, y, h, Q, type, label

where **x** and **y** are the (base map) coordinates of the test point, **h** the head, **Q** the stream flow, **type** is *piezometer, gage* or *Lake stage*, and **label** a test point identifier (combination of numbers and letters). Labels should not contain spaces or commas, because they are delimeters in this and other files. The test points will be added to the current project database file. A brief example file follows:

```
4.521700E+05 ,4.280888E+06 ,4.02E+02 ,0.0E+00 ,Piezometer , MW47
4.516220E+05 ,4.280488E+06 ,4.01E+02 ,0.0E+00 ,Piezometer , MW48
4.522530E+05 ,4.280245E+06 ,4.01E+02 ,0.0E+00 ,Piezometer , MW49
4.525650E+05 ,4.280350E+06 ,4.01E+02 ,0.0E+00 ,Piezometer , MW50
4.523140E+05 ,4.280059E+06 ,4.02E+02 ,0.0E+00 ,Piezometer , MW51
4.532790E+05 ,4.280306E+06 ,4.00E+02 ,0.0E+00 ,Lakestage , lake_1
4.466018E+05 ,4.271959E+06 ,0.00E+00 ,5.0000E+04 ,Gage , city_ditch
```

To import a GFLOW input file created for the previous DOS version of GFLOW select *Tools>Import>GFLOW for DOS fileÖ* The **filename.dat** file, however, must have been created by the GAEP program or by the DATA command in the DOS version of GFLOW in order to ensure error free data transfer.

Note 1: Before importing a .DAT file created by the DOS version of GFLOW you must set up a new project. To do so, close any current project, click on *New Database*, specify a new project filename and click *Open*. On the next dialog box enter a project description and select the proper units. This is an **important** step! If the basemaps you will be using are in UTM coordinates, select *meters* for *Distance units in Basemap Files.* Hence, it is not important if the units in the GFLOW .dat file to be imported are feet or meters; the *Distance units in Basemap Files* depend only on the units associated with the basemaps that you import! Next, make sure that the *Units for Computations* are set to the units used in the .DAT file you will import. Click *OK* and add the appropriate basemaps to the project file (next dialog box). After the basemaps are displayed on the screen you are ready to go to the *Tools* menu to import the .DAT file from the old GFLOW for DOS version.

Note 2: GFLOW does not support 2-D or 3-D sink discs. Consequently, these features in the old .dat file will be ignored during the import procedure. However, you will get a warning for each sink disc it finds (and skips). You are responsible for replacing these features by line-sink strings when they represented lakes and by recharge inhomogeneities when they represent recharge elements. Refer to your original model (graphics) printout and data to make these modifications in GFLOW after the .dat file has been imported.

To import a previously created **solution** select *Tools>Import>Previous Solution FileÖ* Select the **filename.sol** file that belongs to the current project file. This means that the filename must be the same as the *Base Filename* specified on the *Project Settings..* dialog (see *Project>Project Settings*). This action is often useful when opening an existing project file for which a solution has been generated previously.

Important: To be sure that you import the proper solution it is important that you select a unique *Base Filename* on the *Project Settings*.. dialog for each project file that you create (or duplicate). The GUI will create such a unique name by default, but it is non-descriptive.

To import a **HUC file** select *Tools>Import>Hydrologic Unit.* HUC files have the extension .huc and contain a list of nearfield HUCs, farfield HUCs and Inhomogeneities that form a model for a sub-domain of a larger model area. Refer to "Managing HUCs in GFLOW" stored as *Managing HUCs.pdf* in the document folder. You can access this document from the *Help* menu.

To import a **Line-sink string file** select *Tools>Import>Line-sink Strings.* Line-sink string files have the extension .lss.xml and contain line-sink string data to be imported in an existing project file. These data files may have been created by using the *Tools>Export>Line-sink String* option or by any other code designed to produce line-sink strings for import in GFLOW. The format of the *.lss.xml files is documented in the "import-export.lss.xml.pdf" file in the "import-export" subfolder of the folder "GFLOW projects" on the C-drive. It may also be accessed via *Help>PDF Help Files.* This file is an annotated version of "import-export.lss.xml" that is produced by executing a *Tools>Export>Line-sink Strings* command from the project "sampling.gfl."

To import an **Inhomogeneities file** select *Tools>Import>Inhomogeneities.* Inhomogeneities files have the extension .inh.xml and contain inhomogeneities data to be imported in an existing project file. These data files may have been created by using the *Tools>Export>Inhomogeneities* option or by any other code designed to produce Inhomogeneities for import in GFLOW. The format of the *.inh.xml files is documented in the "import-export.inh.xml.pdf" file in the "import-export" subfolder of the folder "GFLOW projects" on the C-drive. It may also be accessed via *Help>PDF Help Files.* This file is an annotated version of "import-export.inh.xml" that is produced by executing a *Tools>Export>Inhomogeneities* command from the project "sampling.gfl."

To import a **Horizontal Barriers file** select *Tools>Import>Horizontal Barriers.* Horizontal Barriers files have the extension .brr.xml and contain Horizontal Barriers data to be imported in an existing project file. These data files may have been created by using the *Tools>Export>Horizontal Barriers* option or by any other code designed to produce Horizontal Barriers for import in GFLOW. The format of the *.brr.xml files is documented in the "import-export.brr.xml.pdf" file in the "import-export" subfolder of the folder "GFLOW projects" on the C-drive. It may also be accessed via *Help>PDF Help Files.* This file is an annotated version of "import-export.brr.xml" that is produced by executing a *Tools>Export>Horizontal Barrier* command from the project "sampling.gfl."

Export

To export model data to DXF files point to *Export* on the *Tools* menu and click on *DXF...* Select *Contour lines, Particle paths, Wellhead Protection Areas* or *Elements* to be exported. For DXF export, a layer name must be provided; either use the default name or type in the name you wish to use (e.g. 2YR-TOT for 2-year time-of-travel path lines). Once written, the DXF file may be imported into Autocadô or any other DXF-compatible program. The coordinates in the DXF file will be the same as the base map coordinates in your model

To export model data to BLN files (for SURFER) point to *Export* on the *Tools* menu and click on *SURFERBLN..*. Select *Contour lines, Particle paths, Wellhead Protection Areas* or *Elements* to be exported. The resulting files may be used as *XYLine* entries in the *SURFER* contouring program or any other program that uses BLN files. The coordinates in the BLN file will be the same as the base map coordinates in your model.

To export gridded data to .GRD files (for SURFER) point to *Export* and click on *SURFERGrid*... Select *Head, Velocity in X direction, Velocity in Y direction*, or *Total Discharge* as data for the grid. The output file will be a *SURFER*-compatible ASCII GRD file, appropriate for importation into many other contouring programs. The coordinates in the GRD file will be the same as the base map coordinates in your model. **Warning**: The solver also produces GRD files, which are in local model coordinates used by the solver. These files have the *Base Filename* as set on the *Project Settings* dialog box (see File menu) and the extension GRD. These files are to be used by the GUI only, specify different filenames (or use a different folder) for the GRD files to be used in SURFER.

To export model data to Shape files point to *Export* and click on *Shapefile*. Select the entity to be exported (e.g. contours) and specify a filename on the file dialog menu.

To export Hydrologic Unit domains first select a HUC domain boundary, next point to *Tools>Export* and click on *Hydrologic Unit*. Specify a filename and save. HUC domains are exported for future modeling use, see *Managing HUCs.pdf* in the document folder.

To export an Inhomogeneity domain first select the inhomogeneity, next point to *Tools>Export* and click on *Inhomogeneity*. Specify a filename and save. Inhomogeneity domains are exported for future modeling use, see *Managing HUCs.pdf* in the document folder.

To export a file with all Discharge Specified Wells that can be imported in another GFLOW model point to *Export* and click on *Discharge Specified Wells...* The coordinates, radii, pumping rates, labels, and screen settings of all discharge specified wells in the model will be written to a FILENAME.WL file in a format as explained under the *Import* option.

To export a file with all Head Specified Wells that can be imported in another GFLOW model point to *Export* and click on *Head Specified Wells...* The coordinates, radii, specified heads, labels, and screen settings of all head specified wells in the model will be written to a FILENAME.WL file in a format as explained under the *Import* option.

To export a file with Test Point data that can be imported in another GFLOW model point to *Export* and click on *Test Points...* The coordinates, heads, stream flows, and labels of all *Test Points* in the model will be written to a FILENAME.TP file in a format as explained under the *Import* option.

To export a file with Hydrography Labels that can be imported in another GFLOW model point to *Export* and click on *Labels.*. The coordinates, a hydrography label code (=1) and the head will be written to a FILENAME.TX file in a format as explained under the *Import* option.

To export a Line-sink string file select *Tools>Export>Line-sink Strings.* Line-sink string files have the extension .lss.xml and contain line-sink string data to be imported in an existing project file. See *"To import a line-sink string file"* under *Tools>Import* for more details.
To export a Inhomogeneities file select *Tools>Export>Inhomogeneities.* Inhomogeneities files have the extension .inh.xml and contain Inhomogeneities data to be imported in an existing project file. See *"To import a Inhomogeneities file"* under *Tools>Import* for more details.
To export a Horizontal Barriers file select *Tools>Export>Horizontal Barriers.* Horizontal Barriers files have the extension .brr.xml and contain Horizontal Barriers data to be imported in an existing project file. See *"To import a Horizontal Barriers file"* under *Tools>Import* for more details.

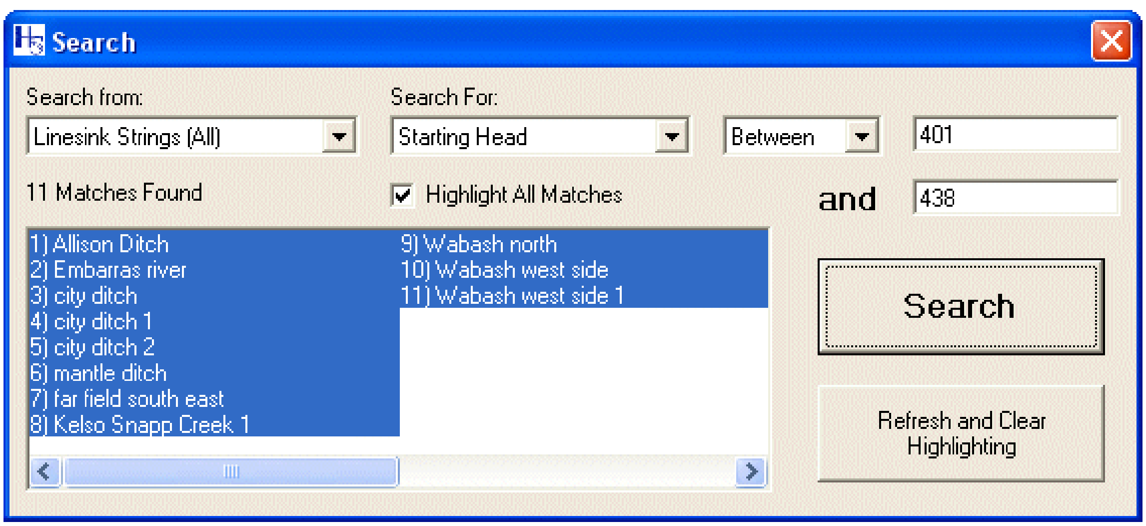
GFLOW Database Viewer

The *GFLOW Database Viewer* on the *Tools* menu provides access to the database used by the GFLOW graphical user interface.
Modifications of database fields are possible after selecting *Options>Allow Modifications.* This feature is for advanced users only!

For routine access to details on the GFLOW model the user is referred to the *GFLOW Results Table* on the *Model* menu.

Search

With the ***Search*** feature you can find elements in the model based on search criteria, which are then highlighted on the graphics screen.



The *Search From* field defines a group of elements from which you want to select elements with a specific property.

The *Search For* field defines the property that forms the search criterion.

The next fields allow you to select particular values for the search criterion.
For instance, in the example shown above, the element group is all linesink strings with the search criterion set to "Starting Head" with values between 401 and 438.

Once you have specified your search parameters as described above, press the "Search" button to let GFLOW find all matches corresponding to your query. The label of each element that is found will be displayed in the bottom left data box, along with the number of matches.

By default the search feature will highlight on screen all elements that match your search. You may uncheck the "*Highlight All Matches*" checkbox to prevent this. You may also click on each match in the list box to highlight or unhighlight elements individually.

The "*Refresh and Clear Highlighting*" button will clear all the elements highlighted by the search feature and refresh the screen. If you run a new search, the highlighting from the previous search will also be cleared.

Note 1: The various *Linesink Strings* fields in the "Search From" box are different from the *Linesinks* field. The *Linesinks* field refers to individual linesinks with properties that are only available when a solution is present. The *Linesink Strings* fields refer to whole linesink strings with properties that are mostly available without a solution present. Therefore *Linesinks* is only available as an option when a solution is present while *Linesink Strings* is always an available option.

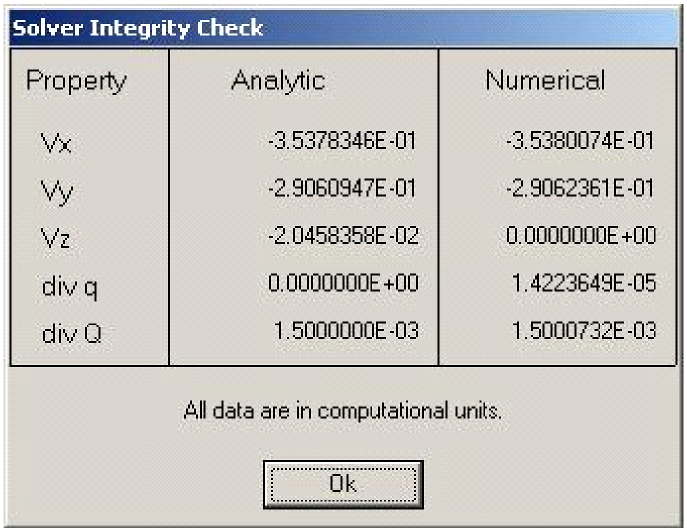
Note 2: When performing a search with a text string in the "Search For" box, for instance, a label or GFLOW label, the search will automatically use wildcard characters to find all strings that include the string you specified. You may also insert your own wildcard characters by adding a ? for a single wildcard character or a * for multiple wildcard characters.

Instant Inspector Settings

The GFLOW graphical user interface offers instant inspection of input and output data at any point in the model domain. To use this feature simply point with the cursor at the desired location, hold down the Shift key and left-click the mouse. Within seconds a dialog box appears with both input and output data pertaining to that point, e.g. hydraulic conductivity, bottom elevation, recharge, head, velocity, etc. The elevation for which these data are provided is defined under the *Instant Inspector SettingsÖ* option on the *Tools* menu. To obtain data at the top of the saturated aquifer thickness, select the box "*Inspect at top of saturated thickness*". This would be the water table in an unconfined aquifer and the aquifer top in a confined aquifer. To select a different elevation, deselect that box and specify the elevation in the input box labeled "*Inspection Elevation*".
The GFLOW graphical user interface also supports an instant path line trace option. Instead of defining a new particle and recreating a contour plot and retracing all existing path lines, you may point at the desired starting point of a new trace, hold down Ctrl-Shift and left-click the mouse. Within seconds a trace appears. The starting elevation of the trace is set via the *Instant Inspector Settings..* option on the *Tools* menu and the direction of the trace is set under that same option. The default is forward in time, but by checking the box in front of *Trace Backward in time* on the *Instant Inspector Settings..* dialog box, the trace occurs backward in time.

Solver Integrity Check

Check this option to bring the *Instant Inspector* feature in the "solver integrity checking mode". Bring the cursor to the desired location on the base map and do a shift-click. In addition to the normal *Model Results* data panel with computational results, a second panel is displayed as shown below.



Property	Analytic	Numerical
Vx	-3.5378346E-01	-3.5380074E-01
Vy	-2.9060947E-01	-2.9062361E-01
Vz	-2.0458358E-02	0.0000000E+00
div q	0.0000000E+00	1.4223649E-05
div Q	1.5000000E-03	1.5000732E-03

The value of the average groundwater velocity vector components as produced by the analytic functions in GFLOW are compared to a finite difference approximation. The spatial step size for these numerical calculations are set on the *Tools>Instant Inspector Settings* dialog. The default value is one unit, which may be 1 foot	The divergence of the specific discharge vector in a Dupuit-Forchheimer model

or
1
meter,
depending
on
the
computational
units
set
on
the
Project>Project
Settings
dialog.

The divergence of the specific discharge vector in a Dupuit-Forchheimer model is always zero. The numerical value, calculated from a finite difference approximation of the analytic specific discharge in GFLOW should be very small.
The divergence of the discharge vector (specific discharge integrated over the saturated aquifer thickness) should be equal to the local recharge rate. The analytical value is the local recharge rate and the numerical value, obtained from a finite difference approximation, should be very close. The values shown in the figure above are considered satisfactory.

Uncheck *Tools>Solver Integrity Check* to leave this mode.

PEST

Set up and execute a PEST parameter optimization session.

Make sure that the current GFLOW model has the appropriate symbolic names defined for the parameters that are to be included in the optimization. These symbolic names are entered on the *Scenarios* tab sheets of the *Properties* dialogs for line-sinks (see also note 2 below) and inhomogeneities and the *Model>Settings* dialog.
Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

Also ensure that the proper *Test Points* have been defined to be used as calibration targets. There are three *Test Point* groups: *Piezometers*, *Gages*, and *Lakestages*. Piezometers define observed heads in the model domain. Stream gages define observed stream flows in the nearest line-sink. That line-sink must be part of a stream network, and the Solver option *Conjunctive Surface Water - Groundwater Solution* must be checked (see *Model>Settings>Solver* tab). Lake stages must be placed close to a line-sink that is part of a boundary of a lake (the *Lake* option must be checked on the *Linesink String Properties* dialog). Lake stages define the observed lake stage of a lake feature for which the lake stage is being solved for.
Note: Symbolic names should not be longer than 12 characters and not contain spaces or any of the following characters: , - _

To conduct a parameter optimization session proceed as follows:

If PEST input and output files exist from a previous run make sure to back them up if they need to be referred to again. Next, click *Delete all PEST Related Files*.

Click *Assign Weights* to modify the default weights for the individual observations (Test Points in GFLOW). These weights will be written to the relevant PEST input file.
Click *Write PEST Input Files*. These files are written by the GFLOW GUI based on the *Symbolic* names and *Test Points* defined for the project.
Edit the "basename.pst" file to set up the desired PEST simulation. Note: the file written by the GUI has default settings that may need to be changed. For information on necessary and proper parameter settings in the "basename.pst" file consult the PEST manual by clicking on the *View Pest Manual* button.
Click *Run PESTCHEK* to check for errors in the View window. Correct any errors.
Click *Run PEST*. The PEST executable will be scheduled by the GFLOW GUI and run in a DOS box. Upon completion of the run you will see a number of PEST output files listed in the *PEST Output Files* window. You can double click on any of these files to view them.
When clicking the *Import Optimized Parameters* button, the optimized parameters reported in the **basename.par** file will be read by the GUI and assigned to the *Current values* on the *Model>Implement Scenarios* dialog. From the *Model>Implement Scenario* dialog you can perform a test run, using these optimized parameters, and/or make them the default parameters for the project.

Note 1: GFLOW writes all PEST input files with the "basename" set on the *Project>Project Settings* dialog as the *Base Filename (DOS)* and writes these files in the current working directory. It is recommended that you keep these filenames.

Note 2: Line-sink strings for which symbolic names are used to redefine the resistance during PEST runs must have either the radio button for "*Along surface water boundary*" or for "*Along stream center line*" selected on the *Linesink String Properties* dialog. The option "*Unkown*" should not be used! This will allow the Solver to recalculate the "effective leakage zones" under changing resistance and transmissivity scenarios.

Window

The *Window* menu facilitates the selection of the model area to be displayed in the graphics box.

Each click on *Zoom In* enlarges the map by 30% keeping the center of the map in the center of the graphics box.

Each click on *Zoom Out* reduces the map by 30% keeping the center of the map in the center of the graphics box.

To display the full map in the graphics box click on *Zoom Extents*.

To display a particular section of the map, click on *Zoom to Window*. Move the cursor to the upper left corner of the domain to be plotted, keep the left mouse button depressed and move the cursor to the lower right corner of the domain to be plotted. A box will appear to show the area selected for plotting. When releasing the mouse button the selected area will be plotted in the graphics box.

Click on *Refresh* to redraw the map. This may be desirable, for instance, after editing has left openings in the base map.

Click on *Save Current Window* to store the current window coordinates in the database under a user specified name. You will be prompted for the name when selecting this option.

Click on *Saved Windows* to retrieve a previously stored window. You may browse through the saved windows and their coordinates and select the one you want implemented. You may also change the coordinates of the lower left and upper right hand corner of the window.

Help

There are three groups of links on the Help menu:

The first group contains three links to access the GFLOW Help system (manual):
GFLOW Manual opens the Table of Contents
Modleing with GFLOW offers a quick start for setting up a GFLOW project.
GFLOW Tutorial offers a step-by-step guide to modeling with GFLOW.

PDF Help Files provides access to a collection of monographs aoffering background information on various topics relevant to modeling with GFLOW

The second group contains some background information on GFLOW's history:
GFLOW Release Notes offers some details related to installation and specifics related to this vesrion of GFLOW.
About GFLOW... is the opening splash screen with bersion information and authors of GFLOW.

The last item if the Register GFLOW option, which will be greyed out once the program is registered as a professional version. Note: Once you register the program with a username and unlock key (obtained after purchasing GFLOW online at www.haitjema.com each successive upgrade will remain registered.

Finally, when hoevring the cursor over an icon on the Tool bar a brief explanation of that icon will appear momentarily. When inside a menu, you may press *F1* to obtain context sensitive help. For more detailed explanations, click on *Help* and refer to the table of *Contents* or *Index* after clicking on *GFLOW Manual*..