Basic Concepts

Projects and Files

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

The graphical user interface (GUI) uses various additional files to communicate with the Solver. The GUI generates a unique *Base Filename* (DOS) that will be used for naming the Solver interaction files.
Below follow the basic steps for opening or creating a project file.

Click on the *Project* menu and select *New Database* to create a new project data base or select *Open Database* to open an existing data base: **`filename.gfl`**

When creating a new project file you may select one or more base maps, see below. Once you have opened or created a data base file (project file), GFLOW will display the map or maps and you are ready to setup a groundwater flow model on screen. Any changes to the model parameters are immediately updated in the database and incorporated in the project file. When it is necessary to create a duplicate project file under a different name you should use the *Make Duplicate DatabaseÖ* option on the *Project* menu.

There is one other filename you may specify: the *Base Filename*. You will find this option on the *Project Settings* dialog box. The *Base Filename* has a maximum length of 8 characters and no spaces (DOS filename) and is the name used for the Solver data files. When creating a new project a default unique *Base Filename* is automatically entered, but it is rather cryptic. You may enter a more recognizable, but unique *Base Filename* yourself.

Warning: If you do not use a unique *Base Filename* for each new project, the GUI may mistake the solver output for one project as the solution for another (currently open) project.

NOTE: The GFLOW installer will create a folder on your c:-drive under <user> (my) documents called *GFLOW projects* in which it places three sub-folders: "*example*," "*example2*," and "*validations*.." Note: An exact path cannot be specified here as it is diferent for different Windows versions.
You are encouraged to use this folder *GFLOW projects* to add sub-folders for each of your own projects.

Base Maps

GFLOW supports various raster and vector graphics file formats to serve as base maps for groundwater flow models. Associating base map files with your project is managed on the *Project Settings* dialog under the *Project* menu. When clicking on *Add Map..* a file management dialog appears. Clicking on *All Supported Types* opens a drop down list of the file types supported by GFLOW. Note: all maps must be georeferenced, which means they must contain coordinate information in either UTM or State Plane coordinates. All file types, except for the vector maps contained in BBM files, are rendered by MapObject LT 2.0, a software product by ESRI.

BBM files contain 1:100,000 USGS DLG data in binary format. This binary format has been developed especially for use in the WhAEM and GFLOW groundwater flow models. The GFLOW installation will deliver some these BBM files as part of the example projects (see Tutorial). BBM maps may also be obtained from a USEPA website:

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

You can graphically locate the area for your bbm maps or click on a state and then quad name below the map of the US and state, respectively.
Select to download the map or maps to the download folder on your computer. The file that is being downloaded is a self-extracting zip file with the extension .exe. You will have to overrule the Windows security warnings and select "run anyway" when trying to run the executable.
IMPORTANT: EPA is in the process of converting old 16-bit self-extract files, which fail to run on 64-bit machines to newer 64-bit executables. In case the xxxx.exe fails to run (16-bit version), simply right-click the file and select Winzip or some other unzipping program on your computer to force the extract process.

Note 1: Base map files from different sources may have different coordinate systems and **units**. GFLOW will maintain the *original units* of the base map! You must know these units when you specify the base maps during the creation of a new project. For instance, the units for the **.bbm** files derived from DLG files are METERS. Hence, you must set the "*Distance* u*nits in Basemap Files*" to meters. You may still select feet and days as the units for entering heads, pumping rates, recharge rates, etc. The latter units are called "*Units for Computations*" in GFLOW. You must tell GFLOW (at the time you create a new project) what "*Units for Computations* " you will be using.
There is an option on the *Project Settings Dialog* to change these units, however, this option is only for the case that you made a mistake during the initial phase of creating a project. Resetting the units will not change any data! For instance, if your distance units in the basemap files are in meters, but you mistakenly specified feet, you can correct this setting on the *Project>Project Settings Dialog*; there will be no change in the coordinate data.

Note 2: Large vector or raster graphics maps may be very slow to draw. In particular, when using a large Shapefile, with many attributes selected as labels for display, some delay may occur before the maps are drawn. Use judgment in the selection of basemap files. Overlaying modeling results on very detailed and complex maps, for instance for report production, may best be done by exporting the model information (*Tools>Export*) and importing it in a professional CAD or GIS program.

Hydrography and Water Levels

A map of all surface water features is called a hydrography map, which should be part of your base maps in GFLOW. In order to translate this hydrography map into boundary conditions for the groundwater flow model (average) water levels along streams, creeks, and in lakes must be known.

Streams and lakes will be represented by strings of straight line elements (line-sinks) with each assigned a head that is set equal to the water level in the stream or lake. In preparation for the task of creating these line-sinks you may want to mark up your hydrography map by adding the contour elevation of the terrain surface next to the stream at those locations where such an elevation contour line crosses the stream. As a source for these surface water levels you may use USGS 71/2 minute topographic maps, which are available in paper form or in digital form, see `http://edcwww.cr.usgs.gov/` Digital topographic maps that are georeferenced, may also be used as a base map in GFLOW.

When entering line-sink strings you will be prompted for the starting and ending surface water elevations for the string (in GFLOW referred to as starting head and ending head, respectively), which you can now read directly from the base map on the screen.

See also Creating new Line-sinks under the *Element* menu.

Aquifer Parameters

GFLOW requires the user to specify four aquifer parameters, which is done by clicking *Settings* on the *Model* menu and selecting the *Aquifer* tab.

Base elevation in feet or meters with respect to mean sea level (msl). The aquifer base in the model is a (fictitious) horizontal plane that acts as a no-flow boundary. In reality, the aquifer base may be a clay layer or rock surface, which is usually not a perfect horizontal surface. The base elevation entered on this menu is the regional aquifer base. Locally, you may select a different base elevation using the *Inhomogeneity* option under *New* on the *Element* menu. *Warning:* It is important that the model aquifer base in a domain is not higher than the lowest water level assigned to line-sink strings in that domain. In the event that the aquifer base is above an assigned surface water elevation, the model will limit the heads in the aquifer to the aquifer base in that area, which is often noticeable by ragged potentiometric contour lines.

Thickness of the aquifer is the actual thickness of aquifer material above the background aquifer base (the one entered on this menu), regardless of how much is saturated. In fact, when the aquifer base elevation plus aquifer thickness exceeds the potentiometric head in the aquifer the model automatically assumes unconfined flow conditions. If you want to ensure unconfined flow conditions throughout the model domain, regardless of the groundwater elevation, you may set the aquifer thickness to a very large value, much larger than the actual aquifer thickness.

Hydraulic conductivity, is the regional background value for the model domain. You may locally select a different hydraulic conductivity by use of the *Inhomogeneity* option under *New* on the *Element* menu.

Porosity in GFLOW is an effective porosity that is constant for the model domain. The porosity value is used for tracing path lines and calculating groundwater residence times (travel times).

The *Add Salt Water Interface* option may be checked to model fresh water flow in coastal aquifers with a stationary salt water interface.

Analytic Elements

Analytic elements are mathematical functions (real or complex) that represent hydrogeologic features in a groundwater flow model. For instance, a meandering creek or river is represented by a string of straight-line elements, which approximates the geometry of the stream. In GFLOW each of these line elements is associated with a "line-sink function", which generates the discharge potential and stream function for a sink distribution along a straight line (with constant sink density). The discharge potential and stream function are the real and imaginary part of a complex potential function generated by the line-sink function.

The discharge potential Φ is defined in terms of the potentiometric head, the hydraulic conductivity, and the saturated aquifer thickness in such a way that the discharge vector is the (negative) derivative of the discharge potential. Under unconfined flow conditions the discharge potential is defined as:

$$\Phi = \tfrac{1}{2}k\phi^2 \qquad (\phi \le H)$$

Under confined flow conditions the discharge potential is defined as:

$$\Phi = kH\phi - \tfrac{1}{2}kH^2 \qquad (\phi \ge H)$$

where k is the hydraulic conductivity, ϕ is the potentiometric head measured with respect to the aquifer base, and where H is the aquifer thickness as defined in the model. The governing differential equation in terms of Φ is linear and is the same for both confined and unconfined flow. This makes superposition (in terms of the discharge potential) possible under both flow conditions. The use of these discharge potentials is transparent to the user; they are handled internally in the solver. The user may specify surface water elevations with respect to mean sea level at streams and lakes and will get potentiometric head contours with respect to mean sea level as output from the groundwater flow model.

Other examples of analytic elements are wells (fully or partially penetrating), line-doublets or "double layers" to represent inhomogeneities or horizontal flow barriers in GFLOW and areal recharge due to precipitation (a recharge inhomogeneity).

The contributions of all analytic elements to the discharge potential (in GFLOW perhaps more than several hundred) are added together (superimposed) to create a comprehensive solution to the groundwater flow problem. The discharge potential is directly related to the potentiometric head in the aquifer (water table in an unconfined aquifer). The derivative of the discharge potential with respect to the two (x and y) coordinate directions yields the discharge vector, which is the total groundwater flow rate (integrated over the aquifer height) per unit width of the aquifer. The discharge vector is calculated in GFLOW by superimposing so-called derivative functions, which are the derivatives of the potential functions. The groundwater flow velocity in GFLOW is calculated by dividing the discharge vector by the saturated aquifer thickness and the (effective) aquifer porosity.

Analytic elements are defined in GFLOW by selecting *New* under the *Element* menu.

Wells

There are three different wells in GFLOW: steady state wells, transient wells, and partially penetrating wells. The first two are fully penetrating and represented by two-dimensional functions (x and y only). The partially penetrating well is a fully three-dimensional analytic function, but can only be included in a confined flow domain. The transient well is a well that starts pumping at a certain time. Its transient response is calculated by use of the Theis solution, see Haitjema (1995).

The most commonly used well in GFLOW is a fully penetrating steady state well that is entered on the *General* tab of the *Well Properties* dialog box. This well in GFLOW is represented by Thiem's equation, which generates the discharge potential for a continuously pumping well under steady state groundwater flow conditions. Thiem's equation in terms of the complex potential λ is:

$\tau = (SL^2)/(4TP)$

where Q is the pumping rate of the well,

$z = x + iy$

is the complex coordinate of the point at which the potential due to the well is calculated,

$z_w = x_w + iy_w$

is the complex coordinate of the location of the well, and *R* is some arbitrary constant with the dimension of length.

In GFLOW a steady state well is assigned a location, pumping rate, radius, and label. When path lines are to be traced from the well (particle tracking), the number of particles to be used is also specified. The particle tracking from the well is performed backwards in time, starting at the well and ending at a specified groundwater residence time from the well. The particle tracking parameters are defined on the *Other* tab of the *Well Properties* dialog.

Wells are defined in GFLOW by selecting *Wells* under *New* from the *Element* menu.

Line-sinks

Line-sinks are used to represent streams (creeks, ditches, and rivers) and lake boundaries. A line-sink in GFLOW has a constant sink density (constant groundwater extraction rate along the line element). The discharge potential for a line-sink is obtained by integrating a point sink (Thiem's equation) along a (straight) line element, see Strack (1989) and Haitjema (1995). The sink density of a line-sink that represents a stream section or a section of a lake boundary is in general not known. In GFLOW this is handled by specifying a potentiometric head at the center of the line-sink, which is selected equal to the average water level in the stream section or lake that is represented by the line-sink. For every unknown sink density there exists one known head. This leads to a system of equations that is solved for the unknown sink densities of the line-sinks. Every change in data in the GFLOW model, therefore, requires a new solution procedure to recalculate the sink densities. The sink density of the line-sink represents the groundwater infiltration rate into the stream section that is represented by the line-sink, in volume per time per unit length of the line-sink (stream or lake boundary).
The use of line-sinks along lake boundaries implies that a lake receives or loses groundwater along its boundary only. This appears to be a good approximation of reality when the size of the lake is large compared to the saturated thickness of the aquifer. For streams that have a width that is large compared to the aquifer thickness, the realism of the model may be improved by positioning line-sinks along both sides of the river.

The number of line-sinks required to represent a stream depends on two factors: (1) the need to follow detailed stream geometry, and (2) the need to allow for variations in the groundwater discharge or recharge along the stream or lake boundary. Representing stream geometry is more important in the near field (area of interest) than in the far field. Similarly, the need to use many small line-sinks to allow discharge or recharge variations along the stream is most pressing in the near field, particularly near a high capacity well, which may draw water from the stream and tends to cause significant variations of stream inflow or outflow opposite the well. Consequently, surface water features in the area of interest (near field) should be represented with more and smaller line-sinks than surface waters in the far field, remote from the area of interest. See also near field and far field.

Line-sinks in GFLOW may be given a resistance (in days), which represents the resistance to flow into or out of the surface water body due to a silt layer on the bottom or due to a low permeable layer between the surface water and the aquifer. To implement this resistance properly, a *width* and a *depth* parameter must be specified in addition to the *resistance* parameter. Detailed information on how to define these parameters is provided as context sensitive help on the *Linesink Properties* dialog and in the document "*Dealing with Resistance to Flow into Surface Waters.pdf*" in the documents folder (and accessible from the *Help* menu) or on the website `www.haitjema.com`.

Line-sink strings that represent ***stream networks*** can support stream flow routing, which improves the realism of the groundwater flow solution by limiting the infiltration rate of loosing streams to the available stream flow. In fact, head waters of streams that do not receive groundwater cannot infiltrate water into the aquifer either and will be removed from the groundwater solution. Line-sink strings can be made part of a stream network by specifying the proper parameters on the *Routing* tab of the *Linesink String Properties* dialog. To implement these stream networks in GFLOW the option *Conjunctive Surface water - Groundwater Solution* must be checked on the *Solver* tab of the *Model Settings* dialog.

Line-sinks are defined in GFLOW by selecting *Linesinks* under *New* from the *Element* menu.

Inhomogeneities

Inhomogeneities are closed domains inside of which the hydraulic conductivity, aquifer base, recharge rate and porosity can be redefined. These domains are defined by polygons of line-doublets (also called "double layers"), which may be nested or abut each other, but may not overlap each other. An exception is an inhomogeneity domain inside of which only the recharge is redefined. That domain may be placed everywhere and overlap with other inhomogeneity domains.

Inhomogeneities may be used to incorporate aquifer heterogeneity in zones. Typical examples are the more permeable channel deposits surrounding many streams. These channel deposits may also have a different (often lower) aquifer base and recharge rate due to precipitation. To add recharge, enter it on the inhomogeneity dialog box.

Areal recharge due to precipitation in the model area may be specified by entering an inhomogeneity with only the "Change recharge rate from default" box checked.

Note: Inhomogeneities that do not redefine the hydraulic conductivity or the aquifer base can be defined by a few large line elements (for instance a rectangle with four line elements).

These line-doublets do not add equations to be solved for.
In contrast, inhomogeneity domains that redefine the aquifer base and/or hydraulic conductivity should be defined with sufficient line elements to allow the conditions along the domain boundary to be adjusted to local flow conditions. Like for the case of line sinks, small line-doublets are needed in areas with rapidly varying flow (heads). Each line-doublet adds two equations to the system of equations to be solved. Note: Line-doublets along common domain boundaries occur only once in the system of equations.

Horizontal Barriers

Horizontal barriers are slurry walls or rock outcrops that block or restrict groundwater flow in the horizontal direction. In GFLOW horizontal barriers may be defined that are open or closed strings of (straight) line elements (line-doublets). The barrier is given a width, depth, hydraulic conductivity and porosity. If the barrier does not reach down to the aquifer base, most or all groundwater flow will occur underneath the barrier. A horizontal barrier with a conductivity of zero defines a no-flow boundary (provided the barrier depth is at or below the aquifer base). Open barriers may be used to represent part of a rock outcrop boundary in the near field or to represent a slurry wall. Closed barriers may be used to define local rock outcrops in the near field, model the inside of a closed flow domain (aquifer basin), or model flow both in and outside a closed leaky slurry wall.
Warning: In order to obtain a stable and accurate solution for barriers with a low resistance (high conductance) a rather large amount of line-doublets are needed. Most partially penetrating barriers have a high conductance.

Horizontal barriers are defined in GFLOW by selecting *Horizontal Barrier* under *New* from the *Element* menu.

Uniform Flow

The Uniform flow option on the *Element* menu allows for conceptual modeling of regional flow without the introduction of areal recharge and surface water features in the model. It is provided for educational purposes. For instance, the modeler may compare capture zones for the well or wellfield obtained from a complex model, with capture zones that would result from approximating the regional flow field by uniform flow in the area of the wellfield. In general, the uniform flow option should not be used in combination with other analytic elements, except with the wells for which capture zones are to be generated.

In summary, the uniform flow option is not intended for routine groundwater flow modeling with GFLOW.

Near Field and Far Field

When defining analytic elements it is important to distinguish between the near field and the far field of the model area. The near field is the area of interest; e.g the area near a well field or containing a plume of contaminated groundwater. The far field is the surrounding area that contains hydrologic features that control the groundwater flow toward or out of the near field. The far field area is only included in the model in order to properly define hydrologic "boundaries" for the near field. Streams and lakes in the far field are represented rather crudely with large line-sinks. Closer to the near field, more care is taken in representing stream and lake-geometry. Inside the near field, close to a well field, for instance, the highest resolution of line-sinks is used. This selective use of line-sink resolution makes it possible to include a large area in the model while at the same time preserving computational efficiency. A computationally efficient model invites experimentation with different assumptions, which makes for a better modeling exercise, hence a better understanding of the groundwater flow.

An example of the use of different line-sink resolutions in the far field and near field is found in the project file **`example.gfl`**. The infiltration or extraction rates of line-sinks in the near field are representative for the actual groundwater discharge or recharge of the stream sections in the field.
For instance, when comparing the line-sink strength of stream sections close to a well field, the groundwater discharge into those sections will be reduced when increasing the pumping rate of the well. Some line-sinks may actually switch from positive strength to negative strength; hence from groundwater discharge to groundwater recharge, in order to supply water to the well field.
In contrast, the line-sink strengths in the far field are devoid of any hydrologic meaning. These line-sinks extract or infiltrate water as needed to maintain specified potentiometric heads in the far field, which is why they were entered in the model. In the absence of more remote line-sinks (hydrologic features), outside the far field, these far field line-sinks strengths are not representative for the actual discharge or recharge of the streams or lakes they represent.

Line-sinks in the far field should **not** be given a *width*, *depth* or *resistance*, nor should they be part of a stream network (no surface water routing). Only line-sinks with zero properties (except for the starting and ending head) will maintain the desired far field conditions. To ensure proper use of line-sinks in the far field you should select the box "Use as Far-Field" on the *Linesink String Properties* dialog.

Model Runs

A groundwater flow solution can be generated after all aquifer parameters and analytic elements have been defined. The analytic element solver is started by clicking *Solve* on the *Model* menu, clicking on the calculator icon or pressing *F9*. The solver generates the sink densities for the line-sinks and strength parameters for the line-doublets (inhomogeneities and horizontal barriers). A solution may require several iterations, depending on the complexity of the model. Most problems converge in 4 to 5 iterations. However, when stream flow routing is active (conjunctive surface water and groundwater flow solutions) many more iterations may be necessary (say 15 or more). A report on the accuracy of the solution scrolls inside the solver box on screen and may be reviewed after the solution process by inspecting the *Runtime Message File* (*Model>View Model Run Files..*).
The user can define so-called convergence criteria for the various types of analytic elements to allow the solver to abort the iteration process if all criteria are being met. This can be useful for series of batch runs as occur during parameter optimalization with PEST (integrated in GFLOW). These criteria are defined in the file *convergence.tab*, which can be edited on the *Model>Settings>Solver* tab.

Depending on the *Settings* on the *Model* menu, heads or stream flow at test points (Piezometers and Gages, respectively), potentiometric contours, and path lines will be generated as well. There are three types of test points: *piezometers* where the head is known, *gages* where the stream flow is known, and *lake stages* to be used inside lakes (defined by line-sinks) for which the lake stage is being calculated. *Piezometers* may be wells in which a static water level has been measured (no pumping) or an open gravel pit with a known water elevation. *Gages* are locations where the stream flow is known, for instance because they coincide with official USGS gaging stations at streams. *Lake stages* are like piezometers, but they compare the expected or measured lake stage to the one calculated by GFLOW. This type of *Test Point* should only be used near lakes for which the lake stage is being solved for

(*Lake* option on the *Linesink String Properties* dialog).*Test points* defined in the model are used for model calibration purposes, comparing the head and stream flow calculated by the model with those observed in the field. Test points and associated observed heads or stream flows can be entered by the *Add Test Point* command on the *Model* menu. Gaging station data will only be used when conjunctive surface water and groundwater flow solutions are created. It is recommended to start generating groundwater flow solutions only with piezometer test points and contours, but without path lines. Once aquifer parameters and analytic elements have been adjusted to obtain a satisfactory groundwater flow solution, conjunctive surface water and groundwater solutions may be created and gaging stations and path line traces may be added for further analysis.

Note 1: After every change in aquifer data or analytic element data, as well as after making changes in *Settings* on the *Model* menu, a new groundwater flow solution must be generated (clicking on the calculator icon).

Note 2: After opening a project file for which a solution has been generated earlier, you may import that solution from *Tools>Import>Previous Solution File..* Select the **basefilename.sol** file with the **basefilename** specified on the *Project Settings* dialog box (*Project>Project Settings*).

Contour Plots

Contours of equal potentiometric head (equipotentials) are plotted on top of the background map after a groundwater solution is created, provided that: (1) the *Contouring* option is selected on the *View* menu and (2) the *Compute Contours* box is checked and contouring parameters are set under the *Contouring* tab of the *Settings* option on the *Model* menu. The latter action ensures that the solver returns the necessary contour data, while the *Contouring* option on the *View* menu controls whether the contours will actually be displayed. Any changes in the contouring parameters (*Model>Settings>Contouring*) require a recalculation of a grid filled with values to be contoured.
When zooming out or selecting a different window, contours will only appear in the original window domain for which they were generated. If it is necessary to display contours in the new window a new grid must be calculated. Simply click on the calculator icon and click *yes* for using the existing groundwater flow solution.
Note: In most cases you will want potentiometric contours, which requires the radio button for *Heads* to be checked on the *Contouring* tab of the *Model>Settings* dialog .

Path Line Tracing

There are three path line tracing options in GFLOW, (1) multiple path lines toward a well, (2) single path lines from selected points in the model domain and (3) instant single path line traces from the cursor position. All path lines in GFLOW are traced in a three-dimensional space.

(1) Multiple path line traces from wells are realized by distributing a number of (fictitious) particles around the well perimeter and tracing them backward in time (moving away from a pumping well).
A bundle of these path lines will fill the capture zone of the well. By adding markers (circles) on the path lines for equal time intervals, for instance one year, isochrones of groundwater travel times emerge when connecting markers of equal time. The number of particles for a well is specified as a property of the well (on the *Other* tab of the *Well Properties* dialog), while the

program automatically distributes these particles around the well bore. The starting elevation of the particles, to be set on the *Other* tab, influences the extent of the traces. Particles near the aquifer bottom will often extend the furthest, while particles traced from an elevation close to the aquifer top may stop after a short distance (at the point where the water enters the aquifer). See also Time of travel capture zones.

(2) Single path line traces from specified points (other than those at wells) can be created both forward or backward in time. These particles are created by the *Add Particle* option on the *Model* menu or by the associated icon on the tool bar.

(3) An instant path line trace from the cursor position is obtained by holding down the *Ctrl* and *Shift* key and left-clicking the mouse. Note: On the Mac running a virtual machine with Windows (e.g. Parallels) this key stroke combination may be different, e.g. *Ctrl-Shift-Command*. The elevation of the starting point and the direction of the trace (forward or backward in time) can be set under *Tools>Instant Inspector Settings*. To set the starting elevation of the instant trace type it in the box labeled *Inspection Elevation.*

Path line traces and an associated maximum travel time are selected prior to solving, see *Tracing* tab under *Settings* on the *Model* menu.

Two different markers may be added to the path lines, *time* markers and *elevation* markers. The intervals for these markers may be selected under the *Pathlines* option on the *View* menu.

Particle traces in GFLOW may or may not stop at a line-sink, depending on circumstances. When the hydraulic gradient on both sides of a line-sink is away from the line-sink (loosing stream) particle traces that occur backward in time from the well will end at that line-sink. However, a stream may be losing water while the hydraulic gradient on the side of the stream (line-sink) opposite from the well may still be toward the line-sink. In that case the well receives water from both the stream and the area across the stream. The particle traces, for that case, will either end at the stream or extend beyond the stream, depending on the elevation of the particle in the aquifer.

Time-of-Travel Capture Zones

Time of travel capture zones are obtained by use of multiple path line traces toward wells with time of travel tic-marks, see *Path Lines* on *View* menu. These path lines are defined when creating wells, while tracing is activated by selecting *Compute Particle Paths* on the *Tracing* tab on the *Model* menu. When defining single time of travel capture zones, for instance 5-year time-of-travel (TOT) capture zones, no time tic-marks are needed. Instead the *Maximum Travel Time* on the *Tracing* tab on the *Model* menu may be set to 1825 days (5 years). This will result in a bundle of path lines, which will occupy the 5-year time of travel zone.

Note: The actual envellope of the capture zone is often not included, but is being approached when increasing the number of particle traces for the well. Hence, for a final assessment of the TOT capture zone it is advisable to set the number of path lines from the well quite high, e.g. 50.

Instant Inspector

Shift left-click (instant data display)

Input and output data at any location in the model domain can be accessed by use of the

"instant inspector" feature. Move the cursor to the desired location on the base map, hold down the *Shift* key and left-click the mouse. A box will appear with the following data: coordinates (x,y,z), porosity, hydraulic conductivity, aquifer bottom elevation, net recharge rate, head, average groundwater flow velocity (Vx,Vy,Vz). The elevation of the point at which this data is obtained is entered in the box *Inspection Elevation* under *Tools>Instant Inspector Settings*.
Solver Integrity Check: If the option *Solver Integrity Check* on the *Tools* menu is checked, a second data box will appear with a comparison of the analytic and numerical values of the three velocity vector components and the divergence of the specific discharge and discharge vectors. This feature is for quality assurance and debugging purposes only.

Ctrl Shift left-click (instant pathline tracing)
A path line may be traced from the current cursor location by holding down the *Shift* and *Ctrl* key and left-clicking the mouse. The starting elevation of the particle and the direction of the trace (forward or backward in time) are defined under *Tools>Instant Inspector Settings*. Use the box *Inspection Elevation* to set the starting elevation of the instant particle trace. An instant particle trace does not define a new particle as is done on the *Model menu*. Consequently, the instant particle trace is a one time event and is not repeated after a new solve or retrace command.

Note: On the Mac, when running a virtual machine with Windows (e.g. Parallels) these keystroke combinations may differ. For instance, Instant Data Display ("Model Results" data box) may require *fn-Shift* or *Command-Shift*. The Instant Pathline Trace may require *CTR-Shift-Command*. Depending on virtual machine setup these combinations may vary and require some experimentation.