Chapter 4 - GFLOW1 Reference

Program GFLOW1 is a single layer steady state Dupuit-Forchheimer model, although it does support some transient and three-dimensional flow features. The three-dimensional features are a partially penetrating well and a sink disc which are embedded in the otherwise two-dimensional Dupuit-Forchheimer model. The model is particularly suitable for solving regional groundwater flow problems, including average (steady state) stream flow. The stream and groundwater flow problem are solved conjunctively, whereby the groundwater solution dictates baseflow, while the stream flow solution may limit the groundwater recharge rate of losing stream sections (*Mitchell-Bruker and Haitjema,1994*).

The Analytic Element Method

Program GFLOW1 is an **analytic element model** which operation differs substantially from the traditional finite difference (or finite element) models. The analytic element method was developed at the end of the seventies by Otto Strack at the University of Minnesota (*Strack and Haitjema, 1981*). For a detailed description of the method refer to (*Strack, 1989)*. A brief overview of the analytic element method follows below.

This new method avoids the discretization of a groundwater flow domain by grids or element networks. Instead, only the surface water features in the domain are discretized, broken up in sections, and entered into the model. Each of these stream sections or lake sections are represented by closed form analytic solutions: the analytic elements. The comprehensive solution to a complex, regional groundwater flow problem is obtained by superposition of all, a few hundred, analytic elements in the model.

Traditionally, superposition of analytic functions was considered to be limited to homogeneous aquifers of constant transmissivity. However, by formulating the groundwater flow problem in terms of appropriately chosen discharge potentials, rather than piezometric heads, the analytic element method becomes applicable to both confined and unconfined flow conditions as well as to heterogeneous aquifers, (*Strack and Haitjema 1981b*).

The analytic elements are chosen to best represent certain hydrologic features. For instance, stream sections are represented by line sinks, lakes or wetlands are represented by areal sink distributions. Streams and lakes that are not fully connected to the aquifer are modeled by area sinks with a bottom resistance. Discontinuities in aquifer thickness or hydraulic conductivity are modeled by use of line doublets (double layers). Specialized analytic elements may be used for special features, such as drains, cracks, slurry walls, etc.

Stepwise Modeling

Perhaps the most practical advantage of the analytic element method is its operational efficiency. In the absence of a mesh or element network, the hydrologist is concerned only with entering hydrologic features in the model. Representing streams by strings of straight line elements is a rather intuitive task. Also, for initial modeling runs, a limited set of surface water features may be introduced. Later, when insight into the groundwater flow regime increases, more data may be added to locally refine the modeling.

This stepwise modeling is not new. For example, Ward applied what he calls a "telescopic mesh refinement modeling approach" to the Chem-Dyne hazardous waste site in southwestern Ohio (*Ward et al, 1987b*). However, Ward had to use three different computer models for the three different scales at which he was modeling. Conditions on the grid boundary of the "local scale" were obtained from the "regional scale" modeling results, while similarly the conditions on the grid boundary of the "site scale" were obtained from the "local scale" modeling results. In contrast, the analytic element method allows these different scales to be treated within the same model by locally refining the input data, thus avoiding transfer of conditions along artificial boundaries from one model into the other. When necessary, even three-dimensional flow features can be included (*Haitjema,1985b*).

Main Program Features

GFLOW1 supports the following analytic elements:

- Steady state fully penetrating wells
- Transient wells (Theis' solution)
- Partially penetrating wells (confined flow only)
- Line sinks (drains, streams, lake and wetland boundaries)
- Sink discs (areal recharge, lakes, wetlands)
- Three-dimensional sink discs (confined flow conditions only)
- Inhomogeneities (regions with different hydraulic conductivity, porosity and recharge)

GFLOW1 supports *interactive graphics* which means that graphics is used within the program and data can be obtained and changed by use of pointing a cursor on a feature and pressing a key or entering data. On-screen editing is supported in the modules for wells, line sinks, sink discs, and inhomogeneity domains. Different colors and line types are used to distinguish between features. Graphical output includes: flow rates in streams, groundwater flow nets and contour plots of heads, potentials, the stream function, and the absolute value of the discharge vector.

Streamline tracing is supported with markers for travel time and streamline depth. Errors in boundary conditions and in observed heads at (observation) wells may be represented graphically, whereby line thickness and marker size are proportional to the error.

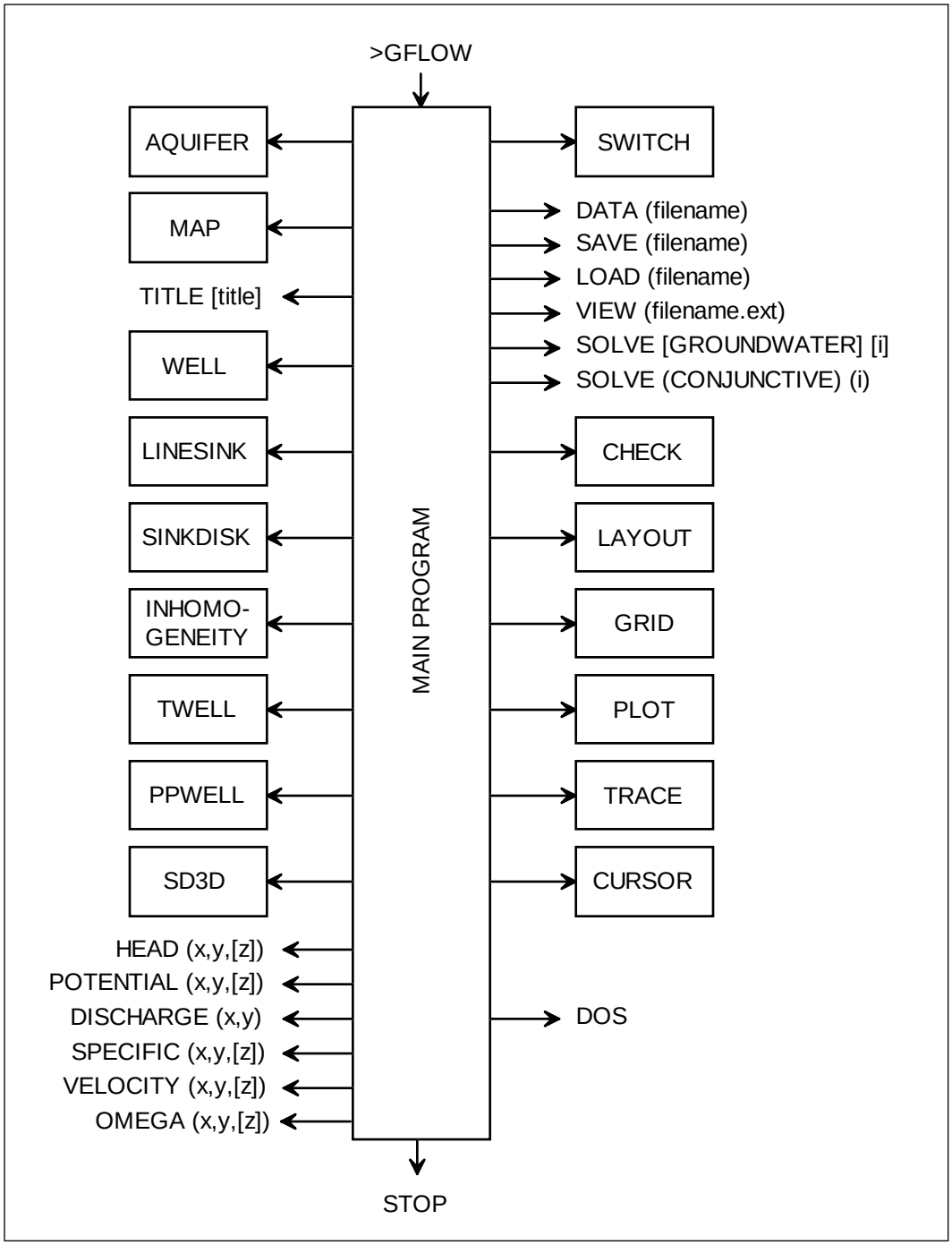


Figure 4.1 - Program structure of GFLOW1

Finally, GFLOW1 supports many numerical input and output options, among which are spread sheet files with both surface water and groundwater data along streams, and SURFER[7] files for additional graphics options. Note: SURFER file options are not available in the educational version.

```
------------------------- IO modules
-----------------------------------------
SWITCH           Reassign logical units for input or output.
SAVE (filename)  Save solution in binary form on disc (extension: .SOL).
LOAD (filename)  Load solution from disc (extension: .SOL).
DATA (filename)  Write input data file (.dat) based on current data in
program.
VIEW (fname.ext) View ASCII file on disc (e.g. input data or error file).

------------------------- GENERAL INPUT DATA modules
-------------------------
AQUIFER          Aquifer background data i.e. hydraulic conductivity,
porosity,
                 aquifer thickness, uniform flow, and a reference point.
MAP              Add background map to the layout.
TITLE (title)    16 character title included in output files.

------------------------- ANALYTIC ELEMENT modules
--------------------------
WELL             Steady state wells (2D).
PPWELL           Partially penetrating wells (3D).
PPWELL           Partially Penetrating wells (2D).
SINKDISC         Circular areal sink distributions (2D).
SD3D             Circular areal sink distributions (3D).
DISC SINK        Line sinks to model streams (2D).
INHOMOGENEITY    Areas of different permeability, recharge, and porosity.

------------------------- SOLUTION modules
SOLVE [GROUNDWATER] [i]  Solve groundwater flow problem using i iterations
                         (default: i=1).
SOLVE BASEFLOW           Solve baseflow (surface water) problem
SOLVE CONJUNCTIVE [i]    Solve groundwater and surface water conjunctively
                         using i iterations (default: i=0)
CHECK                    Check boundary conditions and piezometers.

------------------------- NUMERICAL OUTPUT modules
HEAD      (x)(y)[z] Prints head at specified coordinates.
POTENTIAL (x)(y)[z] Prints potential at specified coordinates.
DISCHARGE (x)(y)    Prints discharge vector comp. at specified coordinates
SPECIFIC  (x)(y)[z] Prints spec. disch. vector comp. at specified
coordinates.
VELOCITY  (x)(y)[z] Prints velocity vector comp. at specified coordinates.
OMEGA     (x)(y)    Prints complex potential function.

------------------------- GRAPHICAL OUTPUT modules
LAYOUT           Plot layout of hydrological features.
GRID             Set up grid for plotting (2D/3D).
PLOT             Contour plots of heads, potentials, stream function,
flownet,
                 and absolute value of the discharge vector (2D/3D)
```

Figure 4.2 - Contents of the command summary file ***GFSUM.HLP***

[7] SURFER is a trademark of GOLDEN SOFTWARE, Incorporated.

Program Organization

GFLOW1 is *modular* in the sense that different program functions are handled in different program modules, each with their own command menu. In Figure 4.1, a diagram is presented of the main structure of GFLOW1.

The available modules in GFLOW1 are shown in in boxes connected to the MAIN PROGRAM box. When typing a module name, the user leaves the MAIN PROGRAM, enters the module, and remains in the module until the command `QUIT` or the `<ESC>` key is typed. In addition to entering modules, the user may also initiate commands which are part of the MAIN PROGRAM itself. These commands are the ones not emboxed in Figure 4.1. After such a command is executed, the user is still in the MAIN PROGRAM. A command may have one or more parameters, indicated in Figure 4.2 between parentheses. An argument between square brackets is optional, if omitted a default value is used.

A brief explanation of the modules and commands shown in Figure 4.1 is provided in Figure 4.2, grouped by function. There are six main functions: input and output (I/O) to disk, input of general data, input of analytic elements, generating and checking a solution, producing numerical output, and producing graphical output. These functions are not strictly separated, for instance, graphics and file I/O is supported in many other modules.

Command Menus

Program GFLOW1 is "command driven" in combination with command menus. A fully "menu driven" program allows the user to "point and shoot" to initiate commands and to type data in data boxes. The advantage of a menu driven program is that the user is prompted for every response, making program operation easy at first. However, when the user becomes proficient in the use of the program, the (graphical) menu interface gets in the way; you have to step through many options and menus before the desired action occurs.

In addition, most data input in GFLOW1 is best done in "batch mode", rather than interactively from the keyboard or by use of a mouse. Such a batch input option requires a command structure. In an attempt to use the best of both worlds, GFLOW1 employs commands in combination with a command menu. The command menu displays the available commands with their current parameters and options. The format of the command menu reflects the syntax of the commands. The command menu is automatically displayed in all modules.

A brief explanation of the commands in the main program is displayed by typing `COMMAND` in the main program; it is reproduced in Figure 4.2. Similarly, in each of the modules you may press `<F1>` to open a context-sensitive help file. This file contains a brief explanation (summary) of the commands followed by a more elaborate discussion of the use of the module with example command sequences. You can scroll or page back and forth through this help file and return to the command menu by pressing `<ESC>`.

Special Keys

- <F1> - Help!

  At any menu in GFLOW1, help is available by pressing the <F1> key. The help file for the current module is displayed , one page at a time. The file can be scrolled up or down using the arrow keys or <Page Up> and <Page Down> keys. Press <ESC> to return to the command prompt.

- <F2> - Go!

  Several modules in GFLOW1 have a GO command. The <F2> key may be used as an alternative to typing GO <CR>.

- <F7> key - Graphics Printing

  At any time while viewing a graphic screen, the graphical display can be prepared for printing by pressing the <F7> key. GFLOW1 will then schedule the GFPRINT output postprocessor, which allows the user to set margins, select output devices, or send results to a variety of graphical output file formats. See the GFPRINT documentation.

- <ESC> - Return to previous menu

  When in any GFLOW1 module, the <ESC> key can be used as an alternative to the QUIT command.

Free-Form Input

- GFLOW1 has no conventions for the spacing of commands and parameters on a command line. GFLOW1 interprets spaces and commas as delimiters between parameters. When entering values greater than 1000, DO NOT use commas as part of the number; if GFLOW1 encounters the following input:

  ```
  TIME 1,000 <CR>
  ```

  it will interpret the line as TIME 1 and ignore the three zeroes. The proper command would be entered as

  ```
  TIME 1000 <CR>
  ```

Main Menu

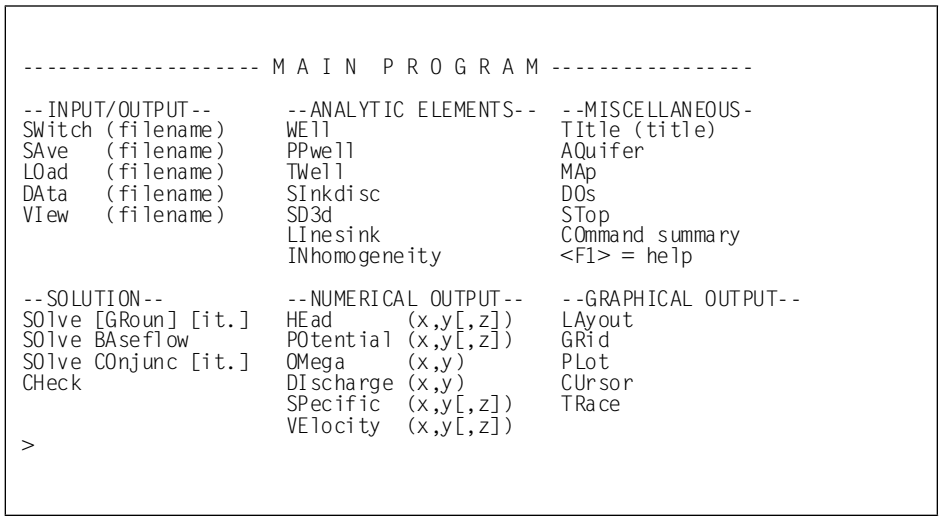


Figure 4.3 - GFLOW1 Main Menu

This menu provides access to modules which support creation of elements and inspection of results. Each of the modules will be discussed separately. In addition, a number of commands at this level are executed immediately.

Program and Data Management Commands (Directly Executed)

- `TItle` — Enters a title for the model. The title (up to 16 characters) is written to a data file created by the `DAta` command.
- `DAta` — Writes an ASCII analytic element data file (filename.DAT). This file may be read by GAEP for element creation and editing if an appropriate digital map file exists. Any changes made to elements during the GFLOW1 session are saved to this file.
- `SAve` — Saves a binary solution (filename.SOL) file. Can be reread by GFLOW1.
- `LOad` — Loads a binary solution file created by the SAVE command.
- `VIew` — Views any ASCII file, such as an analytic element data file (filename.DAT).
- `SOlve` — Solves the groundwater flow problem.

- `DOs` Temporarily escapes to DOS.
- `STop` Terminates GFLOW1 execution.

Model Inspection Commands (Directly Executed)

- `HEad` Reports the piezometric head at a point, given its location.
- `POtential` Reports the discharge potential at a point, given its location.
- `OMega` Reports the complex potential at a point, given its location.
- `DIscharge` Reports the total discharge at a point, given its location.
- `SPecific` Reports the specific discharge at a point, given its location
- `VElocity` Reports the groundwater velocity at a point, given its location.

Analytic Element Modules

- `AQuifer` Allows definition of aquifer properties.
- `MAp` Allows creation of background maps for orientation.
- `WEll` Allows creation and editing of well elements.
- `LInesink` Allows creation and editing of line sink elements.
- `SInkdisk` Allows creation and editing of circular sink disk elements.
- `INhomogeneity` Allows creation and editing of inhomogeneous aquifer domains.
- `TWell` Allows creation of transient wells, governed by the Theis equation.
- `PPwell` Allows creation of 3-dimensional partially penetrating well elements.
- `SD3d` Allows creation of 3-dimensional sink disk elements.

Modules for Special Functions and Analysis

- `SWitch` Manages file input and output. Allows GFLOW1 input and output to be reassigned from the keyboard or screen to files or devices. This module is used, for instance, to read analytic element data (filename.DAT) files.

- `CHeck` Provides a variety of functions for model performance evaluation, including both numerical and graphical checks on specified boundary conditions and comparison of predicted and observed heads.
- `LAyout` Displays a layout of model elements.
- `GRid` Allows computation of grids for contour plots.
- `PLot` Displays contour plots.
- `TRace` Provides streamline tracing functions.
- `CUrsor` Allows inspection of model results using the mouse.

Main Menu

AQuifer

Command Description

Enters the `AQuifer` module, which facilitates the geometry and properties of the regional aquifer (see Aquifer Module section).

Example

- To enter the aquifer module, use the command

  ≥ `AQUIFER <CR>`

Main Menu

CHeck

Command Description

Enters the CHeck module, which facilitates checking of model integrity and comparison of modeled and observed heads (see Check Module section).

Example

- To enter the check module, use the command

  ≥ CHECK <CR>

Main Menu

DAta

Command Description

Writes an ASCII format analytic element data file. The file contains all of the GFLOW1 commands which would be required to create the current model. The file contains all of the analytic elements, a command to input the `.MAP` files to be displayed and the maximum window setting. All changes in input data made during the GFLOW1 session will be reflected in the new data file. The filename will automatically receive the extension `.DAT`; any other extension will be changed to `.DAT`.

Note

Only the input data are saved to the new data file. For head-specified features and inhomogeneities, only the input parameters will be saved; any strengths computed (see `SOlve` command) will not be written to this file. To save a complete solution to disk, see the `SAve` command.

The data file created is readable by GAEP, providing a two-way facility for creation of models with GAEP and editing via GAEP or GFLOW1 (provided an appropriate digital map file exists). If the specified `.DAT` file already exists, GFLOW1 allows the file to be rewritten or for a new filename to be provided.

Example

To save the current model information as RUN1.DAT, use either of the following commands:

```
> DATA RUN1.DAT <CR>
```

or

```
> DATA RUN1 <CR>
```

GFLOW1 will create the file `RUN1.DAT`.

See Also

`SWitch` module
`SAve` and `LOad` commands (Main module)

Main Menu

DIscharge

Command Description

Returns the total discharge vector, the total flow integrated over the aquifer height (in units of length^2/time), at the point specified. Only the (x,y) coordinates need to be specified, since the discharge is a function only of the (x,y) position..

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Note

A valid solution must exist before using this command. An error message will be returned by this command if any changes to the model have been made since the last `SOlve` command.

Example

To find the discharge vector at the location (500,500) in the current coordinate system, use the command

```
≥ DISCHARGE 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

DOs

Command Description

Provides an escape to the MS-DOS shell specified in the `COMSPEC` environmental variable (usually `C:\COMMAND.COM`). Exiting the DOS shell returns control to GFLOW1.

Note

Use of any DOS command which loads a TSR (terminate-and-stay-resident) program, such as the DOS `PRINT` command, may cause unpredictable results. These commands are strongly discouraged - load all necessary TSRs before invoking GFLOW1.

Example

To temporarily escape to DOS, use the command:

```
> DOS <CR>
```

To return to GFLOW1, use the DOS `EXIT` command:

```
C:\> EXIT <CR>
```

Main Menu

GRid

Command Description

Enters the GRid module, which facilitates the generation of a grid of model information for contouring (see Grid Module section).

Example

- To enter the grid module, use the command

  ≥ GRID <CR>

Main Menu

HEad

Command Description

Returns the hydraulic head (in units of length) at the point specified. In 2-dimensional zones, only the (*x*,*y*) coordinates need to be specified. In 3-dimensional zones (near PPWELL and PD3D elements), the head varies in the vertical direction, and the elevation (*z*) must be provided.

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Note

A valid solution must exist before using this command. An error message will be reported for this command if any changes to the model have been made since the last `SOlve` command.

Example

To find the head at the location (500,500) in the current coordinate system, use the command

```
> HEAD 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

LAyout (Graphics Menu)

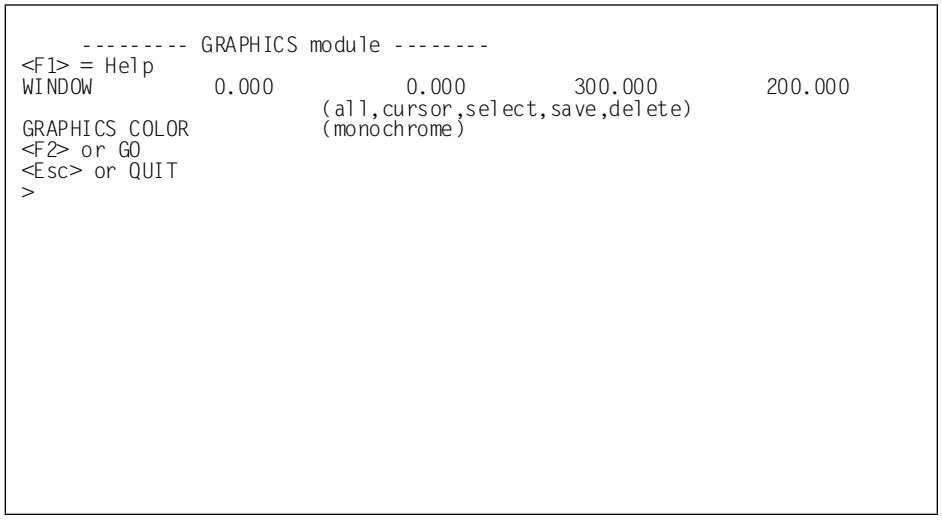


Figure 4.4 - Graphics Menu

Command Description

Provides a simple viewing facility for the model. The Graphics Menu is also made available at other points in the program when graphical displays are constructed.

Commands

- `WIndow` Allows the user to set the size of the window. The command takes several forms;

  To set the window to a size large enough to hold all elements currently defined, enter the command as follows.

  `WINDOW ALL <CR>`

  To set the window to a specified set of model coordinates, enter the command as follows. The coordinates *(x1,y1)* and *(x2,y2)* become the lower left and upper right corners of the window.

  `WINDOW X1 Y1 X2 Y2 <CR>`

  To use the mouse cursor to select a window size and location, enter the command `WIndow CUrsor <CR>`. The graphics screen will appear with a crosshair; select the lower left corner with the mouse and press `<CR>`. Now, select the upper right corner and again press `<CR>`.

- To save the current window for future use, use the command `WIndow SAve <CR>`. The saved window can be reselected by the `WIndow SElect` command.
- To reselect a previously saved window, use the command `WIndow SElect`. The graphics screen appears with all available windows shown in red. Place the mouse cursor at any corner of the desired window and press `<CR>`. It may be necessary to execute the command `WIndow ALl` prior to using this command.
- The WIndow command may also be used to create horizontal or vertical slices over the aquifer in areas of 3-D flow. The command

  `WINDOW` X_1 Y_1 Z_0 X_2 Y_2 Z_0

  creates a horizontal slice at elevation z0 with (x_1,y_1) and (x_2,y_2) the lower left and upper right corners of the window (or site). The command

  `WINDOW` X_1 Y_1 Z_1 X_2 Y_2 Z_2

  creates a vertical slice through the aquifer with the lower left corner at (x_1,y_1,z_1) and the upper right corner at (x_2,y_2,z_2).

  Note

  Streamline tracing in horizontal or vertical slices in 3-D windows is only possible if these windows are symmetry planes, as otherwise streamlines do not remain in the plane of the window.

- `DIsplay` Sets the graphics display mode to either color or monochrome.

  To set the display mode to color, enter the command
  `DISPLAY COLOR <CR>`

  To set the display mode to monochrome, enter the command
  `DISPLAY MONOCHROME <CR>`

- `GO or <F2>` Proceeds to display the current layout (`LAyout` command) or on to the next graphics step as defined in the module selected (see module documentation).
- `QUIT or <ESC>` Returns to the Main Menu (`LAyout` command) or to the outer menu for the module selected.

Main Menu

LInesink

Command Description

Enters the `LInesink` module, which facilitates the creation and editing of line sink elements (see Linesink Module section).

Example

- To enter the linesink module, use the command

  ≥ `LINESINK <CR>`

Main Menu

LOad

Command Description

Reads a binary format solution data file created with the SAve command. The file restored the status of GFLOW1, including all elements and their computed strength parameters from the previous SOLVE command. All previous model information in GFLOW1 is overwritten and lost.

The binary solution file cannot be reread by any other program. The filename specified will be given the extension .SOL; any other extension will be replaced by the extension .SOL. To create an ASCII data file from a .SOL file, use the LOad command followed by the DAta command (see description of the DAta command).

Example

To load the model solution in the binary file RUN1.SOL, use either of the following commands:

```
> LOAD RUN1.SOL <CR>
```

or

```
> LOAD RUN1 <CR>
```

GFLOW1 will restore GFLOW1 to the state saved in the file RUN1.SOL.

See Also

SAve command (Main module)

Main Menu

MAp

Command Description

Enters the MAp module, which facilitates the creation and display of illustrative background maps (see Map Module section).

Example

- To enter the map module, use the command

  ≥ MAP <CR>

Main Menu

OMega

Command Description

Returns the complex potential function at the point specified. Since the complex potential is a purely two-dimensional function, only the (*x*,*y*) coordinates need to be specified. The complex potential consists of two parts; the real part (first number) is the discharge potential (see `POtential` command) and the imaginary part (second number), the stream function (valid only in regions where no recharge exists).

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Example

To find the complex potential at the location (500,500) in the current coordinate system, use the command

```
> OMEGA 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

PLot (Contour Plotting Menu)

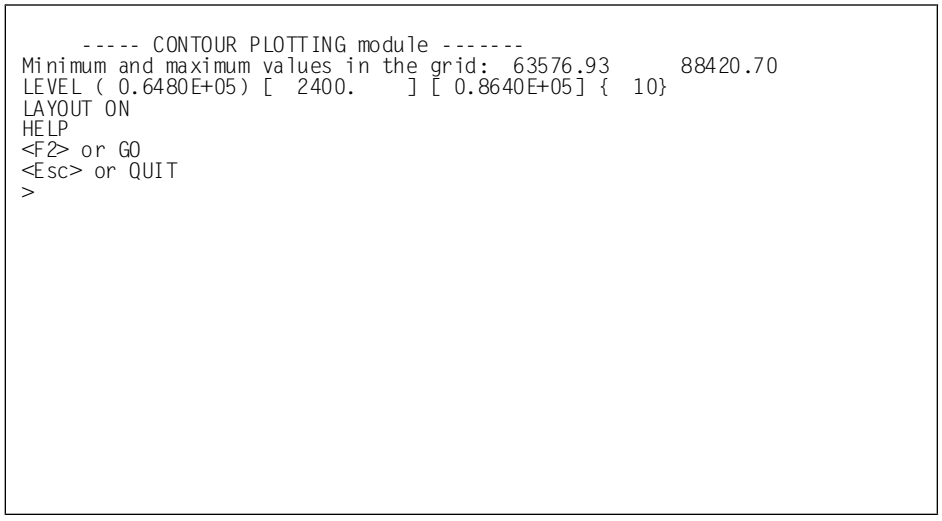


Figure 4.5 - Contour Plot Menu

Command Description

Allows construction of contour plots. The Contour Plot Menu (shown) is also used in other program modules to provide access to contour plotting.

Commands

- `LEvel` Allows the user to set contour levels. The default levels are shown. To set levels, enter one of the following command forms:

To draw a single contour at the level 520 , use the following command:

```
LEVEL 520 <CR>
```

To draw contours starting at 520 with the increment 10, use the command:

```
LEVEL 520 10 <CR>
```

To draw contours starting at some starting level 520 with the increment 10, up to level 600 use a command of the following form:

```
LEVEL 520 10 600 <CR>
```

Note

The parameter between braces {} on the menu display is the number of contour levels which will be plotted based ont he current settings.

- `LAyout` Enables or disables the display of element and background map information. The command takes one of the following forms:

  To enable the element layout:
  `LAYOUT ON <CR>`

  To disable the element layout:
  `LAYOUT OFF <CR>`

- `GO or <F2>` Draws the contour plot (`PLot` command) or proceeds to the next step for other modules.

- `QUIT or <ESC>` Returns to the GFLOW1 Main Menu (`PLot` command) or skips the contour plotting and proceeds to the next module.

Main Menu

POtential

Command Description

Returns the discharge potential (in units of length^3/time) at the point specified. In 2-dimensional zones, only the (x,y) coordinates need to be specified. In 3-dimensional zones (near PPWELL and PD3D elements), the potential is defined as $kH\phi$, where k is the hydraulic conductivity, H is the confined aquifer height and ϕ is the head. Since the head (and thus the potential) varies with depth, the z-coordinate must be provided also.

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Note

A valid solution must exist before using this command. An error message will be returned for this command if any changes to the model have been made since the last `SOlve` command.

Example

To find the discharge potential at the location (500,500) in the current coordinate system, use the command

```
> POTENTIAL 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

PPwell

Command Description

Enters the PPwell module, which facilitates the creation of three-dimensional partially penetrating well elements (see PPWell Module section).

Example

- To enter the PPWell module, use the command

  ≥ PPWELL <CR>

Main Menu

SAve

Command Description

Writes a binary format solution data file. The file contains the current status of GFLOW1, including all elements and their computed strength parameters from the previous SOLVE command. The saved file can be reloaded for future analysis, such as streamline tracing.

The binary solution file cannot be reread by any other program. The filename specified will be given the extension .SOL; any other extension will be replaced by the extension .SOL. To create an ASCII data file from a .SOL file, use the LOad command followed by the DAta command (see descriptions elsewhere in the manual).

If the specified file already exists, GFLOW1 allows the file to be rewritten or for a new filename to be provided.

Example

To save the current model solution in the binary file RUN1.SOL, use either of the following commands:

```
≥ SAVE RUN1.SOL <CR>
```

or

```
≥ SAVE RUN1 <CR>
```

GFLOW1 will create the file RUN1.SOL.

See Also

LOad command (Main module)

Main Menu

SD3d

Command Description

Enters the SD3d module, which facilitates the creation of three-dimensional sink discs (see SD3D Module section).

Example

- To enter the SD3D module, use the command

  ≥ SD3D <CR>

Main Menu

SInkdisc

Command Description

Enters the `SInkdisk` module, which facilitates the creation and editing of sink disc elements (see Sinkdisc Module section).

Example

- To enter the sinkdisc module, use the command

  ≥ `SINKDISC <CR>`

Main Menu

SOlve

Command Description

Initiates the matrix solution procedures necessary to determine unknown (strength) parameters of the analytic elements (e.g. discharge rate of head specified line sinks, sink discs, and wells). When solving the surface water and groundwater flow problem conjunctively, groundwater and surface water solutions are performed alternately during each iteration.

The `SOlve` command allows the matrix to be solved iteratively, to allow convergence of resistance-specified boundary conditions and baseflow. The user specifies the desired number of iterations as part of the `SOlve` command. The user should continue to perform the solution procedure until consecutive solutions converge (this can be as many as 6 or 8 times, for complex models perhaps more than 10).

If the desired accuracy is reached well before the number of iterations you specified, press the `<CTRL>` and `<BREAK>` keys (simultaneously) to prevent unnecessary further iterations.

Considerations in the Solution Procedure

- When the head drops below the bottom of a resistance layer underneath a stream or lake, or when the infiltration rate exceeds the saturated hydraulic conductivity, the line sink or sink disc is treated as percolating and is removed from the system of equations. When line sinks or sink discs are given a resistance, a second solution is added to the first solution to determine the existence of such percolating features. Several additional iterations may be necessary to determine which surface waters are percolating and to obtain acceptable errors in the specified boundary conditions.
- It is good practice to inspect the integrity of the solution by plotting errors in boundary conditions; see the `CHeck` module. In general, reported errors in specified boundary conditions should be less than 1%.
- Occasionally, a rather large error may occur in a resistance specified surface water, while that surface water has a very small infiltration or extraction rate. This may be verified in the "check" module where errors of individual analytic elements may be listed. Such a relatively large error in an hydrologically insignificant surface water feature may be ignored without compromising the accuracy of the overall solution.
- The command `SOLVE BASEFLOW` initiates the procedure to calculate baseflow in the stream network (stream commands in the line sink module).

This command is usually not needed, however, since the stream flow should be solved conjunctively with the groundwater flow.

- Conjunctive groundwater/surface water solutions automatically alternate between groundwater and streamflow calculations at each iteration. The solution procedure ends with a groundwater solution and stream flow solution without corrections on the stream infiltration rates in order to preserve the integrity of the groundwater flow solution.

Examples

- For a simple model with all element resistances zero, no iterations are required. Use the command:

  ≥ SOLVE <CR>

- For a more complex problem with nonlinear features (nonzero resistances for boundary conditions), an iterative solution procedure is required. Use the command:

  ≥ SOLVE N <CR>

  or

  ≥ SOLVE GROUNDWATER N <CR>

  where N is the desired number of iterations. You might start with 5 iterations and use more if errors in specified boundary conditions are still too large.

- For a problem which includes surface water/groundwater interactions, a conjunctive solution is required. Use the command:

  ≥ SOLVE CONJUNCTIVE N <CR>

  where N is the desired number of iterations.

Note

If sufficient accuracy is obtained before the specified number of iterations (N) are completed, press the <CTRL> and <BREAK> keys (simultaneously) to abort the solution procedure. The current iteration will be completed before aborting the solution. This guarantees a valid solution after the procedure is aborted.

See Also

CHeck module

Main Menu

SPecific

Command Description

Returns the specific discharge (in units of length/time) at the point specified. In 2-dimensional zones, only the (*x*,*y*) coordinates need to be specified. In 3-dimensional zones (near PPWELL and PD3D elements), the discharge varies in the vertical direction, and the elevation (*z*) must be provided. Note that in Dupuit-Forschheimer 2-dimensional models with recharge, the specific discharge is also variable in the vertical (*z*) direction, as q_z is approximated from continuity of flow. If the *z* coordinate is omitted, the specific discharge vector is computed at the saturated aquifer top.

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Note

A valid solution must exist before using this command. An error message will be returned by this command if any changes to the model have been made since the last `SOlve` command.

Example

To find the specific discharge at the location (500,500) in the current coordinate system, use the command

```
> SPECIFIC 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

STop

Command Description

Exits GFLOW1 and returns to MS-DOS. If any of the the current elements have been modified since they were created, GFLOW1 will ask if the user wishes to write a new `.DAT` file (using the `DAta` command).

Example

To leave GFLOW1, use the command:

```
> STOP <CR>
```

Main Menu

SWitch

Command Description

Enters the `SWitch` module, which facilitates management of input and output files and devices (see Switch Module section).

Example

- To enter the switch module, use the command

  ≥ `SWITCH <CR>`

Main Menu

TItle

Command Description

Sets a title for the current model. The title will be saved in all files created by the SAve and DAta commands for future model runs.

Examples

- To enter the title RUN#1 for the current model, the command

  ≥ TITLE RUN#1 <CR>

  may be entered from the main menu.
- To display the current title, type the command

  ≥ TITLE <CR>

Main Menu

TRace

Command Description

Enters the TRace module, which facilitates streamline tracing (see Trace Module section).

Example

- To enter the trace module, use the command

  ≥ TRACE <CR>

Main Menu

TWell

Command Description

Enters the `TWell` module, which facilitates the creation of transient well elements (see TWell Module section).

Example

- To enter the transient well module, use the command

  ≥ `TWELL <CR>`

Main Menu

VElocity

Command Description

Returns the groundwater velocity (in units of length/time) at the point specified. In 2-dimensional zones, only the (*x*,*y*) coordinates need to be specified. In 3-dimensional zones (near PPWELL and SD3D elements), the velocity varies in the vertical direction, and the elevation (z) must be provided. Note that in Dupuit-Forschheimer 2-dimensional models with recharge, the vertical velocity is variable in the vertical (z) direction, where v_z is approximated from continuity of flow. If the *z* coordinate is omitted, the velocity is computed at the saturated aquifer top.

If a transient well (TWELL) element is in use, GFLOW1 will request that the time value be entered. Enter the time since the (first) well started pumping.

Note

A valid solution must exist before using this command. An error message will be returned by this command if any changes to the model have been made since the last `SOlve` command.

Example

To find the groundwater velocity at the location (500,500) in the current coordinate system, use the command

```
> VELOCITY 500 500 <CR>
```

Note

The results of this command can be echoed to a file by use of the `OUtput` command in the `SWitch` module.

See Also

`SWitch` module

Main Menu

VIew

Command Description

Provides a file viewing facility. This is useful for examining the contents of any ASCII file on disk, such as a `.DAT` file containing GFLOW1 commands or an output file containing textual data created using the `SWitch` module. The VIew command examines the text file one page at a time and allows the display to be scrolled using the up and down arrow keys and the `<Page Up>` and `<Page Down>` keys. Press `<ESC>` to return to GFLOW1.

Note

Since any ASCII file may be viewed, there is no default filename extension associated with the `VIew` command. Hence a filename extension must be provided by the user when using the `VIew` command.

Example

To view the contents of the ASCII file RUN1.DAT, use the following command:

```
≥ VIEW RUN1.DAT <CR>
```

Press page up/down or arrow keys to scroll through the file, then press `<ESC>` to return to GFLOW1.

Main Menu

WEll

Command Description

Enters the `WEll` module, which facilitates the creation and editing of well elements (see Well Module section).

Example

- To enter the well module, use the command

  ≥ `WELL <CR>`

```
      ---------- AQUIFER module -----------
PERMEABILITY   1.000000
POROSITY  0.2000000
THICKNESS   1.000000
BASE  0.0000
REFERENCE x,y         0.0        0.0 head   1.000000
UNIFLOW Qx  0.0000000     Qy  0.0000000
<Esc> or QUIT
>
```

Figure 4.6 - Aquifer Module Menu

The aquifer module allows the user to define the regional aquifer properties. In case the aquifer is heterogeneous, the properties defined in the `AQuifer` module are the "background properties" for the aquifer. Separate domains may be introduced in which hydraulic conductivity, porosity and areal recharge differ from the background values (see `INhomogeneity` module).

The `AQuifer` module menu illustrates the command syntax and displays current parameter settings.

Commands

- `PErmeability` Sets aquifer permeability (hydraulic conductivity).
- `POrosity` Sets aquifer porosity.
- `THickness` Sets aquifer thickness.
- `BAse` Sets aquifer base elevation.
- `REference` Defines the reference point for the model.
- `UNiflow` Creates a uniform flow field.
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu

Aquifer Module

BAse

Command Description

Sets the base elevation for the aquifer. In inhomogeneous aquifers, the value chosen will apply within all `INhomogeneity` domains. Differences in aquifer base elevation are not supported in this version of GFLOW1.

Note

The aquifer base in GFLOW1 is assumed to be horizontal. You should select the average elevation of the top of an appropriate clay layer or bedrock surface that underlies your aquifer. Select the aquifer base such that it is most accurate in the area of interest ("near field").

Also Note

Since GFLOW1 works in a dimensionless system, it is up to the user to ensure that the units of length and time are consistent when entering properties. The units for base elevation are [length].

Example

To set the aquifer base elevation at 500 feet above mean sea level, (assuming that all other units are consistent with feet and days), use the command

```
> BASE 500 <CR>
```

Aquifer Module

PErmeability

Command Description

Sets the permeability (hydraulic conductivity) for the regional aquifer. In inhomogeneous aquifers, the value chosen should be that desired for the area outside all `INhomogeneity` domains.

Note

Since GFLOW1 works in a dimensionless system, it is up to the user to ensure that the units of length and time are consistent when entering properties. The units for permeability are [length/time]. Recommended units: feet/day or meters/day.

Example

To set a permeability of 100 feet/day (assuming that all other units are consistent with feet and days), use the command

```
> PERMEABILITY 100 <CR>
```

See Also

`INhomogeneity` module

Aquifer Module

POrosity

Command Description

Sets the effective porosity for the regional aquifer. In inhomogeneous aquifers, the value chosen should be that desired for the area outside all `INhomogeneity` domains. The value should be a fraction between 0 and 1.0.

Adjustments to porosity will not affect modeled hydraulic heads, but will change groundwater velocities and travel times.

Example

To set a porosity of 20% (0.2), use the command

```
> POROSITY 0.20 <CR>
```

Aquifer Module

REference

Command Description

Sets the location and hydraulic head at the reference point. The reference point is a location which provides a base point for all of GFLOW1's internal computations (see one of the detailed texts about the Analytic Element Method for a further discussion). In regional models, the reference point should be chosen far from the study area being modeled and outside all hydrological features (streams and lakes) included in the model. The head at the reference point is best selected roughly equal to the average heads specified at streams and lakes.

Note

Since GFLOW1 works in a dimensionless system, it is up to the user to ensure that the units of length and time are consistent when entering properties. The units for hydraulic head are [length]. Coordinates should be entered in the same units as the head.

Example

To place the reference point at the location (100000,100000) with a reference elevation of 505 feet, use the command

```
> REFERENCE 100000 100000 505 <CR>
```

Note

If sufficient "far field" features surround the "near field", changes in the head at the reference point will not noticeably affect the groundwater flow solution in the "near field". It is good practice to test this by examining solutions for several different values of the reference head.

Also Note

There is no need to associate the location and water level of the reference point with an actual surface water feature. It is better to choose an arbitrary point, because an actual surface water feature may have a head too high or too low to be an appropriate reference head; the reference head should be approximately equal to the average head in the model domain.

Aquifer Module

THickness

Command Description

Sets the thickness of the aquifer. In inhomogeneous aquifers, the value chosen will apply inside all `INhomogeneity` domains. Heterogeneous aquifer thickness is not supported in GFLOW1.

Note

If unconfined flow is to be modeled throughout the aquifer, the aquifer thickness should be set to some arbitrarily large value (for example, 1000).

Also Note

Since GFLOW1 works in a dimensionless system, it is up to the user to ensure that the units of length and time are consistent when entering properties. The units for thickness are [length]. Recommended units are feet or meters.

Example

To set an aquifer thickness of 25 feet (assuming that all other units are consistent with feet and days), use the command

```
> THICKNESS 25 <CR>
```

Aquifer Module

UNiflow

Command Description

Creates a uniform flow field in the aquifer domain. The user must compute the magnitude of the discharge vector from the direction and magnitude of the hydraulic gradient and aquifer transmissivity. If changes are made in the hydraulic conductivity (see the `PErmeability` command), the user will need to recompute the magnitude of the discharge vector and reenter it. The user specifies the magnitude of the discharge vector components in both the *x* and *y* directions.

This facility is intended for conceptual modeling of simple local problems. It is generally inappropriate for use in regional modeling.

Note

Since GFLOW1 works in a dimensionless system, it is up to the user to ensure that the units of length and time are consistent when entering properties. The units for uniform flow discharges are [length²/time].

Example

To create a uniform flow field with magnitude 1 ft²/day and direction 30º measured clockwise from the *x* axis, first compute the components of the discharge vector (0.866 ft²/day in the *x* direction and 0.5 ft²/day in the *y* direction), and then enter the command

≥ UNIFLOW 0.866 0.5 <CR>

This page is intentionally left blank.

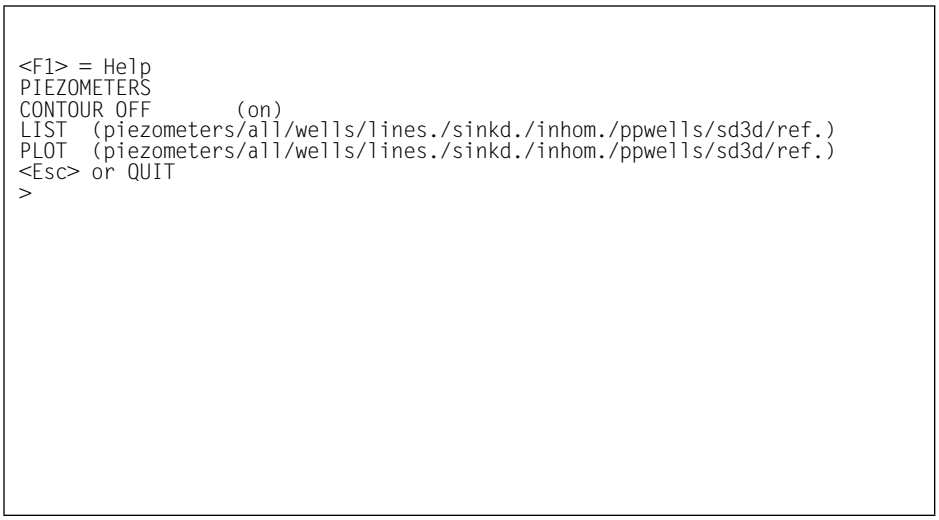


Figure 4.7 - Check Module Menu

The CHECK module provides tools for checking boundary conditions and matches between observed and modeled heads (at "piezometers"). These data are provided in both numerical and graphical form.

Commands

- `PIezometers` Allows definition of piezometers (points of known heads).
- `COntour` Allows contour lines to be plotted in CHECK module graphics.
- `LIst` Prints a textual list of errors for boundary conditions or piezometers.
- `PLot` Plots a graphical display of errors for boundary conditions or piezometers.
- `QUit` Returns to the GFLOW1 Main Menu.

Check Module

COntour

Command Description

Enables or disables the display of contour lines on CHECK module graphical displays. By default, GFLOW1 does not include this information as part of graphical displays. Contour lines may be displayed only if a current grid exists (see `GRid` module). If contour lines are enabled, GFLOW1 will prompt for levels after the `GO` command is issued.

The `COntour` setting remains in effect until changed.

Example

- To enable contour lines, enter the following commands:

  ≥ `CHECK <CR>`

  ≥ `CONTOUR ON <CR>`

- To disable contour lines (the default), enter the following commands:

  ≥ `CHECK <CR>`

  ≥ `CONTOUR OFF <CR>`

Check Module

FIlter

Command Description

Allows control of the scaling parameters used to determine the triangle sizes that mark differences in modeled and observed heads (see `PLot PIezometers` command, Check Module). This command becomes available when issuing the `PLot PIezometer` command.

The user may select a lower bound on the absolute differences when some piezometers are outside the area of interest with large differences, forcing all triangles in the area of interest to be small, thus obscuring the distribution of differences.

Example

- Assuming the user has entered a set of piezometers, the piezometer comparison plot is initiated by the commands below (GFLOW1 reports the minimum and maximum errors for all piezometers):

  ```
  ≥ CHECK <CR>
  ≥ PLOT PIEZOMETERS <CR>
  MIN -23.7   MAX +31.4
  ```

  To set the scaling for the marker size to the range -10 feet to +10 feet, enter

  ```
  ≥ FILTER -10 10 <CR>
  ```

- It is advised that "outlyer" piezometers be removed and the same set of FIlter values be used throughout a series of runs to facilitate comparison of results:

  ```
  ≥ CHECK <CR>
  ≥ PLOT PIEZOMETERS <CR>
  MIN -5.8   MAX +7.9
  ≥ FILTER -10 10 <CR>
  ```

  Note

  If the set of piezometers used was created using GAEP, the command `FILTER -10 10` is included in the piezometer file for default scaling, simplifying the comparison of results.

Check Module

LIst

Command Description

Lists the errors at element control points or the difference between modeled and actual heads for piezometers in a text (numerical) format. For analytic elements, the percentage error between conditions specified at control points are shown. For piezometers, the difference between modeled and actual heads is shown. The user must specify a parameter which determines which type of feature is to be checked (see below).

Output is in a tabular format, and can be routed to a file by use of the `OUtput` command in the `SWitch` module.

Options

- `PIezometers` Show piezometer differences
- `ALl` Show all analytic elements
- `WElls` Show wells
- `LInesinks` Show line sinks
- `SInkdiscs` Show disc sinks
- `INhom` Show inhomogeneities (line doublets)
- `SD3D` Show 3-dimensional sink discs
- `PPwell` Show 3-dimensional partially penetrating wells
- `REf` Show the error at the reference point

Example

- To examine the errors in all linesinks, enter the following commands:

```
> CHECK <CR>
> LIST LINESINKS <CR>
```

Check Module

Piezometers

Command Description

Allows definition of points of known head. These points are entered as *(x,y)* points, the known head and an optional label.

Note

If a set of piezometers is already defined, re-entry of the `PIezometer` command clears the loaded piezometers. Piezometer creation is supported in the GAEP program as well, for digitizer support.

Example

- To create a set of piezometers, use the following commands:

```
> CHECK <CR>
> PIEZOMETERS <CR>
```

Now, enter the piezometer locations, heads and labels,

```
100 110  45  PIEZO1
105 140  46  PIEZO2
...
```

All piezometers are defined. Use the command

```
QUIT <CR>
```

Check Module

PLot

Command Description

Plots the errors at element control points or the difference between modeled and actual heads for piezometers in a graphical format. For analytic elements, the percentage error in the boundary conditions at the control points is shown as a line width, with the width of the line proportional to the error. For piezometers, the differences between modeled and actual heads are shown as small triangles which point up when the modeled head is higher than observed and down when the modeled head is lower than observed; the triangle's size is scaled to the maximum error (or to the `FIlter` value, if specified). The user must specify a parameter which determines which type of feature is to be checked (see below).

When the `PLot` command is entered, the GRAPHICS menu (see the LAyout command from the Main Menu) is displayed. Type GO <CR> to proceed. If the COntour ON parameter is set (see above), the Plot menu will be shown. Type GO <CR> to view the plot. As with all graphical displays, the <F7> key will send the graphics screen to the GFPRINT program for printing.

Options

- `PIezometers` Plot piezometer differences (see also `FIlter` command)
- `ALl` Plot all analytic elements
- `WElls` Plot wells
- `LInesinks` Plot line sinks
- `SInkdiscs` Plot disc sinks
- `INhom` Plot inhomogeneities (line doublets)
- `PPwell` Plot 3-dimensional partially penetrating wells
- `SD3D` Plot 3-dimensional disc sinks
- `REf` Plot the error at the reference point

Example

- To examine the errors in all linesinks, enter the following commands:

```
> CHECK <CR>
> PLOT LINESINKS <CR>
```

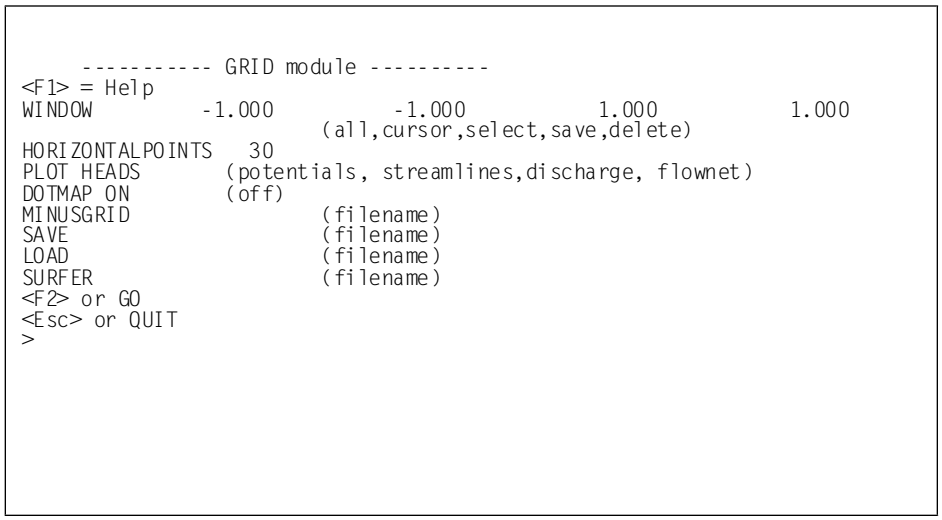


Figure 4.8 - Grid Module Menu

The GRID module provides facilities for creating a grid of data from the model solution for contouring purposes. It is necessary to create a grid when prior to creating a contour plot.

Commands

- `WIndow` Sets the graphical window size in the model coordinates.
- `HOrizontal` Sets the horizontal resolution of the grid.
- `PLot` Selects the function to be gridded (head, potential, stream function, discharge magnitude or a flownet).
- `DOtmap` Instructs GFLOW1 whether or not gridpoints should be plotted during calculations.
- `MInusgrid` Subtracts a grid file from the current grid.
- `SAve` Saves the current grid to a GFLOW1 grid file.
- `LOad` Reloads a grid from a GFLOW1 grid file.
- `SUrfer` Saves the current grid in SURFER format.
- `QUit` Returns to the GFLOW1 Main Menu.

Grid Module

DOtmap

Command Description

Directs GFLOW1 whether or not to plot the points at which functions are being calculated. The dot map is a complete layout plot with dots plotted as they are computed. If the grid is to be computed as part of an `INput` file (see the `SWitch` module), the user is advised to turn the `DOtmap` option `OFF`.

Example

- To disable the dot map, use the commands:

  ≥ `GRID <CR>`

  ≥ `DOTMAP OFF <CR>`

- To enable the dot map (default), use the commands:

  ≥ `GRID <CR>`

  ≥ `DOTMAP ON <CR>`

Grid Module

GO

Command Description

Generates a grid according to the window, plot function and horizontal resolution specified (see the `WIndow`, `PLot` and `HOrizontalpoints` command discussions). The computations may be aborted by use of the `Ctrl-Break` keys..

`GO <CR>` may also be executed using the `<F2>` key.

Example

- To generate a grid accepting the current parameter settings, use the commands:

  ≥ `GRID <CR>`

  ≥ `<F2>`

Grid Module

HOrizontalpoints

Command Description

Sets the horizontal resolution for the grid. The specified number of grid points will be generated in the *x* direction. The number of grid points in the *y* direction will be determined by use of the same grid spacing as used in the *x* direction (1:1 aspect ratio for grid cells).

Note

The maximum number of grid points in the *x* or *y* direction is 100.

Examples

- To create a grid with 50 points in the *x* direction, use the commands:

  ≥ GRID <CR>

  ≥ HORIZONTALPOINTS 50 <CR>

  All grids created after these commands will have 50 grid cells in the *x* direction and an appropriate number in the *y* direction for a 1:1 grid cell aspect ratio.

Grid Module

LOad

Command Description

Loads the specified GFLOW1 grid file. The grid is then available for contour plotting, saving as a SURFER file, or for other uses.

GFLOW1 grid files must have the extension `.GRI`. If no extension is provided, the `.GRI` extension is added.

Example

- To load the grid in the file `MODEL.GRI`, use the commands:

  ≥ `GRID <CR>`

  ≥ `LOAD MODEL <CR>`

Grid Module

MInusgrid

Command Description

Subtracts the contents of the specified GFLOW1 grid file (see the SAve command in the GRid module) from the current grid. A grid must already exist, and the current and saved grids must have the same WIndow and HOrizontalpoints settings.

GFLOW1 grid files must have the extension .GRI. If no extension is provided, the .GRI extension is added.

Example

- To subtract the grid in the file PREVIOUS.GRI from the current grid, use the commands:

  ≥ GRID <CR>

  ≥ MINUSGRID PREVIOUS <CR>

  You may now plot the difference grid or export it to a GFLOW1 or SURFER grid file.

Grid Module

PLot

Command Description

Selects a function to be plotted. The parameter to this command selects the function to use when generating the grid. The selected function remains in effect until changed by the user. Available functions:

- `HEad` Compute the head at each grid point (default)
- `STream` Compute the stream function at each grid point.
- `POtential` Compute the discharge potential at each grid point.
- `DIscharge` Compute the magnitude of the discharge vector at each grid point.
- `FLownet` Compute a flownet, using the discharge potential and stream functions at each grid point. The grid will be computed twice; first for the potential and then for the stream function.

Warning - Stream Function

The stream function is defined only for regions governed by Laplace's equation (that is, areas with no areal recharge). The `STream` and `FLownet` options will produce irrelevant results when areal recharge is in use.

Example

- To select the stream function for gridding, use the commands:

```
≥ GRID <CR>
≥ PLOT STREAM <CR>
```

Grid Module

SAve

Command Description

Saves the current grid to the specified GFLOW1 grid file. A grid must already have been generated using the `GO` command, loaded from disk or computed via the `MInusgrid` command.

GFLOW1 grid files must have the extension `.GRI`. If no extension is provided, the `.GRI` extension is added.

Example

- To save the current grid in the file `MODEL.GRI`, use the commands:

  ```
  > GRID <CR>
  > SAVE MODEL <CR>
  ```

Grid Module

SUrfer

Command Description

Saves the current grid to the specified SURFER grid file. A grid must already have been generated using the `GO` command, loaded from disk or computed via the `MInusgrid` command. In addition to the `file.GRD` file, a boundary line file `file.BLN` with all elements and background map features is written to disk.

Since both a grid file and a boundary line file are produced by this command, no file extension should be provided by the user.

Example

- To save the current grid and background map to SURFER files `MODEL.GRD` and `MODEL.BLN`, use the commands:

  ≥ `GRID <CR>`

  ≥ `SURFER MODEL <CR>`

Grid Module

WIndow

Command Description

Sets the window for graphics. This command is functionally identical to the WIndow command in the graphics module. See the LAyout command (Main Menu) for details.

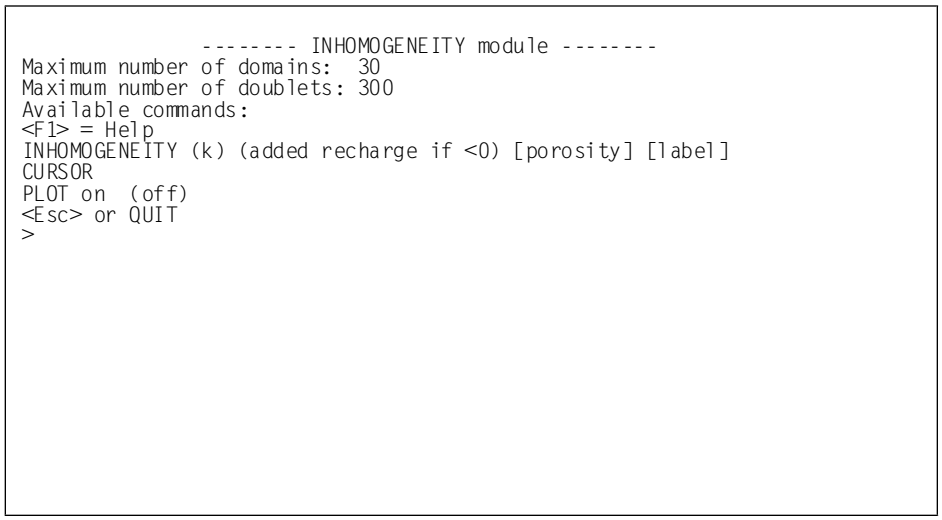


Figure 4.9 - Inhomogeneity Module Menu

The INHOMOGENEITY module provides facilities for the creation and editing of closed aquifer domains with different properties (hydraulic conductivity, porosity and/or recharge) than the rest of the model domain. A cursor function exists for graphical editing of inhomogeneous domains in a model.

Important

Domains with different hydraulic conductivity may be nested, but may not overlap, nor may they have common boundaries.

Note

GAEP does not provide facilities for creation of inhomogeneity domains, but graphical tools for domain creation are provided in the `CUrsor` facility inside this module. This is usually the preferred method for domain creation.

Commands

- `INhomogeneity` Begins creation of an inhomogeneity domain.
- `CUrsor` Enters cursor mode for creation and editing of inhomogeneity domains.
- `PLot` Sets display options for inhomogeneity domains.
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu.

Inhomogeneity Module

CUrsor

Command Description

Selects the graphical editing facility for inhomogeneity domains. The graphics menu (see above description) will be displayed; set the window and type `GO <CR>` to enter the cursor facility. A layout will be shown, and information about inhomogeneities may be requested or modified by placing the mouse cursor (white arrow) at the domain (or point on a domain) of interest and selecting a command (listed below).

Important

When selecting a domain for data retrieval or modification, position the cursor on or near a vertex of the domain before pressing the appropriate key. Positioning the cursor inside a domain may lead to improper domain selection.

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge inhomogeneity.

Cursor Commands

Commands are selected by placing the cursor at a point on the domain of interest and pressing a single key. The available commands are:

- `<CR>` Displays information about the selected domain.
- `R` Change added exfiltration for the domain. GFLOW1 will display the current added exfiltration rate and ask for a new value. See note regarding exfiltration (`INhomogeneity` command in the Inhomogeneity module).
- `P` Change porosity inside the domain. GFLOW1 will display the current porosity and ask for a new value.
- `B` Boundary check. GFLOW1 will check the continuity of the head across the domain boundary at that point and print the % error.
- `C` New color. GFLOW1 will display the current color code (0-15) for graphical display and asks for a new value. A period (`.`) may be entered to display the domain as dotted yellow lines. See the `INhomogeneity` command for color codes.

- `M` Moves a point on a domain. GFLOW1 will ask that the new location for the point be selected. Place the mouse cursor at the new center and press `<CR>`.
- `A` Add a domain (or a point on the domain closest to the mouse cursor). GFLOW1 will prompt whether a domain or point is to be added. To add a point to the selected domain, type `P <CR>` to add a point at the cursor location. To add a new domain, type `D` label `<CR>`, where label is an optional label. If a new domain is added, GFLOW1 will prompt for the hydraulic conductivity, porosity and exfiltration rate of the new domain. Points which enclose the domain are then added by placing the cursor at points and pressing `<CR>`. Press `Q` at the last point of the domain to complete the process (GFLOW1 will close the domain).
- `D` Deletes the selected domain (or a point on the domain nearest the mouse cursor). GFLOW1 will ask whether a point or domain is to be deleted. To delete the nearest point, type `P <CR>`. To delete a domain, type `D <CR>`.
- `<ESC>` Exits the cursor editing facility and returns to the Inhomogeneity Menu.

Other Considerations

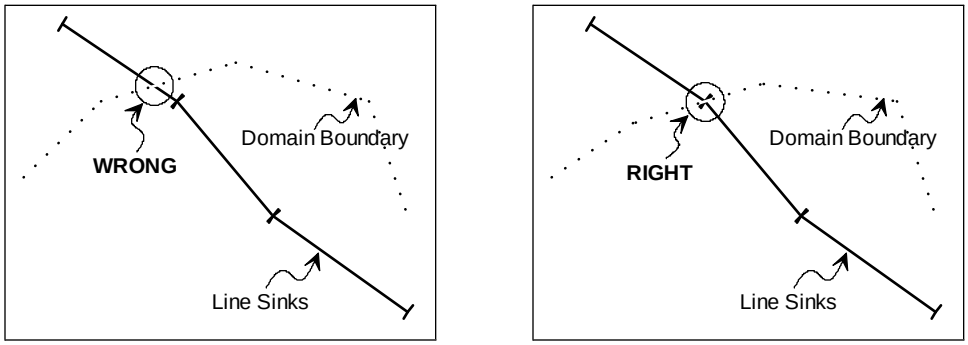
Changes made are not automatically reflected in the user's data file. Use the `DAta` or `SAve` command at the main menu to save changes for future work. Changes in conductivity, recharge or domain geometry will require a new solution (see `SOlve` command, Main Menu).

Ragged Contours - Proper Use of Inhomogeneity Domains

Ragged contours (on piezometric plots) near inhomogeneity domains indicate improperly defined domains. You may have to refine the polygon in areas of much hydraulic "action" (near wells, etc.). Follow the following suggestions for proper use of inhomogeneities:

Crossing Streams

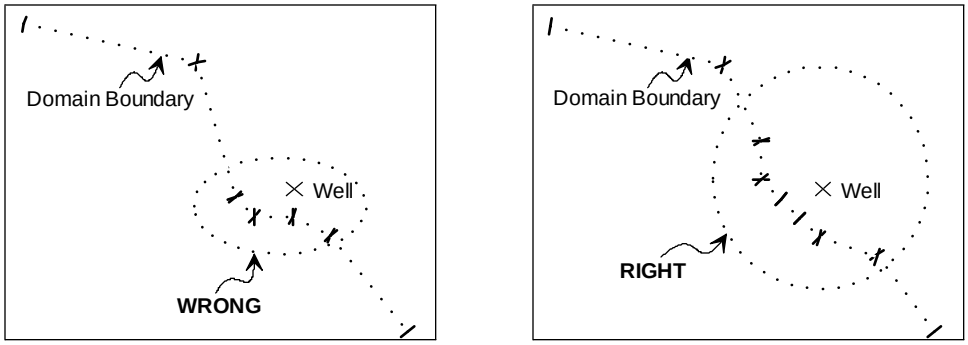
Important: Make sure that each line sink or sink disc is either completely outside or completely inside the domain. The polygon (domain boundary) should be created such that intersections with line sink strings occur at the end points of line sinks (marked by tic-marks in the inhomogeneity editor).



Example: Crossing line sinks with inhomogentity domain boundaries

Length of Polygon Sides

Important: Use smaller line doubles (polygon sides) near streams or wells where you expect strong curvatures in the potentiometric surface. Do not use too drastic a change in the size of consecutive polygon sides; if you wish to change from long sides to short, gradually adjust the lengths of consecutive sides.



Example: Using shorter line doublets near a regoin of strong hydraulic gradients (e.g. near a well)

Inhomogeneity Module

INhomogeneity

Command Description

Begins the process of creating an inhomogeneity domain. The domain's permeability, porosity and additional rate of recharge must be provided. This command may be entered from the keyboard or by use of the `CUrsor` facility (see discussion).

Once the INhomogeneity command is issued, the user enters the *(x,y)* coordinates of the points which make up the domain. Inhomogeneity domains must be closed polygons; the last point is automatically connected to the first point by GFLOW1. An optional label may be specified for each point. The label for the first point (if provided) allows for options:

- If the first character is a period (.) then the domain is shown as dotted yellow lines on graphical displays.
- If the first characters are a one or two digit number, the value is interpreted as the color code for the domain. The following colors are available:
-

0	Not available	8	Dark Grey
1	Blue	9	Bright Blue
2	Green	10	Bright Green
3	Cyan	11	Bright Cyan
4	Red	12	Bright Red
5	Magenta	13	Bright Magenta
6	Brown	14	Yellow
7	Light Grey	15	Bright White

Note

An inhomogeneity domain only adds equations to the matrix solution if the domain is given a hydraulic conductivity different than its surroundings. Each segment in the boundary of a domain with differing hydraulic conductivity than its surrounding domain adds two equations to the solution matrix; domains which change only recharge or porosity add no equations to the matrix.

Note - Exfiltration

All functions used in GFLOW1 are superimposed onto each other. When adding exfiltration to an inhomogeneity domain, the user should be aware that the exfiltration or infiltration is added to that defined by overlapping domains.

Examples

- To create an inhomogeneity "OUTWASH" with hydraulic conductivity of 100 ft/d, an additional recharge of 1 inch/year (2.28e-4 ft/d) and a porosity of 25% the following commands can be entered. The domain will be displayed with dotted lines.

  ≥ INHOMOGENEITY

  (enters the INHOMOGENEITY module)

  ≥ INHOMOGENEITY 100 2.28E-4 0.25 <CR>

  Now, enter the vertices of the polygon; each is given its own unique label (OUTWASH1,etc.)...

  0 0 .OUTWASH1 <CR>

  50 50 .OUTWASH2 <CR>

  ...

  -50 45 .OUTWASH12 <CR>

  The domain is complete. There is no need to repeat the first point; the domain is closed atuomatically by GFLOW1. Exit the INHOMOGENEITY module.

  QUIT <CR>

Important

- When a domain is entered with only a porosity or recharge which differs from the surrounding values, the conductivity parameter must be set equal to that of the surrounding domain. If at some point the surrounding hydraulic conductivity is to be changed, the user should remember to also change the conductivity value inside the domain.
- After creating or editing inhomogeneities, it is good practice to enter the CUrsor module at the Main Menu and verify the local values of the hydraulic conductivity, recharge and porosity.

Inhomogeneity Module

PLot

Command Description

Sets plotting options for inhomogeneities. The user may select whether to display inhomogeneity domains or disable plotting.

The option, once set, remains in force for all graphical displays until changed by a later `PLot` command.

Examples

To plot all inhomogeneity domains, use the commands

```
> INHOMOGENEITY <CR>
> PLOT ON <CR>
```

To disable plotting of inhomogeneity domains, use the commands

```
> INHOMOGENEITY <CR>
> PLOT OFF <CR>
```

This page is intentionally left blank.

```
                    -------- LINE SINK module --------
Maximum number of line sinks: 800
Maximum number of streams: 500
Available commands:
<F1> = Help
HEAD
DISCHARGE
STREAM [headwater inflow L^3/T]
WIDTH   1.000
RESISTANCE  0.0000
DEPTH  0.0000
CURSOR
HIGHLIGHT NONE (recharging/percolating)
PLOT LAYOUT (baseflow/overlandflow/streamflow)
<Esc> or QUIT
>
```

Figure 4.10 - Line Sink Module Menu

The LINESINK module provides facilities for the creation and editing of LINESINK analytic elements. A cursor function is available for graphical editing of line sinks in a model. Line sinks can be created as "head specified", "discharge specified" or as part of a stream network.

Commands

- `HEad` Initiates creation of head specified linesinks.
- `DIscharge` Initiates creation of discharge specified linesinks.
- `STream` Initiates creation of head specified stream networks which use the surface water/groundwater interaction facility..
- `WIdth` Sets a default width for successive linesink definitions.
- `REsistance` Sets a default resistance for successive linesink definitions.
- `DEpth` Sets a default value for the depth of the bottom of the resistance layer below the water table for successive line sinks.
- `CUrsor` Enters cursor mode for editing of linesinks.
- `HIghlight` Sets highlight options for linesinks.

- `PLot` Sets display options for linesinks.
- `QUit` Returns the the GFLOW1 Main Menu.

Linesink Module

CUrsor

Command Description

Selects the graphical editing facility for linesink elements. The graphics menu (see LAyout command in the Main Menu) will be displayed; set the window and press <F2> to enter the cursor facility. A layout of all linesinks in the current graphics window will be shown, and information about linesinks may be requested or modified by placing the mouse cursor (white arrow) at the linesink of interest and selecting a command (listed below).

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge linesink or losing stream.

Cursor Commands

Commands are selected by placing the cursor at the linesink of interest and pressing a single key. The available commands are:

- <CR> Displays information about the selected linesink.
- Q Change discharge (discharge specified linesinks only). GFLOW1 will display the current discharge rate and ask for a new value.
- H Change head (head specified linesinks only). GFLOW1 will display the current head and ask for a new value.
- R Change resistance. GFLOW1 will display the current resistance and ask for a new value.
- D Change resistance layer depth. GFLOW1 will display the current depth and ask for a new value. Depth: distance between the water table in a stream and the bottom of the resistance layer.
- W Change width. GFLOW1 will display the current width and ask for a new value.
- C Check on boundary conditions: reports percentage error of heads or discharges at head specified line sinks.

- S — Write a spreadsheet file containing information about the selected stream reach (highlighted on the screen). Note that this function may only be performed for linesinks that are part of a stream network (see the STream command).
- P — This command is analogous to the PLOT command in the main LINESINK menu. After pressing <P>, GFLOW1 prompts for the line sink plotting option (LAyout, BAseflow, OVerlandflow or STreamflow). The plot option specified will be applied to all line sink features, regardless of the cursor positon. The plot mode selected remains in effect until the P key is pressed or a PLot command is entered at the LINESINK menu. See the PLot command discussion for additional details.
- F — Print the flows for the stream selected. Note that this function may only be performed for linesinks that are part of a stream network (see the STream command).
- <ESC> — Exits the cursor editing facility and returns to the Linesink Menu.

Other Considerations

Changes made are not automatically reflected in the user's data file. Use the DAta or SAve command at the main menu to save changes for future work.

Linesink Module

DEpth

Command Description

Sets the default resistance layer depth for linesink element creation. Once set, all following linesinks created will have this depth unless a depth is specified as part of the linesink definition (see `HEad`, `DIscharge` or `STream` commands). The depth parameter is defined as the distance between the water table in the stream and the bottom of the resistance layer beneath the stream.

Note: Use of resistance specified line sinks

If resistance is used on any head-specified linesink, a depth for the resistance layer MUST be specified. Improper results will occur if a zero depth is specified for a linesink with resistance. The discharge for resistance specified line sinks is computed according to the difference in the specified head at the line sink and the head in the aquifer beneath the line sink, and distributed over the rectangular region delineated by the line sink's width and length (limited by available stream flow when part of a stream network).

Note: Percolating line sinks

If the hydraulic head beneath a line sink falls below the elevation of the bottom of its resistance layer, the line sink is considered to be "percolating". Percolating line sinks are removed from the solution matrix and their discharges computed according to the leakage through a resistance layer, as shown below:

- If a resistance layer is specified (resistance $\neq 0$), the discharge is limited by the rate at which water can travel through the resistance layer based upon the water level in the stream and assuming atmospheric pressure beneath the resistance layer (and by available stream flow when part of a stream network).
- If no resistance layer is specified (resistance $= 0$), the discharge is limited to the saturated hydraulic conductivity of the aquifer (and by available stream flow when part of a stream network). This is the steady state infiltration rate under conditions of incipient ponding.

The discharge limits for percolating conditions may be disabled by the user by specifying a line sink width of zero (0.0). This will be especially useful for "far-field" boundary conditions, when the specified head condition at the linesink must be met regardless of the required infiltration rate. Note: a width of zero may only be assigned to a line sink with no resistance.

Please see Appendix A for a detailed discussion of GFLOW1's surface water/groundwater interaction modeling facilities.

Examples

To set the default resistance layer depth to 3 feet use the following commands:

> LINESINK <CR>

> DEPTH 3 <CR>

Unless overridden as part of a linesink definition, all future linesinks will have a 3 foot depth.

Linesink Module

DIscharge

Command Description

Begins the process of creating linesink elements which are discharge specified. A discharge specified line sink adds no equations to the solution matrix. The user will be prompted for the *(x,y)* locations of the ends of the linesink and the average discharge (sink density) over the length of the linesink. An optional label may be specified for each linesink.

Discharge specified linesinks may be used to model so-called "French Drains" with known groundwater extraction rates. They may also be used to model losing stream sections, provided the modeler has insight into the aquifer recharge rate along the section (use a negative line sink strength to represent recharge).

Examples

- To create a discharge specified linesink "LINE1" with ends at (50,50) and (200,50) and with a discharge rate of 100 ft^3/ft/day, enter the following commands from the main menu:

```
≥ LINESINK <CR>
≥ DISCHARGE <CR>
50 50 200 50 100 LINE1 <CR>
≥ QUIT <CR>
```

Linesink Module

HEad

Command Description

Begins the process of creating linesink elements which are head specified. When the model is `SOlved`, a strength parameter (sink density) will be computed for all head specified linesinks. Each head specified linesink adds one equation to the solution matrix. The user will be prompted for the *(x,y)* locations of the ends of the linesink, the head at the center of the linesink and (optionally) the resistance, width and resistance layer depth of the linesink. Default values for resistance, width and resistance layer depth are used if not specified. An optional label may be specified for each linesink.

Note: Use of resistance specified line sinks

If resistance is used on any head-specified linesink, a depth for the resistance layer MUST be specified. Improper results will occur if a zero depth is specified for a linesink with resistance. The discharge for resistance specified line sinks is computed according to the difference in the specified head at the line sink and the head in the aquifer beneath the line sink, and distributed over the rectangular region delineated by the line sink's width and length (limited by available stream flow when part of a stream network).

Note: Percolating line sinks

If the hydraulic head beneath a line sink falls below the elevation of the bottom of its resistance layer, the line sink is considered to be "percolating". Percolating line sinks are removed from the solution matrix and their discharges computed according to the leakage through a resistance layer, as shown below:

- If a resistance layer is specified (resistance $\neq 0$), the discharge is limited by the rate at which water can travel through the resistance layer based upon the water level in the stream and assuming atmospheric pressure beneath the resistance layer (and by available stream flow when part of a stream network).
- If no resistance layer is specified (resistance $= 0$), the discharge is limited to the saturated hydraulic conductivity of the aquifer (and by available stream flow when part of a stream network). This is the steady state infiltration rate under conditions of incipient ponding.

The discharge limits for percolating conditions may be disabled by the user by specifying a line sink width of zero (0.0). This will be especially useful for "far-field" boundary conditions, when the specified head condition at the

linesink must be met regardless of the required infiltration rate. Note: a width of zero may only be assigned to a line sink with no resistance.

Please see Appendix A for a detailed discussion of GFLOW1's surface water/groundwater interaction modeling facilities.

Examples

- To create a head specified linesink "LINE1" with ends at (50,50) and (200,50) and with a head of 505 feet and other values defaulted, enter the following commands from the main menu:

```
> LINESINK <CR>
> HEAD <CR>
50 50 200 50 505 LINE1 <CR>
> QUIT <CR>
```

- To create a head specified linesink "LINE1" with ends at (50,50) and (200,50) and with a head of 505 feet, width of 20 feet, resistance of 1 day and resistance layer depth of 5 feet, enter the following commands from the main menu:

```
> LINESINK <CR>
> DISCHARGE <CR>
50 50 200 50 505 20 1 5 <CR>
> QUIT <CR>
```

Considerations in Line Sink Creation

- It is recommended that a line sink's width should never be larger than 30% of its sink length (see also the `WIdth` command).
- The resistance between the stream and aquifer is defined as the thickness of the resistance layer divided by its hydraulic conductivity.
- The resistance layer depth is defined as the distance between the (average) water table in the stream and the bottom of the resistance layer.

See Also

- `WIdth`, `REsistance` and `DEpth` commands (Linesink module)

Linesink Module

HIghlight

Command Description

Sets the highlight display option for linesinks. This command allows the user to display linesinks which are either recharging the aquifer or percolating in white (instead of the normal yellow) for visual analysis on screen displays.

The option, once set, remains in force for all graphical displays until changed by a later `HIghlight` command.

Examples

To highlight recharging linesinks on graphical displays, use the commands

```
≥ LINESINK <CR>
≥ HIGHLIGHT RECHARGE <CR>
```

To highlight percolating linesinks, use the commands

```
≥ LINESINK <CR>
≥ HIGHLIGHT PERCOLATING <CR>
```

To turn off the highlight feature,

```
≥ LINESINK <CR>
≥ HIGHLIGHT NONE <CR>
```

Note

Line sinks in which the strength is zero (e.g. those turned off by the surface water solution due to a lack of water to infiltrate) are plotted as dashed lines.

PLot

Command Description

Sets plotting options for linesinks which are part of a stream network (entered with the `STream` command). Linesinks may be shown as simply a layout of their locations or with widths in proportion to their baseflow, overland flow or total streamflow.

The option, once set, remains in force for all graphical displays until changed by a later `PLot` command.

Note

Linesinks which are not part of stream networks (entered with the `HEad` or `DIscharge` commands) will not appear on graphical displays unless the `PLot Layout` option is selected (see below).

Also Note

Linesinks which are part of a stream network (entered with `STream` command) and that have been removed from the solution (no water available for infiltration) are shown as dashed lines, regardless of other `PLot` settings.

Examples

To plot linesinks as a simple layout, use the commands

```
≥ LINESINK <CR>
≥ PLOT LAYOUT <CR>
```

To plot linesinks associated via the `STream` command with widths in proportion to the baseflow at the linesink, use the commands

```
≥ LINESINK <CR>
≥ PLOT BASEFLOW <CR>
```

To plot linesinks associated via the `STream` command with widths in proportion to the overland flow at the linesink, use the commands

```
≥ LINESINK <CR>
≥ PLOT OVERLANDFLOW <CR>
```

To plot linesinks associated via the `STream` command with widths in proportion to the total stream flow at the linesink, use the commands

```
≥ LINESINK <CR>
≥ PLOT STREAMFLOW <CR>
```

Linesink Module

REsistance

Command Description

Sets the default resistance for linesink element creation. Once set, all following linesinks created will have this resistance unless a resistance is specified as part of the linesink definition (see `HEad`, `DIscharge` or `STream` commands). The resistance between the stream and the aquifer is defined as the thickness of the resistance layer divided by its hydraulic conductivity.

Note: Use of resistance specified line sinks

If resistance is used on any head-specified linesink, a depth for the resistance layer MUST be specified. Improper results will occur if a zero depth is specified for a linesink with resistance. The discharge for resistance specified line sinks is computed according to the difference in the specified head at the line sink and the head in the aquifer beneath the line sink, and distributed over the rectangular region delineated by the line sink's width and length (limited by available stream flow when part of a stream network).

Note: Percolating line sinks

If the hydraulic head beneath a linesink falls below the elevation at the bottom of its resistance layer, the linesink is considered to be "percolating". Percolating line sinks are removed from the solution matrix and their discharges computed according to the leakage through a resistance layer, as shown below:

- If a resistance layer is specified (resistance $\neq 0$), the discharge is limited by the rate at which water can travel through the resistance layer based upon the water level in the stream and assuming atmospheric pressure beneath the resistance layer (and by available stream flow when part of a stream network).
- If no resistance is specified (resistance $= 0$), the discharge is limited to the saturated hydraulic conductivity of the aquifer (and by available stream flow when part of a stream network). This is the steady state infiltration rate under conditions of incipient ponding.

The discharge limits for percolating conditions may be disabled by the user by specifying a line sink width of zero (0.0). This will be especially useful for "far-field" boundary conditions, when the specified head condition at the linesink must be met regardless of the required infiltration rate. Note: a width of zero may only be assigned to a line sink with no resistance.

Please see Appendix A for a detailed discussion of GFLOW1's surface water/groundwater interaction modeling facilities.

Examples

To set the default resistance to 5 days use the following commands:

> LINESINK <CR>

> RESISTANCE 5 <CR>

Unless overridden as part of a linesink definition or a new REsistance command, all future linesinks will have a 5 day resistance.

Linesink Module

STream

Command Description

Begins the process of creating linesink elements for use with the surface/groundwater interaction facility. When the model is `SOlved`, a strength parameter will be computed for these linesinks. Conjunctively, a streamflow analysis will be carried out to assess whether or not losing stream sections can infiltrate the amount of water estimated by the groundwater flow solution. Iterating between groundwater flow solutions and surface water flow solutions leads to a final (steady state) solution for both streamflow and groundwater flow. The user will be prompted for the *(x,y)* locations of the ends, the heads at the centers and (optionally) the resistance, width and resistance layer depth for each linesink making up the stream. If not specified, default values for resistance, width and resistance layer depth are used. In addition, the entire stream can have water added from the upstream end and overland flow added along the stream. An optional label may be specified for each linesink.

Note

A low resistance (or no resistance) between a stream and the aquifer may occasionally result in slow convergence of the conjunctive surface water and groundwater solution. When resistance is used on any linesink, a depth for the resistance layer MUST be specified. Improper results will occur if a zero depth is specified for a linesink with resistance.

Options

- The `STream` command has an optional parameter, the "headwater inflow" (in units of length^3/time), which may be used to add streamflow from headwaters not included in the model. If the stream is not a "headwater" stream ina network, its "headwater inflow" will be ignored.
- At any time during stream definition, an `OVERLANDFLOW` value may be specified. This should be interpreted as a "steady-state" overland flow which is received at the stream - not the average runoff to the stream. It should be entered as a total rate of inflow (in units of length^3/time) to the stream; GFLOW1 will evenly distribute it over the stream length.

  Note: "Overland flow" in GFLOW1 may be viewed as groundwater seeping into the stream from low permeability surface formations which overlie the regional aquifer. Hortonian overland flow is generally too transient to be of importance to the steady state (read: "average conditions") stream flow.
- Normally, each stream section (set of line sinks defined in a `STream` command) will be connected to another stream section. The `ENd` command

may be added to a stream section to indicate that it ends and should not be connected to another stream. A stream network should have at least one stream section with an ENd command.

- You cannot include discharge specified linesinks in a stream.
- The REsistance, WIdth and DEpth commands may be used to change default values while creating a stream section.

Examples

- To create a stream section with no headwater inflow, nor overland flow and which should be connected to another stream section, enter the following commands from the main menu:

```
> LINESINK <CR>
> STREAM <CR>
50 50 505 LINE1 <CR>
75 100 506 LINE2 <CR>
...
1000 1050 <CR>
> QUIT <CR>
```

Note

Following the STream command, line sinks are defined by the coordinate pairs on successive lines from the input file. The head specified at the center of the line sink is entered after the first coordinate pair. A stream section (line sink string) therefore should always end with a pair of end coordinates without a given head (1000 1050 <CR> in the example).

- To create a stream section with 500 ft^3/day headwater inflow, an overland flow of 1000 ft^3/day and which should not be connected to another stream section, enter the following commands from the main menu:

```
> LINESINK <CR>
> STREAM 500 <CR>
OVERLANDFLOW 1000 <CR>
END <CR>
50 50 505 LINE1 <CR>
75 100 506 LINE2 <CR>
...
1000 1050 <CR>
```

```
≥ QUIT <CR>
```

- To change the resistance for the second line sink, specify a resistance layer depth and enter the desired head, width and resistance for each linesink:

```
≥ LINESINK <CR>
≥ STREAM 500 <CR>
DEPTH 5 <CR>
OVERLANDFLOW 1000 <CR>
END <CR>
50 50 505 10 1.0 LINE1 <CR>
75 100 506 10 1.5 LINE2 <CR>
...
```

where 10 is the resistance in days and 1.5 is the resistance layer depth.

Considerations in Line Sink Creation

- The line sink width should never be larger than 30% of the line sink length. The resistance between the stream and aquifer is defined as the thickness of the resistance layer divided by its hydraulic conductivity.
- The resistance layer depth is defined as the distance between the (average) water table in the stream and the bottom of the resistance layer.

See Also

`WIdth`, `REsistance` and `DEpth` commands (Linesink module)

WIdth

Command Description

Sets the default width for linesink element creation. Once set, all following linesinks created will have this width unless a width is specified as part of the linesink definition (see `HEad`, `DIscharge` or `STream` commands), or a new `WIdth` command is given.

Note: Use of resistance specified line sinks

If resistance is used on any head-specified linesink, a depth for the resistance layer MUST be specified. Improper results will occur if a zero depth is specified for a linesink with resistance. The discharge for resistance specified line sinks is computed according to the difference in the specified head at the line sink and the head in the aquifer beneath the line sink, and distributed over the rectangular region delineated by the line sink's width and length (limited by available stream flow when part of a stream network).

Note: Percolating line sinks

If the hydraulic head beneath a line sink falls below the elevation of the bottom of its resistance layer, the line sink is considered to be "percolating". Percolating line sinks are removed from the solution matrix and their discharges computed according to the leakage through a resistance layer, as shown below:

- If a resistance layer is specified ($\text{resistance} \neq 0$), the discharge is limited by the rate at which water can travel through the resistance layer based upon the water level in the stream and assuming atmospheric pressure beneath the resistance layer (and by available stream flow when part of a stream network).
- If no resistance is specified ($\text{resistance} = 0$), the discharge is limited to the saturated hydraulic conductivity of the aquifer (and by available stream flow when part of a stream network). This is the steady state infiltration rate under conditions of incipient ponding.

The discharge limits for percolating conditions may be disabled by the user by specifying a line sink width of zero (0.0). This will be especially useful for "far-field" boundary conditions, when the specified head condition at the linesink must be met regardless of the required infiltration rate. Note: a width of zero may only be assigned to a line sink with no resistance.

Please see Appendix A for a detailed discussion of GFLOW1's surface water/groundwater interaction modeling facilities.

Examples

To set the default width to 10 feet use the following commands:

```
> LINESINK <CR>
> WIDTH 10 <CR>
```

Unless overridden as part of a linesink definition or a new `WIdth` command, all future linesinks will have a 10 foot width.

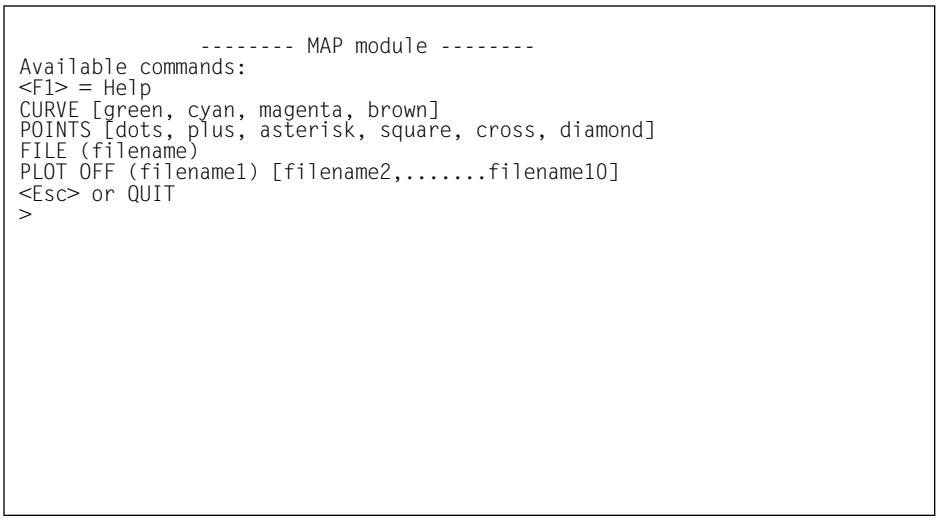


Figure 4.11 - Map Module Menu

The MAP module provides facilities for the creation and management of background maps which show the locations of roads, political boundaries and other features not used for creating groundwater flow solutions.

The user creates binary background map (`.MAP`) files by using the commands `FIle`, `CUrve` and `POints`, then tells GFLOW1 which of the `.MAP` files are to be displayed.

During any GFLOW1 graphics operation, all `.MAP` files specified by the `MAp` module `PLot` command are read from disk and plotted on the screen. The use of disk-based `.MAP` files provides almost unlimited capabilities for adding background features to plots of analytic elements and piezometric contours or streamlines.

Commands

- `CUrve` Creates a background map curve.
- `POints` Creates individual points in a background map.
- `FIle` Creates a background map (.MAP) file.
- `PLot` Tells GFLOW1 which background map file(s) are to be displayed.
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu.

Map Module

CUrve

Command Description

Creates a curve in a background map. The output .MAP file must have been specified prior to issuance of the CUrve command (see FIle command description). Background map curves may be created in any of four colors: green, cyan, magenta or brown. A single background map file may contain several different curves or groups of points (see also POINTS command).

All vertices on the curve to be created should be in (*x*,*y*) coordinates relative to the same origin used to create analytic elements.

Note

Simply creating the .MAP file does not result in its being displayed in graphical screens. The .MAP file must be specified in a MAp module PLot command if it is to be displayed.

Example

The user wishes to create a background map containing two curves and put the resulting binary background map in the file CURVES.MAP. The following commands can be used, either from the keyboard or in a SWitch file:

```
> MAP <CR>
> FILE CURVES <CR>
```

At this point, the file may already exist; to overwrite, say yes ("Y")

```
> Y <CR>
```

When creating a SWitch file which creates a .MAP file, you should always include this Y command. If the file does not exist, the Y command will be ignored. To create a curve in the color cyan:

```
> CURVE CYAN <CR>
```

Now enter vertices for the curve...

```
0 0 <CR>
100 150 <CR>
...
```

To end this curve, simply give a new command. Begin the second curve in green...

```
CURVE GREEN <CR>
100 100 <CR>
...
```

All curves are complete. To leave the MAP module and close CURVES.MAP, issue the command

≥ QUIT <CR>

or press the <ESC> key.

See Also

POints and FIle commands (MAp module)

Map Module

FIle

Command Description

Creates a binary background map file. The output file will be given the extension `.MAP`; no extension needs to be specified. Once a background map file is created, use the `CUrve` or `POints` commands in the MAP module to create background map features for display.

Note

Simply creating the `.MAP` file does not result in its being displayed in graphical screens. The `.MAP` file must be specified in a `MAp` module `PLot` command if it is to be displayed.

Example

The user wishes to create a background map file `CITY.MAP`. The following commands must be issued prior to entering the `CUrve` or `POints` commands:

```
≥ MAP <CR>
≥ FILE CITY.MAP
```

or

```
≥ FILE CITY
```

If the file already exists, GFLOW1 will ask if you wish to overwrite it. When creating a `SWitch` file which creates a `.MAP` file, you should always include this `Y` command. If the file does not exist, the `Y` command will be ignored.

```
Y <CR>
```

You are now ready to proceed with creation of map features.

Map Module

POints

Command Description

Creates a set of points in a background map. The output `.MAP` file must have been specified prior to issuance of the `POints` command (see `FIle` command description). Background map points may have several shapes: dots, plus, asterisk, square, cross or diamond. A single background map file may contain several different curves or groups of points (see also `CUrve` command).

The coordinates of all points to be created should be in (*x*,*y*) coordinates relative to the same origin used to create analytic elements.

Note

Simply creating the `.MAP` file does not result in its being displayed in graphical screens. The `.MAP` file must be specified in a `MAp` module `PLot` command if it is to be displayed.

Example

The user wishes to create a background map containing two sets of points and put the resulting binary background map in the file `POINTS.MAP`. The following commands can be used, either from the keyboard or in a `SWitch` file:

```
> MAP <CR>
> FILE POINTS <CR>
```

At this point, the file may already exist, to overwrite, say yes ("Y")

```
> Y <CR>
```

When creating a `SWitch` file which creates a `.MAP` file, you should always include this `Y` command. If the file does not exist, the `Y` command will be ignored. To create a points which appear as "plus signs":

```
> POINTS PLUS <CR>
```

Now enter the points...

```
0 0 <CR>
100 150 <CR>
...
```

To end this curve, simply give a new command. Begin the second group of points as diamonds...

```
POINTS DIAMOND <CR>
100 100 <CR>
...
```

All sets of points are complete. To leave the MAP module and close POINTS.MAP, issue the command

≥ QUIT <CR>

or press the <ESC> key.

See Also

CUrve and FIle commands (MAp module)

Map Module

PLot

Command Description

Instructs GFLOW1 as to which background map (`.MAP`) files are to be displayed, if any. Two versions of the command exist; one selects files to be plotted and the other turns off background map plotting.

After background maps are selected, GFLOW1 will display the background map information each time the user enters graphics mode (for example, on a contour plot).

Example

The user wishes to display the background map file `CITY.MAP` as part of the background map. The file `CITY.MAP` already exists on disk. The commands

```
≥ MAP <CR>
≥ PLOT CITY.MAP
```

or

```
≥ PLOT CITY
```

turn on the plotting of the file `CITY.MAP`. GFLOW1 will list `CITY.MAP` while in the MAP module and superimpose curves and points stored in `CITY.MAP` onto all graphics. GFLOW1 will check for the existance of the specified file and issues and error message is the file is not found..

To turn off background map plotting, use the following commands:

```
≥ MAP <CR>
≥ PLOT OFF <CR>
```

Note

The command `PLOT OFF` is a special case of the `PLOT` command. Do not attempt to use a background map file called `OFF.MAP`.

This page is intentionally left blank.

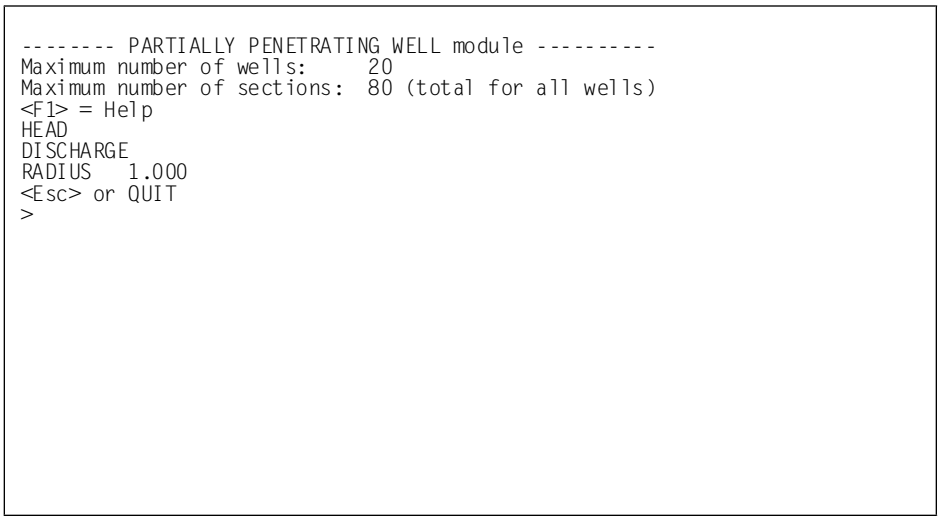


Figure 4.12 - Partially Penetrating Well Module Menu

The PPWELL module provides facilities for the creation of three-dimensional partially penetrating well analytic elements. Partially penetrating wells can only be used in regions where confined flow conditions exist.

Partially penetrating wells may be head specified or discharge specified.

Important

It is the modeler's responsibility to ensure that flow conditions near the partially penetrating well are confined at all times.

Commands

- `HEad` Begins creation of head specified partially penetrating wells.
- `DIscharge` Begins creation of discharge specified partially penetrating wells.
- `RAdius` Sets a default radius for successive partially penetrating wells.
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu.

PPwell Module

DIscharge

Command Description

Begins the process of creating partially penetrating well elements which are discharge specified. Two (2)equations will be added to the matrix. The user will be prompted for the well location *(x,y)*, the elevations of the top and bottom of the well (z_1 and z_2) the discharge rate of the well and (optionally) the radius of the well. If no radius is specified, the default radius (see `RAdius` command) will be used. An optional label may be specified for each well.

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge well or losing stream.

Examples

- To create a partially penetrating pumping well "PUMP1" at the location (50,50) with bottom and top elevations at 350 and 360 feet which pumps 100,000 ft^3/day using the default radius, enter the following commands from the main menu:

```
> PPWELL <CR>
> DISCHARGE <CR>
50 50 350 360 100000 WELL1 <CR>
> QUIT <CR>
```

- To create a partially penetrating recharge well "RECH2" at the location (150,150) with bottom and top elevations at 330 and 340 feet which injects 50,000 ft^3/day using a radius of 0.5 feet, enter the following commands from the main menu:

```
> PPWELL <CR>
> DISCHARGE <CR>
150 150 330 340 -50000 0.5 WELL1 <CR>
> QUIT <CR>
```

PPwell Module

HEad

Command Description

Begins the process of creating partially penetrating well elements which are head specified. When the model is `SOlved`, GFLOW1 will solve for the discharge of head specified wells. A head specified partially penetrating well adds 3 equations to the matrix. The user will be prompted for the well location *(x,y)*, the elevations of the top and bottom of the well (z_1 and z_2) the head at the well and (optionally) the radius of the well. If no radius is specified, the default radius (see `RAdius` command) will be used. An optional label may be specified for each well.

Examples

- To create a head specified partially penetrating well "WELL1" at the location (50,50) with bottom and top elevations at 350 and 360 feet and with head 405 feet and the default radius, enter the following commands from the main menu:

```
≥ PPWELL <CR>
≥ HEAD <CR>
50 50 350 360 405 WELL1 <CR>
≥ QUIT <CR>
```

- To create a head specified partially penetrating well "WELL2" at the location (150,150) with bottom and top elevations at 330 and 340 feet and with head 405 feet and the radius 0.5 feet, enter the following commands from the main menu:

```
≥ PPWELL <CR>
≥ HEAD <CR>
150 150 330 340 405 0.5 WELL1 <CR>
≥ QUIT <CR>
```

PPwell Module

RAdius

Command Description

Sets the default radius for partially penetrating well element creation. Once set, all following partially penetrating wells created will have this radius unless a radius is specified as part of the partially penetrating well definition (see `HEad` or `DIscharge` commands).

Examples

To set the default partially penetrating well radius to 6 inches (0.5 feet) use the following commands:

```
> PPWELL <CR>
> RADIUS 0.5 <CR>
```

Unless overridden as part of a partially penetrating well definition, all future partially penetrating wells will have a 0.5-foot radius.

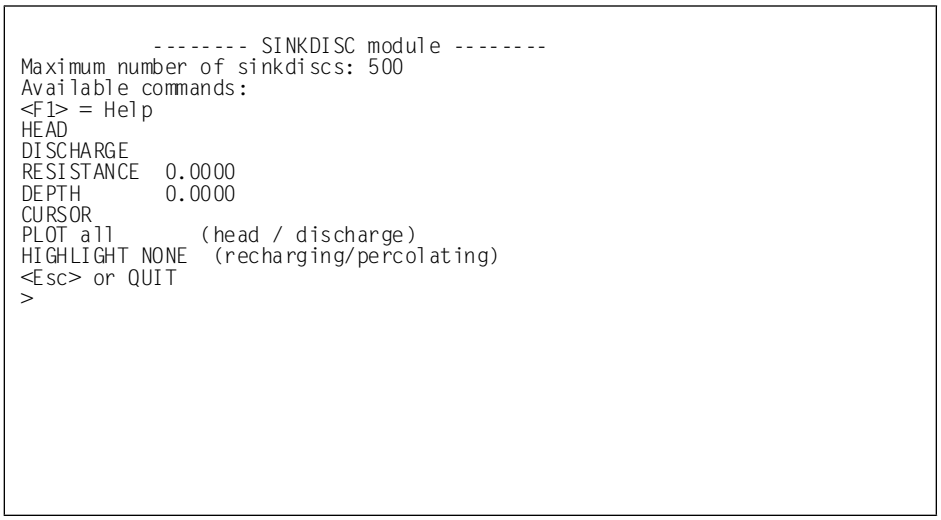


Figure 4.13 - Sinkdisc Module Menu

The SINKDISC module provides facilities for the creation and editing of disc-shaped sink distributions. A cursor function exists for graphical editing of sink discs in a model. Sink discs can be created as "head specified" or "discharge specified". Discharge specified sink discs can have a discharge either at the aquifer top, the aquifer bottom or both.

Note

The word SINKDISC (instead of "disc sink") has been chosen to ensure that all GFLOW1 commands may be uniquely abbreviated to their first two letters.

Commands

- `HEad` Initiates creation of head specified sink discs.
- `DIscharge` Initiates creation of discharge specified sink discs.
- `REsistance` Sets a default resistance for successive sink disc definitions.
- `DEpth` Sets a default resistance layer depth for successive sink disc definitions.
- `CUrsor` Enters cursor mode for editing of sink discs.
- `PLot` Sets display options for sink discs.
- `HIghlight` Sets highlight options for sink discs.

- `QUit or <ESC>` Returns the the GFLOW1 Main Menu.

Sinkdisc Module

CUrsor

Command Description

Selects the graphical editing facility for sink disc elements. The graphics menu (see above description) will be displayed; set the window and type GO <CR> to enter the cursor facility. A layout of all sink discs will be shown, and information about sink discs may be requested or modified by placing the mouse cursor (white arrow) at the sink disc of interest and selecting a command (listed below).

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge sink disc or losing stream.

Cursor Commands

Commands are selected by placing the cursor at the sink disc of interest and pressing a single key. The available commands are:

- <CR> Displays information about the selected sink disc.
- Q Change discharge (discharge specified sink discs only). GFLOW1 will display the current discharge rate (length/time) and ask for a new value.
- H Change head (head specified sink discs only). GFLOW1 will display the current head and ask for a new value.
- R Change resistance. GFLOW1 will display the current resistance and ask for a new value.
- T Change resistance layer depth. GFLOW1 will display the current depth and ask for a new value.
- C Checks the accuracy of the boundary condition, comparing the actual head at the sink disc center with the specified head. Available for head specified sink discs only.
- M Moves the sink disc. GFLOW1 will ask that the new location be selected. Place the mouse cursor at the new center and press <CR>.

- `A` Add a sink disc. GFLOW1 will request that the center be specified. Place the cursor at the center and press `<CR>`. Next, GFLOW1 requests a point on the edge; place the cursor at an arbitrary point through which the circle should go and press `<CR>`.
- `D` Deletes the selected sink disc.
- `<ESC>` Exits the cursor editing facility and returns to the Sinkdisc Menu.

Other Considerations

Changes made are not automatically reflected in the user's data file. Use the `DAta` and/or `SAve` command at the main menu to save changes for future work. After changes in sink disc parameters, a new solution is required. See the `SOlve` command (Main Menu).

Sinkdisc Module

DEpth

Command Description

Sets the default resistance layer depth (in units of length) for sink disc element creation. Once set, all following sink discs created will have this depth unless a depth is specified as part of the sink disc definition (see `HEad` command). The depth of a sink disc is defined as the distance between the surface water level and the bottom of the resistance layer.

Note

This parameter MUST be specified if resistance is used, or improper results will occur (GFLOW1 issues a warning if a resistance is specified with a depth of 0). If the hydraulic head at a linesink falls below the resistance layer, the linesink is considered to be percolating. The discharge of a percolating linesink is limited by the rate at which water can travel through the resistance layer.

Examples

To set the default resistance layer depth to 3 feet use the following commands:

```
> SINKDISC <CR>
> DEPTH 3 <CR>
```

Unless overridden as part of a sink disc definition, all future sink discs will have a depth of 3 feet.

Sinkdisc Module

DIscharge

Command Description

Begins the process of creating sink discs which are discharge specified. The user will be prompted for the *(x,y)* locations of the center of the and an arbitrary point on the edge of the sink disc and the average discharge (in units of length/time) over the area of the sink disc at the top and bottom of the aquifer. An optional label may be specified for each sink disc.

A common use for a discharge specified sink disc is to create a regional areal recharge ("rain") element by positioning a sink disc with negative sink density (recharging) over the entire model area.

Note: Bottom discharging sink discs

GFLOW1 allows two discharge rates to be specified for discharge specified sink discs. The first discharge is the discharge (extraction rate) at the top of the aquifer, while the second discharge is the extraction rate at the bottom of the aquifer. The bottom discharge may be used to specify leakage to a lower aquifer. Since GFLOW1 is based upon the Dupuit-Forschheimer assumption, the use of top or bottom sink discs does not affect the regional groundwater flow solution. The choice of "top or bottom" only affects the computation of the vertical velocity during streamline tracing (see the TRace module).

Examples

- To add a recharge rate due to precipitation to the model area, the following sink disc may be introduced:

```
> SINKDISC <CR>
> DISCHARGE <CR>
250 300 20000 300 -0.000228 0 RAINELEMENT <CR>
> QUIT <CR>
```

  The sink disc strength of -0.000228 ft/day represents 1 inch per year and is the first discharge. It occurs at the aquifer top.

- To create a leakage element representing leakage of 0.001 ft/day to a lower aquifer, use a discharge specified disc sink "DISK1" with its center at (50,50) and a point on its edge at (200,50):

```
> SINKDISC <CR>
> DISCHARGE <CR>
50 50 200 50 0.0 0.001 DISK1 <CR>
```

```
≥ QUIT <CR>
```

The extraction rate of 0.001 ft/day is the second discharge and occurs at the aquifer bottom.

Sinkdisc Module

HEad

Command Description

Begins the process of creating sink discs which are head specified. When the model is SOlved, a strength parameter (sink density) will be computed for all head specified sink discs. Each sink disc element adds one equation to the solution matrix. The user will be prompted for the *(x,y)* locations of the center of the sink disc and an arbitrary point on the edge of the sink disc, the head at the center of the sink disc and (optionally) the resistance and resistance layer depth of the sink disc. Default values for resistance and resistance layer depth are used if not specified. An optional label may be specified for each sink disc.

Note

If resistance is used on any sink disc, a depth for the resistance layer MUST be specified. A warning message will be issued if a resistance layer is specified with a thickness of zero.

Note

The use of a point at the edge of the sink disc is used (as opposed to the entry of the sink disc radius) to simplify the use of a digitizer in creating GFLOW1 input files.

Examples

- To create a head specified sink disc "DISK1" with its center at (50,50) and a point on its edge at (200,50) and with a head of 505 feet and other values defaulted, enter the following commands from the main menu:

```
> SINKDISC <CR>
> HEAD <CR>
50 50 200 50 505 DISK1 <CR>
> QUIT <CR>
```

- To create a head specified sink disc "DISK1" with its center at (50,50) and a point on its edge at (200,50) and with a head of 505 feet, resistance of 1 day and resistance layer depth of 5 feet, enter the following commands from the main menu:

```
> SINKDISC <CR>
> DISCHARGE <CR>
50 50 200 50 505 1 5 DISK1 <CR>
> QUIT <CR>
```

Sinkdisc Module

HIghlight

Command Description

Sets the highlight display option for sink discs. This command allows the user to display sink discs which are either recharging the aquifer or percolating in white (instead of the normal yellow) for visual analysis on screen displays.

The option, once set, remains in force for all graphical displays until changed by a later `HIghlight` command.

Note

If `HIghlight Recharge` is selected, percolating sink discs are, of course, also highlighted.

Examples

To highlight recharging sink discs on graphical displays, use the commands

```
≥ SINKDISC <CR>
≥ HIGHLIGHT RECHARGE <CR>
```

To highlight percolating sink discs use the commands

```
≥ SINKDISC <CR>
≥ HIGHLIGHT PERCOLATING <CR>
```

To turn off the highlight feature,

```
≥ SINKDISC <CR>
≥ HIGHLIGHT NONE <CR>
```

Sinkdisc Module

PLot

Command Description

Sets plotting options for sink discs. The user may select whether to display all disc sinks, only the head specified sink discs or only those which are discharge specified.

The option, once set, remains in force for all graphical displays until changed by a later `PLot` command.

Examples

To plot all sink discs, use the commands

```
≥ SINKDISC <CR>
≥ PLOT ALL <CR>
```

To plot only discharge specified sink discs, use the commands

```
≥ SINKDISC <CR>
≥ PLOT DISCHARGE <CR>
```

To plot only head specified sink discs, use the commands

```
≥ SINKDISC <CR>
≥ PLOT HEAD <CR>
```

Sinkdisc Module

REsistance

Command Description

Sets the default resistance (in units of time) for sink disc element creation. Once set, all following sink discs created will have this resistance unless a resistance is specified as part of the sink disc definition (see HEad command).

Examples

To set the default resistance to 5 days use the following commands:

```
> SINKDISC <CR>
> RESISTANCE 5 <CR>
```

Unless overridden as part of a sink disc definition, all future sink discs will have a resistance of 5 days.

Note

Resistance is defined as the thickness of the resistance layer (for example, in feet) between the stream and aquifer divided by its hydraulic conductivity (for example, in ft/day). Consequently, the resistance parameter is that of time (for example, in days).

This page is intentionally left blank.

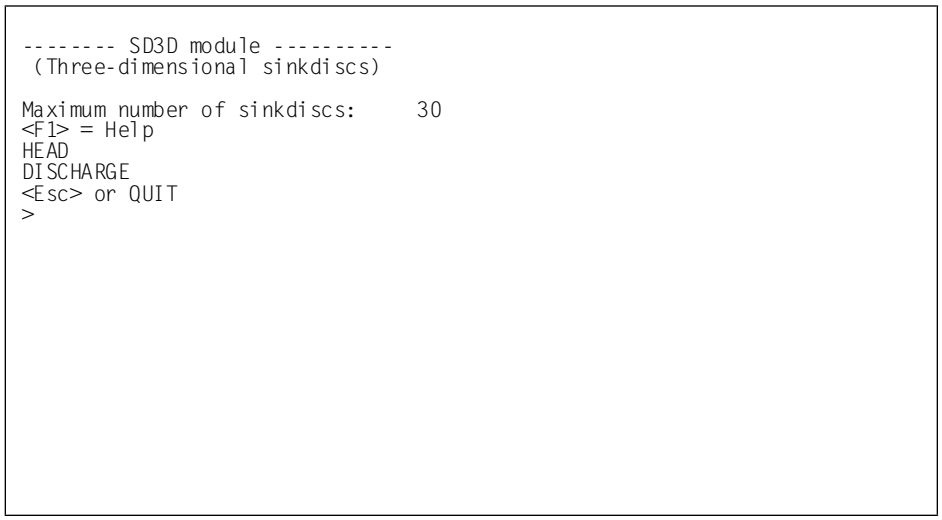


Figure 4.14 - Three-Dimensional Sinkdisc Module Menu

The SD3D module provides facilities for the creation of three-dimensional sink disc elements. 3-D disc sinks can be created as "head specified" or "discharge specified". These elements may only be used in regions where the flow conditions are confined.

Important

It is the modeler's responsibility to ensure that flow conditions near the 3-D sink discs penetrating well are confined at all times.

Commands

- `HEad` Selects creation of head specified three-dimensional sink discs.
- `DIscharge` Selects creation of discharge specified three-dimensional sink discs.
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu.

SD3D Module

DIscharge

Command Description

Begins the process of creating three-dimensional sink disc elements which are discharge specified. The user will be prompted for the *(x,y,z)* location of the center, the radius of the sink disc, and the exfiltration rate of the sink disc. An optional label may be specified for each sink disc.

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge feature.

Examples

- To create a discharge specified three-dimensional sink disc "DISK1" with its center at (50,50,350), a radius of 100 feet and an exfiltration rate of 0.003 ft/day, enter the following commands from the main menu:

```
≥ SD3D <CR>
≥ DISCHARGE <CR>
50 50 350 100 0.003 DISK1 <CR>
≥ QUIT <CR>
```

SD3D Module

HEad

Command Description

Begins the process of creating three-dimensional disc sinks which are head specified. When the model is `SOlved`, a strength parameter (sink density) will be computed for all head specified disc sinks. Each three-dimensional disc sink adds one equation to the solution matrix. The user will be prompted for the *(x,y,z)* location of the center, the radius of the disc sink, the head at the control point on the disc sink, and the offset of the control point from the center of the disc. An optional label may be specified for each disc sink.

Note

The control point offset makes it possible to create a circular lake with an approximately constant head along the lake bottom. This may be done by stacking concentric disc sinks of different radii and locating the control points midway on the "washers". By giving each control point the same head an approximate equipotential surface is obtained. Keep in mind that the solution remains radially symmetric hence the effect of a nearby well may not be properly accounted for.

Examples

- To create a head specified three-dimensional disc sink "DISK1" with its center at (50,50,350) a radius of 100 feet, a head of 505 feet, and a control point offset of 10 feet, enter the following commands from the main menu:

```
≥ SD3D <CR>
≥ HEAD <CR>
50 50 350 100 505 10 DISK1 <CR>
≥ QUIT <CR>
```

This page is intentionally left blank.

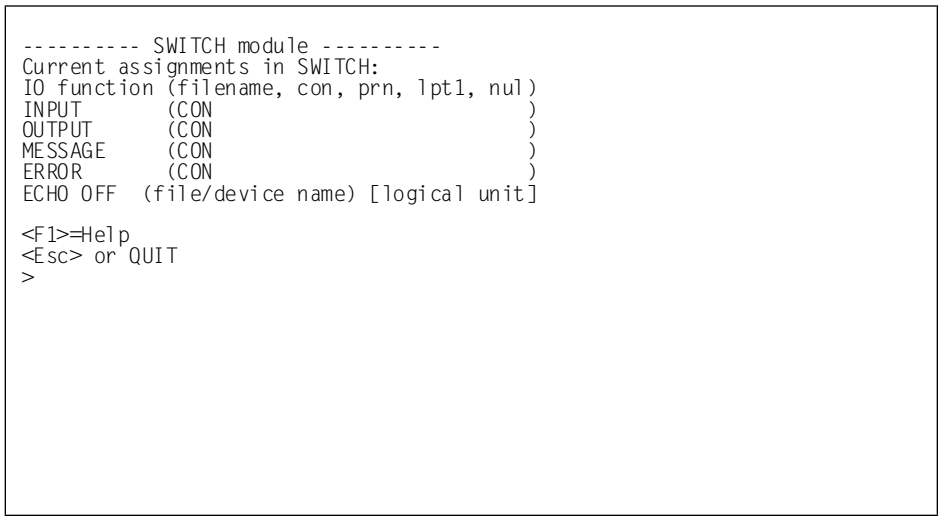


Figure 4.15 - Switch Module Menu

The SWITCH module provides tools for reading data files from and writing files to the computer's disk. Typically, the user creates a GFLOW1 data file (with the extension .DAT) using GAEP or TABLET (provided with GFLOW1) or a text editor and reads it into GFLOW1 by use of the SWitch commands. In addition, GFLOW1 allows the user to switch (or echo) most of its textual or numerical output to data files.

Commands

- `INput` — Switches program input from the user's keyboard to a data file (and back to the keyboard). The file must be properly constructed (see discussion of the `INPUT` command).
- `OUtput` — Switches numerical output from various GFLOW1 functions to go to a file.
- `MEssage` — Switches GFLOW1's messages (the user prompts and menus) to an output file. This command is most often used to disable menu printing when commands are input from a file by assigining `MEssage` to the device `NUL`.
- `ERror` — Switches GFLOW1's error messages to an output file. This command is most often used to enable the saving of error

messages while processing an input file (see discussion of the ERror command).

- ECho — Instructs GFLOW1 whether to echo the commands typed or read from an input file. Commands may be echoed to the screen, to a file or not at all.
- QUit or <ESC> — Returns to the GFLOW1 Main Menu.

Switch Module

ECho

Command Description

Echoes all commands to a file. This can be used to save a log of the commands typed during a session for review purposes, or to echo commands to the screen while processing an `INput` file.

Examples

- The user wishes to maintain a log of his session in the file `SESSION.LOG`. To do this, echo commands to the file, using the following commands:

  ≥ `SWITCH <CR>`

  ≥ `ECHO SESSION.LOG <CR>`

  ≥ `QUIT <CR>`

  Control is now at the main menu, and commands will be echoed as they are typed.
- To echo commands to the screen, place the following command at the beginning of your input file (see the `INput` command, Switch module):

  ≥ `ECHO CON <CR>`
- To disable command echoing, use the following commands:

  ≥ `SWITCH <CR>`

  ≥ `ECHO OFF <CR>`

  ≥ `QUIT <CR>`

Switch Module

ERror

Command Description

Switches GFLOW1 error messages from the screen to a disk file. To end the `ERror` switching and close the file, return to the `SWitch` module and switch errors back to `CON`. If the user switches errors to the file ERROR.LOG in an `INput` file, then errors (if any) will be automatically viewed at the end of the input file processing (see the `INput` command discussion).

Examples

- The user wishes to send error messages to the file `ERROR.LOG`. To initiate this, use the following commands to enable the error switching and return to the main menu:

  ≥ `SWITCH <CR>`

  ≥ `ERROR ERROR.LOG <CR>`

  ≥ `QUIT <CR>`

  Control is now at the main menu, and errors will go to the file.

- To return errors to the screen, use the commands:

  ≥ `SWITCH <CR>`

  ≥ `ERROR CON <CR>`

  ≥ `QUIT <CR>`

  The file `ERROR.LOG` is an ASCII file that can be viewed by use of the `VIew` command at the main menu or read using other DOS programs.

Note

If and only if the special filename `ERROR.LOG` is used to receive errors in an input file, the `SWitch` module automatically checks upon error file closure to see if any errors were recorded. If so, GFLOW1 warns the user and implicitly executes a `VIew ERROR.LOG` command so that the user may inspect the error messages. Always use the `ERROR.LOG` file to receive errors in your `.DAT` files.

Switch Module

INput

Command Description

Switches GFLOW1 program input from the keyboard to a disk file. The commands in the disk file are then processed as if they had been typed. The file must have the extension `.DAT`. To use this command properly, the file must return to the SWitch module and end with the command `INput CON`. If these commands are not present, GFLOW1 will attempt to return control to the keyboard by using a standard file `TRAIL.DAT`.

If an asterisk (*) appears as the first character of a line, GFLOW1 considers the input line to be a comment.

In addition to entering the switch module, the INput command can be invoked from the DOS command line when executing GFLOW1 or when entering the SWitch command from the main menu. See the examples below.

Note

- When processing an input file, GFLOW1 begins its processing inside the SWitch module. It is useful to set several SWitch options inside the file before the QUIT command leaves the SWitch module. The first lines of a `.DAT` file may read:

```
* disable messages
MESSAGE NUL
* echo commands from file to the screen
ECHO CON
* send errors to error.log
ERROR ERROR.LOG
* now leave switch module
QUIT
...
```

  You may create you own `.DAT` files by including these lines from the file `START.DAT`.

- To return GFLOW1 to interactive input from the keyboard and automatically examine the error log file, place the following commands at the end of the input file.

```
...
* reenter switch module (from main menu)
```

```
SWITCH
* return errors to the screen
ERROR CON
* echo all commands to the file INPUT.LOG
ECHO INPUT.LOG
* reenable the message prompts
MESSAGE CON
* now, return input to the console
INPUT CON
```

You may create you own .DAT files by including these lines from the file START.DAT. At the end of input file processing, the user will be inside the SWitch module. Use the QUit command to return to GFLOW1.

Examples

There are three different ways in which you may invoke the command INput filename.DAT:

- To process the input file MODEL.DAT by first entering the SWitch module, type the following commands from the main menu (the extension .DAT is assumed and may be omitted):

  > SWITCH <CR>

  > INPUT MODEL <CR>

  If a proper trailer exists on the file (see above), control will return to the keyboard after processing.

- To process the input file MODEL.DAT directly upon entering the SWitch module, type the following command at the main menu (the extension .DAT is assumed):

  > SWITCH MODEL <CR>

- To process the input file MODEL.DAT directly upon entering GFLOW1, type the following command at the DOS prompt (the extension .DAT is assumed):

  C:\MYMODEL> GFLOW1 MODEL <CR>

Switch Module

MEssage

Command Description

Switches GFLOW1 messages (prompt menus) from the screen to a file. To end the `MEssage` switching and close the file, return to the `SWitch` module and switch messages back to `CON`.

Examples

- The user wishes to disable message printing on screen (for example, during processing of an `INput` file). To do this, the messages can be sent to the null device, `NUL`. Use the following commands to send messages to NUL:

  ≥ SWITCH <CR>

  ≥ MESSAGE NUL <CR>

  ≥ QUIT <CR>

  Control is now at the main menu, and no messages will be printed. Note that if you type these commands by hand, all GFLOW1 prompts will be disabled, thought the program is "listening" if you type commands.
- To return messages to the screen, use the commands:

  ≥ SWITCH <CR>

  ≥ MESSAGE CON <CR>

  ≥ QUIT <CR>

Switch Module

OUtput

Command Description

Switches (or echoes) GFLOW1 program output from the screen to a disk file. Any output from user requests such as the `HEad` or `DIscharge` commands, `CURSOR` module, `CHeck` module, trace module and other outputs will go to the file instead of the screen (in certain cases, output will be sent to both the screen and output file). To end the `OUtput` switching and close the file, return to the `SWitch` module and switch output back to `CON`.

Examples

- The user wishes to send the output of several commands to the file `MODEL.OUT`. To initiate this, use the following commands to enable the output switching and return to the main menu:

  ≥ `SWITCH <CR>`

  ≥ `OUTPUT MODEL.OUT <CR>`

  ≥ `QUIT <CR>`

  Control is now at the main menu, and output will go to the file.
- To return output to the screen, use the commands:

  ≥ `SWITCH <CR>`

  ≥ `OUTPUT CON <CR>`

  ≥ `QUIT <CR>`

  The file MODEL.OUT is an ASCII file that can be viewed by use of the VIew command at the main menu or read using other DOS programs.

```
        --------- TRACE module ---------
<F1> = Help
POINTS         (  0 specified, maximum  200)
SHOW           (to display contents of POINTS buffer.)
TIME   0.1000000E+31 (max. residence time for a streamline)
STEP   0.2000000E-01 (step size along a streamline)
ZMARK  0.0000000
TMARK  0.0000000
DIRECTION FORWARD      (backward)
STREAMLINES VISIBLE    (dark)
SURFER OFF             (filename)
REPORT ON              (off)
LAYOUT ON              (off)
CONTOUR OFF            (on)
CURSOR  OFF            (on)
<F2> or GO
<Esc> or QUIT
>
```

Figure 4.16 - Trace Module Menu

The TRACE module provides a streamline tracing facility based upon a predictor-corrector integration procedure using the analytically calculated velocity vector. Streamlines may be traced forward or backward in time and may have tic marks added to indicate time or elevation.

Commands

- `POints` Allows starting points for streamlines to be entered from the keyboard or a file.
- `SHow` Shows the starting points currently entered for streamlines. Also used to save starting points in an input file.
- `TIme` Sets the maximum residence time for streamlines.
- `STep` Sets a step size interval (in terms of distance) for the numerical integration procedure.
- `ZMark` Sets up tic marks for particle elevations.
- `TMark` Sets up tic marks for travel time.
- `DIrection` Sets the direction for tracing (forward or backward in time).

- `STreamlines` Allows streamlines to be visible or invisible (tic marks are always displayed).
- `SUrfer` Sends streamline coordinates to a SURFER-format boundary line file (in *x*,*y*,*z* triplets).
- `REport` Enables/disables reporting of tracing progress on screen.
- `LAyout` Enables/disables the element and background map layout on streamline graphics.
- `COntour` Emables/disables contour lines if a current grid exists.
- `CUrsor` Enables/disables the use of the mouse cursor for starting streamlines.
- `GO` Enter the streamline tracing graphic display.
- `QUit` Returns to the GFLOW1 Main Menu.

Trace Module

COntour

Command Description

Enables or disables the display of contour lines during streamline tracing. By default, GFLOW1 does not include this information as part of the streamline tracing graphical display. Contour lines may be displayed only if a current grid exists (see `GRid` module). If contour lines are enabled, GFLOW1 will prompt for levels after the `GO` command is issued.

The `COntour` setting remains in effect until changed.

Example

- To enable contour lines, enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `CONTOUR ON <CR>`

- To disable contour lines (the default), enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `CONTOUR OFF <CR>`

Trace Module

CUrsor

Command Description

Enables or disables the use of the mouse cursor for entering the starting points for streamlines from graphic mode. If this option is turned off, points currently in the starting point buffer will be used as starting points for streamlines and will be traced automatically upon entering the graphic display. When these streamline traces are completed, the mouse cursor is enabled for (optional) additional streamline tracing. If the cursor option is on, the points buffer will be cleared upon entering the graphics display and the mouse cursor appears for streamline tracing.

Note

All starting points of streamlines selected with the mouse cursor are automatically added to the points buffer.

Example

- To enable the use of the mouse cursor for starting point entry, enter the following commands:

  > TRACE <CR>

  > CURSOR ON <CR>

- To disable the use of the mouse cursor for starting point entry, enter the following commands:

  > TRACE <CR>

  > CURSOR OFF <CR>

Comments about the cursor function

- When leaving the graphics screen and reentering the TRACE menu, the CUrsor option is automatically set to OFF.
- If you want to have the points currently in the buffer and also wish to use the cursor, select the CUrsor OFF option; after the streamlines from the buffer starting points are complete, the cursor will become available for additional streamline tracing.

Trace Module

DIrection

Command Description

Sets the direction (in time) for streamline tracing. This command is also available in cursor mode (> and < commands).

The `DIrection` setting remains in effect until changed.

Example

- To trace streamlines forward in time (the default), enter the following commands:

  ≥ TRACE <CR>

  ≥ DIRECTION FORWARD <CR>

- To trace streamlines backward in time, enter the following commands:

  ≥ TRACE <CR>

  ≥ DIRECTION BACKWARD <CR>

Trace Module

GO

Command Description

Begins the process of streamline tracing. The GRAPHICS menu (see the LAyout command from the Main Menu) will be displayed, allowing for window adjustment, etc. Type GO <CR> to proceed to the PLOT menu (if contour lines are enabled by the COntour ON command). Then type GO <CR> to proceed to the graphics display. As with all graphical displays, the <F7> key will send the graphics screen to the GFPRINT program for printing.

If points are currently in the starting point buffer, their streamlines will be plotted. After these points are plotted, the cursor will be enabled and other commands are available.

At any time during streamline tracing, you may press the `Ctrl` and `Break` keys simultaneously to end the streamline.

Commands Available at Graphics Screen:

- `S` — Begin tracing a single streamline, starting from the mouse cursor position. Starting point is added to starting point buffer.
- `W` — Begin tracing a group of streamlines from a well. Cursor must be placed at a well, and the direction must be set to backward. GFLOW1 will prompt for the number of streamlines to be plotted. Useful for generating well capture zones. Starting points are added to starting point buffer.
- `H` — Sets the starting elevation *z* for streamlines. GFLOW1 will prompt for the starting elevation. The starting elevation will remain in effect until changed. The default starting point is the saturated aquifer top; this value may need to be adjusted when `DIrection` is set to `BAckward`.
- `T` — Sets the spacing for time of travel markers (replicates the `TMark` command).
- `Z` — Sets the spacing for elevation tic marks (replicates the `ZMark` command).
- < or > — Switches direction for tracing. The < command replicates `DIrection BAckward` and > replicates `DIrection FOrward`.

- `D` or `V` — Sets streamline plotting mode. The `D` command replicates `STreamlines DArk` and `V` replicates `STreamlines VIsible`. Time of travel tic marks remain visible even when `STreamlines DArk` is selected.
- `<ESC>` — Exits the streamline cursor entry mode and proceeds to the graphic editing mode for adding contour levels and text labels (see the `PLot` command at the Main Menu).

Trace Module

LAyout

Command Description

Enables or disables the display of elements and background map information during streamline tracing. By default, GFLOW1 includes this information as part of the streamline tracing graphical display.

The `LAyout` setting remains in effect until changed.

Example

- To enable the layout display (the default), enter the following commands:

  ≥ TRACE <CR>

  ≥ LAYOUT ON <CR>

- To disable the layout display, enter the following commands:

  ≥ TRACE <CR>

  ≥ LAYOUT OFF <CR>

Trace Module

POints

Command Description

Defines starting points for streamlines from the keyboard or from a file. Starting points are entered as (*x*,*y*,*z*) triplets. If the *z* value is omitted, the streamline will start from the top of the aquifer (or from the water table in unconfined zones). Use the `QUIT` command to terminate point entry.

If the starting point buffer already has points defined (either by use of the POINTS command or by using the cursor), this command clears the starting point buffer.

Note

When tracing backwards in time (see `DIRECTION` command), starting from the aquifer top will result in almost immediate streamline termination if recharge is present, since the traced point will quickly leave the aquifer. When backward tracing, select an appropriate starting elevation. An elevation slightly above the aquifer base will give complete streamlines, while avoiding the many stagnation points on the aquifer bottom.

Example

- To set several streamline starting points, enter the following commands from the main menu:

```
≥ TRACE <CR>
≥ POINTS <CR>
```

Omitting the z value starts at the aquifer top:

```
≥ 100 400 <CR>
≥ 150 400 <CR>
...
```

Including the *z* values sets the starting elevation:

```
≥ 120 350 350 <CR>
...
≥ QUIT <CR>
```

Trace Module

REport

Command Description

Enables or disables the reporting of streamline tracing progress during tracing. Normally, GFLOW1 prints information about the progress of streamline tracing as text at the top of the graphical screen. Reporting adds overhead to the tracing procedure and slows the plotting of streamlines to some degree. The reporting function can be disabled or reenabled using the `REport` command.

The `REport` setting remains in effect until changed.

Example

- To enable the reporting of tracing progress (the default), enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `REPORT ON <CR>`

- To disable the reporting of tracing progress, enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `REPORT OFF <CR>`

Trace Module

SHow

Command Description

Displays the contents of the streamline starting point buffer at the console. If OUtput (see SWitch module) is set to a file, the points are echoed to that file in a format which may be read directly by GFLOW1 (using the INput command in the SWitch module). Starting points of streamlines that are aborted by Ctrl-Break are are included in the file but marked accordingly.

Example

- To view the streamline starting point buffer, enter the following commands from the main menu:

  ≥ TRACE <CR>

  ≥ SHOW <CR>

- To create the file POINTS.DAT after points have been stored in the buffer, leave the TRace module and type the commands

  ≥ SWITCH <CR>

  ≥ OUTPUT POINTS.DAT <CR>

  ≥ QUIT <CR>

  ≥ TRACE <CR>

  ≥ SHOW <CR>

  ≥ QUIT <CR>

  ≥ SWITCH <CR>

  ≥ OUTPUT CON <CR>

  ≥ QUIT <CR>

  The file POINTS.DAT may be read (INput command in SWitch module) at any time to restore the points in the point buffer.

  Note

  When giving the command SW POINTS, a warning is issued that data in the file POINTS.DAT will be added to the existing data. Since you are legitimately adding points to the points buffer in the TRace module, you may disregard this warning.

Trace Module

STep

Command Description

Sets the maximum step size (in distance units) for streamline segments. This command allows the user to increase or decrease streamline precision if desired. For increased precision near boundary conditions (such as linesinks), the step size will be adjusted by GFLOW1 during tracing.

The default step size set by GFLOW1 is 0.01 times the horizontal window size and is usually adequate.

Example

- To set the maximum step size to 100 feet, enter the following commands from the main menu:

  > TRACE <CR>

  > STEP 100 <CR>

Note

- When you select a step size that is more than twice the default step size you will be reminded of the default (recommended) step size. Smaller step sizes will yield more accurate streamlines and residence times, at least up to a certain point, as limited by the numerical accuracy of the integration routines. Too large a step size may cause premature termination of streamlines, oscillation of streamlines and inaccurate residence times.
- An inaccurate solution to an inhomogeneity may cause streamline tracing problems for streamlines which cross the inhomogeneity domain boundary. Remedy this by improving the representation of the inhomogeneity in the model (see the `INhomogeneity` module).

Trace Module

STreamlines

Command Description

Enables or disables the display of streamlines. This feature is included specifically for use in Wellhead Protection studies when isochrones of travel times are to be determined near wells. Suppressing the streamline trace (`STreamlines DArk`), while plotting travel time markers will result in the plotting of both dashes and arrowheads for alternating time intervals. These alternating time markers make it easy to separate successive isochrones. In addition, the markers are plotted perpendicular to a line from the marker to the well to which they belong. This helps to associate the proper markers with their wells.

The `STreamlines` setting remains in effect until changed. This command is also available in cursor mode (`D` and `V` commands).

Example

- To make streamlines visible (the default), enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `STREAMLINES VISIBLE <CR>`

- To make streamlines invisible, enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `STREAMLINES DARK <CR>`

Trace Module

SUrfer

Command Description

Allows the coordinates of points that define streamlines to be saved in a SURFER-compatible `.BLN` file. The *(x,y,z)* triplets can then be used to make 2- or 3-dimensional streamline plots in SURFER.

All streamlines traced will go to the .BLN file until the option is disabled.

Example

- To create the SURFER file `STREAM.BLN` containing streamlines, enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `SURFER STREAM <CR>`

  And then proceed with the usual streamline tracing commands.
- To disable the SURFER option, enter the following commands:

  ≥ `TRACE <CR>`

  ≥ `SURFER OFF <CR>`

Trace Module

TIme

Command Description

Sets the maximum travel time (or residence time) for streamlines. Streamlines will end after the specified time period.

Note

The time entered with the TIme command has the same units as the aquifer permeability and other time-dependent parameters.

Example

- Assuming that the permeability is measured in ft/day, the following commands will set streamlines to terminate after 10 years (3650 days):

  > `TRACE <CR>`

  > `TIME 3650 <CR>`

  The travel time termination setting remains in force until it is changed by another `TIme` command.

Trace Module

TMark

Command Description

Enables or disables time of travel marks on streamlines, and sets the time interval. The time-of-travel marks are arrowheads which point in the direction of flow; the base of the arrowhead is the actual time marker. To support time of travel marks, set the `TMark` setting to a nonzero value. The value will be interpreted as the time of travel between arrow heads on the streamline. The time of zero represents the beginning of the streamline.

This command is also available in cursor mode (`T` command).

Example

- To create time of travel marks 1 year (365 days) apart on streamlines, enter the following commands from the main menu:

  ≥ `TRACE <CR>`

  ≥ `TMARK 365 <CR>`

- To disable time of travel marks, set the `TMark` setting to zero:

  ≥ `TRACE <CR>`

  ≥ `TMARK 0 <CR>`

Trace Module

ZMark

Command Description

Enables or disables elevation tic marks on streamlines, and sets the elevation interval. To support elevation tic marks, set the `ZMark` setting to a nonzero value. The value will be interpreted as the elevation distance between tics. The tic marks will be made starting from the initial elevation of the streamline. Descending tic-marks (as streamline depth increases) are plotted to the left of the streamline when viewed into the direction of groundwater flow. Conversely, ascending tic-marks (as streamline depth decreases) are plotted on the right of the streamline.

This command is also available in cursor mode (`Z` command).

Example

- To create elevation tic marks at 10 foot depth intervals, enter the following commands from the main menu:

  ≥ `TRACE <CR>`

  ≥ `ZMARK 10 <CR>`

- To disable elevation tic marks, set the `ZMark` setting to zero:

  ≥ `TRACE <CR>`

  ≥ `ZMARK 0 <CR>`

Note

Streamlines may jump up or down underneath line sinks that are extracting or infiltrating water, respectively. As a result, tic marks may "bunch up" near the line sink and may be hard to count. You may experiment with larger `ZMark` values or large `STep` sizes to improve on the situation. You may also count tic marks from both the start and end of the line sink in order to determine the jump.

This page is intentionally left blank.

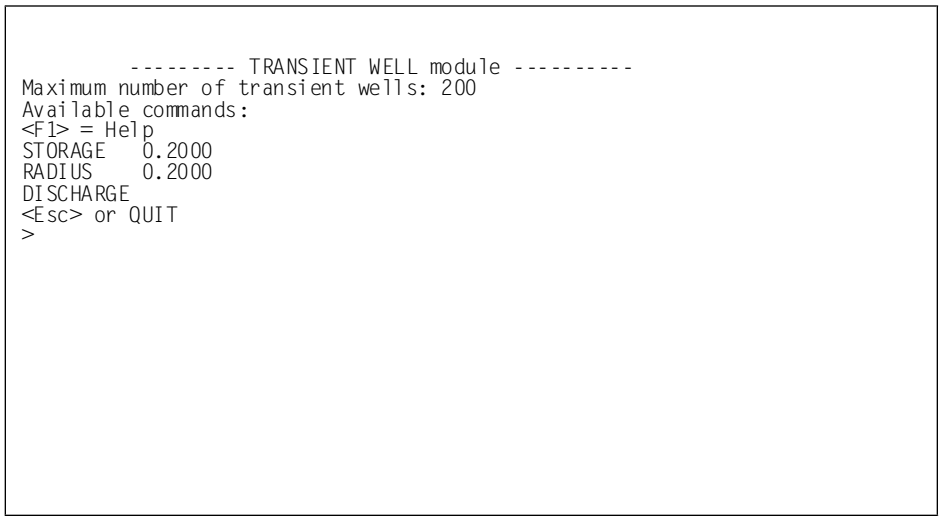


Figure 4.17 - Transient Well Module Menu

The TWELL module provides facilities for the creation of transient well analytic elements. Transient wells are modeled by use of the Theis equation. All transient wells must be discharge specified.

Commands

- `STorage` Specifies the storage coefficient for the aquifer.
- `RAdius` Sets a default radius for successive transient well definitions.
- `DIscharge` Begins creation of discharge specified transient wells.
- `QUit or <ESC>` Returns the the GFLOW1 Main Menu.

Twell Module

DIscharge

Command Description

Begins the process of creating transient well elements which are discharge specified. Transient wells are superimposed on an entirely steady-state solution. DO NOT ISSUE A `SOlve` COMMAND AFTER ENTERING TRANSIENT WELLS. The user will be prompted for the well location *(x,y)*, the discharge (pumping rate) of the well, the starting head at the well, the starting time for pumping and (optionally) the storage coefficient and radius of the well. If no storage coefficient is specified, the default storage coefficient (see `STorage` command) will be used. If no radius is specified, the default radius (see `RAdius` command) will be used. An optional label may be specified for each well.

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge well or losing stream.

Examples

- To create a transient pumping well "PUMP1" at the location (50,50) which pumps 150 gpm (28900 ft^3/day) and the default radius and storage coefficient, enter the following commands from the main menu (the well starts pumping at time=0 days and has an initial head of 400 feet):

```
> TWELL <CR>
> DISCHARGE <CR>
50 50 28900 400 0 PUMP1 <CR>
> QUIT <CR>
```

- To create a transient recharge well "RECH1" at the location (150,150) with an injection rate of 150 gpm (28900 ft^3/day), radius 1 foot and storage coefficient 10^{-5} , enter the following commands from the main menu (the well starts pumping at time=0 days and has an initial head of 400 feet):

```
> TWELL <CR>
> DISCHARGE <CR>
50 50 -28900 400 0 1E-6 1 RECH1 <CR>
> QUIT <CR>
```

Comments

- The storage coefficient, set independently for each transient well, is valid only in the area of influence of the well. Consequently, it is improper to specify different storage coefficients for wells which have overlapping areas of influence (overlapping cones of depression).
- If the transient wells affect head specified boundary conditions (streams, lakes, etc.), the groundwater flow solution becomes invalid. It is the user's responsibility to verify that transient wells do not invalidate (nearby) boundary conditions.
- If only one head specified boundary condition becomes affected by the well, an acceptable solution may be obtained (in the domain of the well) by use of a properly chosen image recharge well.

Twell Module

RAdius

Command Description

Sets the default radius for transient well element creation. Once set, all following transient wells created will have this radius unless a radius is specified as part of the transient well definition (see `DIscharge` command).

Examples

To set the default transient well radius to 6 inches (0.5 feet) use the following commands:

```
≥ TWELL <CR>
≥ RADIUS 0.5 <CR>
```

Unless overridden as part of a transient well definition, all future transient wells will have a 0.5-foot radius.

Twell Module

STorage

Command Description

Sets the default storage coefficient (dimensionless) for use with transient wells. GFLOW1 supports both confined and unconfined flow in Theis' equation. For unconfined flow, the storage coefficient is equal to the effective porosity of the aquifer. For confined flow conditions, the "storage coefficient" equals the "specific storage coefficient" multiplied by the aquifer thickness. The storage coefficient is dimensionless.

Once set, all following transient wells created will have this storage coefficient unless one is specified as part of the transient well definition (see `DIscharge` command).

Note

The storage coefficient, set independently for each transient well, is valid only in the area of influence of the well. Consequently, it is improper to specify different storage coefficients for wells which have overlapping areas of influence (overlapping cones of depression).

Examples

To set the storage coefficient to 10^{-2} use the following commands:

```
≥ TWELL <CR>
≥ STORAGE 1E-2 <CR>
```

This page is intentionally left blank.

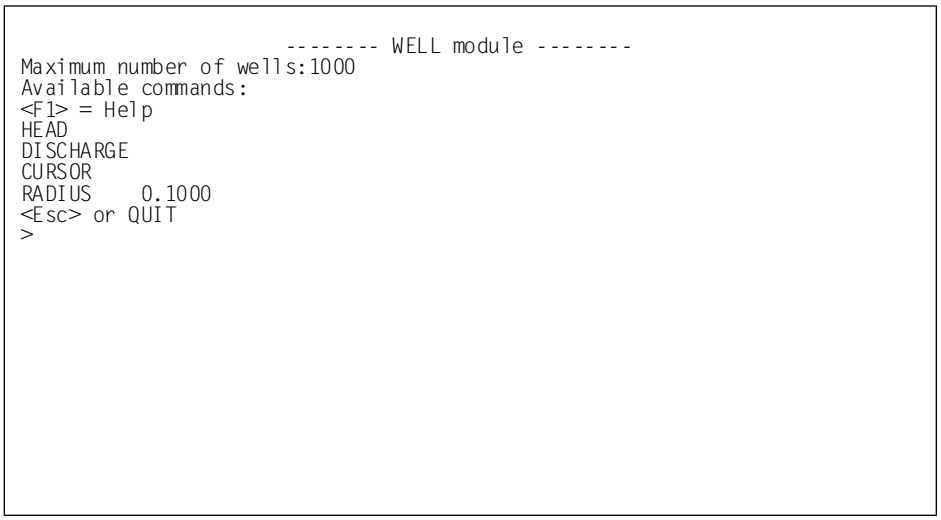


Figure 4.18 - Well Module Menu

The WELL module provides facilities for the creation and editing of WELL analytic elements. A cursor function exists for graphical editing of wells in a model. Wells can be created as "head specified" or "discharge specified".

Commands

- `HEad` Initiates creation of head specified wells.
- `DIscharge` Initiates creation of discharge specified wells.
- `CUrsor` Enters cursor mode for editing of wells.
- `RAdius` Sets a default radius for all subsequent well entries
- `QUit or <ESC>` Returns to the GFLOW1 Main Menu.

Well Module

CUrsor

Command Description

Selects the graphical editing facility for well elements. The graphics menu (see LAyout command in the Main Menu) will be displayed; set the window and press <F2> to enter the cursor facility. A layout of all wells in the current graphics window will be shown, and information about wells may be requested or modified by placing the mouse cursor (white arrow) at the well of interest and selecting a command (listed below).

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge well or losing stream.

Cursor Commands

Commands are selected by placing the cursor at the well of interest and pressing a single key. The available commands are:

- <CR> Displays information about the selected well.
- Q Change discharge (discharge specified wells only). GFLOW1 will display the current pumping rate and ask for a new value.
- H Change head (head specified wells only). GFLOW1 will display the current head and ask for a new value.
- R Change radius. GFLOW1 will display the current radius and ask for a new value.
- M Move the selected well. GFLOW1 will request that the new location be selected. Move the mouse cursor to the desired location and press <CR> to move the well.
- D Delete the selected well.
- <ESC> Exits the cursor editing facility and returns to the Well Menu.

Other Considerations

- Changes to well parameters will require a new solution (see SOlve command, Main Menu)

- Changes made are not automatically reflected in the user's data file. Use the `DAta` or `SAve` command at the main menu to save changes for future work.

Well Module

DIscharge

Command Description

Begins the process of creating well elements which are discharge specified. A discharge specified well adds no equations to the solution matrix. The user will be prompted for the well location *(x,y)*, the discharge (pumping rate) of the well and (optionally) the radius of the well. If no radius is specified, the default radius (see `RAdius` command) will be used. An optional label may be specified for each well.

Note - Sign Convention

For all discharge specified analytic elements in GFLOW1, a positive discharge value indicates water taken from the aquifer. A negative value means a recharge well or losing stream.

Examples

- To create a pumping well "PUMP1" at the location (50,50) which pumps 150 gpm (28900 ft^3/day) and the default radius, enter the following commands from the main menu:

```
> WELL <CR>
> DISCHARGE <CR>
50 50 28900 PUMP1 <CR>
> QUIT <CR>
```

- To create a recharge well "RECH1" at (150,150) which injects 100 gpm (19200 ft^3/day) and has a 1 foot radius (overriding the current default radius), enter the following commands from the main menu:

```
> WELL <CR>
> DISCHARGE <CR>
50 50 -19200 1 RECH1 <CR>
> QUIT <CR>
```

Well Module

HEad

Command Description

Begins the process of creating well elements which are head specified. When the model is SOlved, well discharges will be computed such that the specified heads at the wells are matched. Each head specified well adds one equation to the solution matrix. The user will be prompted for the well location *(x,y)*, the head at the well and (optionally) the radius of the well. If no radius is specified, the default radius (see RAdius command) will be used. An optional label may be specified for each well.

Note

Head specified wells are seldom needed. They may be of use when, for instance, a dewatering activity is modeled which has water level control switches on the wells. AT NO TIME SHOULD WELLS WITH OBSERVED STATIC WATER LEVELS BE ENTERED AS HEAD SPECIFIED WELLS. Such observed heads should be used only for judging model performance (see PIezometer command in the CHeck module).

Examples

- To create a head specified well "HEAD1" at the location (50,50) with head 105 feet and the default radius, enter the following commands, starting at the main menu:

```
> WELL <CR>
> HEAD <CR>
50 50 105 HEAD1 <CR>
> QUIT <CR>
```

- To create a head specified well "HEAD1" at (50,50) with head 105 feet and a 1 foot radius (overriding the current default radius), enter the following commands from the main menu:

```
> WELL <CR>
> HEAD <CR>
50 50 105 1 HEAD1 <CR>
> QUIT <CR>
```

Well Module

RAdius

Command Description

Sets the default radius for well element creation. Once set, all following wells created will have this radius unless a radius is specified as part of the well definition (see `HEad` or `DIscharge` commands).

Examples

To set the default radius to 6 inches (0.5 feet) use the following commands:

```
> WELL <CR>
> RADIUS 0.5 <CR>
```

Unless overridden as part of a well definition, all future wells will have a 0.5-foot radius.

This page is intentionally left blank.