Chapter 3 - GFLOW Tutorial

Introduction

This chapter contains two tutorial sections, a GFLOW1 and GAEP modeling tutorial and a GAEP digital map creation and maintenance tutorial. The tutorial sections are divided because some users will not need to use the digital map maintenance facilities, because they are provided with digital maps by purchasing prepared digital maps from Haitjema Software, LLC. It is strongly recommended that new users read Chapter 2 prior to performing the tasks in the tutorials. Before starting the tutorials, install GFLOW on your system (see Chapter 1) and be ready to begin at your computer system.

Please note that the purpose of the GFLOW tutorial is to introduce the new user to general operation of the software. A new user who has completed the tutorial should be ready to experiment with all parts of GFLOW, working with the reference manuals and exploring the program menus and online help. Good Luck!

Conventions: Commands, Files and Error Handling

Before proceeding with the actual tutorial, a discussion of conventions in both the manual and the software is necessary:

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>
  ```

- All GFLOW1 commands may be abbreviated to their first two letters, for example, the command:

  ```
  LAYOUT <CR>
  ```

  may be abbreviated as

  ```
  LA <CR>
  ```

  For clarity, all of the GFLOW1 commands in this tutorial are entered completely. As you work through the tutorial, experiment with the use of the command abbreviations.

Input Data Handling

The basic input data structure is an ASCII data file with readable command lines and parameters. This "filename.dat" file may be constructed by use of a text editor, a digitizing program (e.g. TABLET), the preprocessor GAEP, or writing a data file from GFLOW1. In the latter case, the data may have been entered in GFLOW1 by use of another filename.dat file, by giving input commands interactively, or both. Also, all graphical editing of data will be reflected in the ASCII data file written by GFLOW1.

The ASCII input data file contains the basic data, not a solution to the groundwater flow problem (or the surface water and groundwater flow problem). Once a solution is created in GFLOW1, the data and solution parameters may be written to disc in binary form: filename.sol files. The latter files can only be read by GFLOW1!

Error Handling

GFLOW1 has extensive error handling logic, in an attempt to avoid user frustration and to maximize modeling productivity and quality. During data input from an ASCII data file data is tested for integrity and consistency. If problems are detected (or suspected) three different actions may occur:

- Data is repaired and a warning is issued.
- Data is ignored and an error message is issued.

- Data is accepted, but a warning is issued.

All these messages are recorded on a "error file", which is automatically presented to the user at the completion of data entry. During interactive program operation, numerous command and data verifications are performed and error messages or warnings are issued when appropriate. Illegal or potentially incorrect command sequences are prevented or reported.

File Naming Conventions

All parts of GFLOW are designed to read and write files without (extensive) path names and without extensions. To do so, the program supports default extensions for all its input and output files. To avoid path names in the file IO commands, it is important that GFLOW1 is run from the current directory where all input and output files reside. The user is advised to set up a separate directory for each modeling project! Since all files have default extensions, there is no need to distribute these files over different directories. The default extensions are:

- `.DAT` ASCII analytic element input files or user-written GFLOW1 command files for input into GFLOW1. Analytic element files may be read by either GFLOW1 or by GAEP.
- `.DM` GAEP digital map files.
- `.SOL` Binary input and solution data.
- `.GRI` Binary grid data for contouring in GFLOW1.
- `.GRD` ASCII grid data for contouring in SURFER.
- `.MAP` Binary background map data for GFLOW plots.
- `.BLN` ASCII layout data for SURFER (boundary lines).
- `.INI` Binary initialization data for GFLOW executables.
- `.SPS` ASCII spreadsheet data for stream reaches.
- `.LOG` ASCII program responses (errors, input echo).
- `.EPS` ASCII encapsulated postscript instructions written by GFLOW1 and read by GFPRINT.
- `.PS` ASCII PostScript output from GFPRINT, compatible with monochrome or color PostScript devices or appropriate for importation into other programs.

- `.PCX` Bitmapped graphical image from GFPRINT appropriate for importation into other programs (for example, paint programs).
- `.PRN` Raster printer output from GFPRINT appropriate for dot-matrix or (non-PostScript) laser printers.
- `.DXF` Drawing Interchange Format files, either exported by CAD or GIS software or writte by GFPRINT for importation into CAD or GIS packages.

Only the VIEW command in GFLOW1 requires the specification of a filename extension, because it may be used to view any ASCII file.

Tutorial 1: Groundwater Modeling With GFLOW

Creating a groundwater model with GFLOW1 differs from the creation of traditional finite difference or finite element models. Most importantly, GFLOW1 has no grid or element network and, therefore, no grid or element boundaries. GFLOW1 solves groundwater flow in an infinite flow domain. The boundaries of the model area are formed by hydrological features (streams, lakes, wetlands, etc.) that surround the area of interest. This outer shell of flow features is referred to as the *farfield*. We are not interested in the hydrological behavior of the farfield itself, but need it in our model to obtain a proper solution in the *nearfield*: the area we are interested in.

To illustrate these concepts I suggest we use GFLOW. Assuming that you successfully installed GFLOW on your computer, set the current directory to the `\GFLOW\DEMO` subdirectory.

Examine the model with GAEP

Start GAEP: at the DOS prompt, type: `GAEP <CR>`. Move on to the main menu (press any key) and enter the File Menu; press `<F>`. Read in a digital map of our model area by pressing `<R>` and typing `DEMO` and pressing `<CR>` at the prompt. The program responds with the message that it entered 45 items and it displays the map name and project title: `DEMO.DM` and `GFLOW DEMONSTRATION DATA SET`, respectively. Next load an "element file", which is an ASCII `.DAT` file with input data for GFLOW1. Press `<L>` and type `DEMO` followed by pressing `<CR>`. Now the status menu reports that the current element file is `DEMO.DAT`. Leave the "file menu" by pressing `<ESC>`. Now let's look at the analytic element layout in our model area. Press `<E>` to enter the "element menu" and press `<V>` to view the data.

The light blue lines are streams (some times also including lake and wetland boundaries), while the yellow lines with crosses are line sink strings representing some of the streams. The big yellow circle is a sink disc with a negative strength to simulate recharge due to precipitation. The brown, green and magenta lines are back ground map features indicating roads (green), city layout (magenta), and the boundary of highly permeable channel deposits along one of the streams (brown). By moving the mouse cursor to a feature it will turn yellow and its name (label) will be printed above the graphics. Notice that the line sinks near the center of the screen are short (crosses marking line sink end points are close together), while the line sinks representing surrounding streams are longer. This reflects the distinction between the high resolution near field and the low resolution farfield. Also note that some of the farfield line sinks are represented as dotted lines. This indicates that the features have a zero width (in this case) or have properties which may lead to numerical instabilities. Zero width line sinks have no limits on infiltration rates so that the specified head at their centers are met regardless of the required infiltration or extraction rate. Zero width line sinks are good choices for the farfield. While viewing, the following keys may be used to navigate: `<PgDn>` zooms in, `<PgUp>` zooms out, and the arrow keys scoll the picture in the four arrow directions.

Ideally there should be three zones: a high resolution nearfield, around it a shell of medium resolution features, and outside that the low resolution farfield. However, the demonstration problem was set up for the educational version of GFLOW1 which supports only a small amount of analytic elements. Consequently, I compromised on the transition from nearfield to farfield. The nearfield in the demo problem is considered to be the area inside the magenta area and the high resolution stream network to the left of it. Note that there are farfield line sinks all around the near field. It is not proper to leave a big opening to the area outside the farfield, as much water may enter or leave through that opening. A reference point is defined outside the farfield (not shown on the screen) with a head that is selected roughly equal to the average head of the line sinks in the model. This reference point may be entered by the user in the aquifer module of GAEP, or if omitted will be generated by GAEP. If the nearfield and farfield are properly defined by line sinks, the automatic reference point generation by GAEP is the preferred procedure. If the farfield is properly defined different choices for the reference head will not affect the near field solution. This should be checked in GFLOW1 to ensure a proper farfield definition. Note: there is no need to associate the reference point with a surface water feature in the farfield. In fact, it is better not, since that surface water feature does not likely have a head (water level) that is the average of the line sinks in our model.

Solving with GFLOW1

I leave experimentation with GAEP to you. Let's leave GAEP and enter GFLOW1. Press `<ESC>` three times and answer `YES <CR>` to exit GAEP. Type `GFLOW DEMO <CR>` at the DOS prompt. The file contents of DEMO.DAT scrolls over the screen. Press `<ESC>` to leave the "switch" module. You are told that errors have been detected during input. Press `<CR>` for a report. This is a very common warning: "line sink may be too short". GFLOW1 compares the length of line sinks with the size of the domain specified by a window command. The default domain size is large enough to include all elements, including the farfield line sinks. This large domain makes GFLOW1's test rather conservative. Most of the time the line sinks are fine. You may revisit them, however, if numerical difficulties are encountered. Press `<ESC>` to return to the program. Type `LAYOUT` and press `<F2>`. Notice that all line sinks are dashed (although some short line sinks show up as solid lines on the scale of this figure). Dashed line sinks have zero strength parameters, they do not participate in the groundwater flow solution. This is obviously true at the moment because we did not yet create a solution. Press `<ESC>` and type `SOLVE 5 <CR>`. This starts a solution procedure with 5 iterations (5 successive solutions). For each solution the maximum errors in the boundary conditions are reported. For line sinks without resistance (not resistance layer between stream and aquifer) the error is the difference between the head in the aquifer and the specified head at a control point (line sink center) on the line sink. For resistance specified line sinks the error is the difference between its infiltration or exfiltration rate and the rate that is found from the difference between the head in the aquifer and the water level in the stream divided by the resistance coefficient.

The maximum error in satisfying boundary conditions appears to occur in line sinks with resistance: 0.4416%. To see where the errors occur, press <CR>, then type CHECK <CR> and PL ALL <CR> followed by pressing <F2>. The thick lines occur in the nearfield. Let's zoom in. Press <ESC>, and repeat the command PL ALL <CR>, now type WINDOW CURSOR <CR>. Move the cross-hair to the lower left corner of an imaginary box (window) around the near field and press <CR>.

Move the second cross-hair to the upper right corner of the desired window and press again <CR>. Now press <F2>, and note that the maximum error occurs at a line sink just outside the inhomogeneity domain defining the channel deposits around the main vertical river. The error reflects the difference between the line sink strength and the inflow or outflow calculated from the difference between the water level in the stream and the head in the aquifer and the resistance between the stream and the aquifer.

Leave the check module (press <ESC> twice) and enter the line sink module by typing LI <CR>, type CU <CR>, press <F2>, and move the mouse cursor (arrow) to the line sink that has the biggest error. Press <CR> when pointing at that line sink and repeat this for the surrounding line sinks. Occasionally you may get a large error (more than 10%) in a line sink with a very small discharge as compared to the surrounding line sinks. If so, the large error is inconsequential, as very small (numerical) variations in the discharge create large percentage errors. In our case, however, the discharges of the line sinks are all in the same order of magnitude, but the errors are small.

Inhomogeneities

Notice the yellow polygon around a portion of a stream. Its boundary is close to the brown layout of the high permeable channel deposits. The polygon is an inhomogeneity area which properties we may inspect in the inhomogeneity module. Press <ESC> twice to return to the main menu. Type IN <CR>, CU <CR>, and press <F2>. The red markers are the polygon vertices which are the end points of "line doublets" used to model the difference in hydraulic conductivity and recharge inside and outside the polygon. Move the mouse cursor to a vertex and press <CR> for data. The exfiltration rate is what is ADDED inside the inhomogeneity area, hence the total is that defined by the rain element (sink disc we saw in GAEP) and the 0.001 defined by this domain. As seen from the menu there are various editing options for inhomogeneities. In fact, GAEP does not support the creation of inhomogeneities, as it is easier to create them in this environment and then write a new data file (.dat) file by use of the DATA command from the main GFLOW1 menu. You may make some changes to this domain by moving points or adding and deleting points.

There are some important rules for inhomogeneities in GFLOW1:

- Inhomogeneity domains cannot overlap or share common boundaries.
- The perimeter of an inhomogeneity domain must cross line sink strings at end points of the line sinks.
- If you expect much variation in the head (e.g. near wells) smaller polygon sides (more control points) are needed.
- Do not switch from very large sides to very small sides; make a gradual transition.

None of these restrictions apply if the inhomogeneity is only used to vary the areal recharge, hence keeping the inside and outside hydraulic conductivity the same. Leave the inhomogeneity module by pressing <ESC> twice.

Stream Networks

Enter the line sink module by entering LI <CR> and type HI RE <CR>, CU <CR>. Now, set the viewing window using the mouse cursor by entering WI ALL <CR>, WI CU <CR>. Select a window large enough to include the stream network with the white head waters, and press <CR> to exit the window setting screen. Press <F2>. Notice the white head waters: these are losing stream sections! It is highly unlikely that head waters will have water to infiltrate year around (or on the bases of a yearly average). We can avoid such incorrect boundary conditions by solving surface and groundwater conjunctively.

Leave GFLOW1 (press <ESC> twice and type ST <CR>) and start GAEP. Use the file menu to read the DEMO.DM file and load the DEMO.DAT file. Return to the main menu and press <E> and then <P> to invoke the "properties" environment. Move the mouse cursor to the stream network that had the recharging sections and bring them in close-up with the page down and arrow keys. Click on POTTS DITCH and and enter Y <CR> to change the elements, then type STREAM <CR> (just S <CR> will do) to make the line sinks part of a stream network. Answer Y when asked if this is and END stream. Our stream network will consist of this ditch and the two branches to the left. Our ditch is the downstream end of the "network": "end stream". Accept the defaults of 0 for "overland flow" and "end inflow" by pressing <CR> twice. Accept the defaults for "width" , "resistance" and "resistance layer depth" by pressing <CR> three times. Notice that the stream section (line sink string) Potts Ditch is now plotted in white and solid! Select (click with the mouse) BRANCH1, answer Y <CR> to change properties, then S <CR> for stream, but NO <CR> for end stream. This branch must be connected to POTTS DITCH! Again 0 overland flow, 0 end inflow, and also accept the defaults for width, resistance and resistance layer depth. Repeat this procedure for BRANCH2. Press <ESC> twice to return to the main menu and select the file menu. Press

`<S>` and give the name DEMO1 and hit `<CR>` to accept the current model origin. GAEP creates a new GFLOW1-compatible analytic element data file `DEMO1.DAT`.

Leave GAEP and enter GFLOW1 with DEMO1, thus type at the DOS prompt: `GFLOW DEMO1 <CR>`. Skip over the error messages, we discussed them before. Type `SOLVE CONJ 8 <CR>`, which will start 8 solutions for both groundwater and surface water. Notice the report of negative stream flows in the line sinks labeled BRANCH1 and BRANCH2, we expected this based on our earlier solution for this problem. After a few iteration, no messages appear from the stream flow solution procedure, we are now just refining the groundwater flow solution. The groundwater solution does not significantly improve after the fourth iteration, so you might have hit `<CTRL-BREAK>` to skip the last few solutions. Since no significant maximum errors in boundary conditions are reported (more than a few percent) we may skip the check module.

Enter the line sink module and type `PL ST <CR>` (plot streamflow). Go to the main menu and type `LA <CR>` followed by `<F2>` to look at a layout. Only our stream network is plotted with a line width that increases down stream. Reenter `LAYOUT <CR>`, type `WI CU <CR>` and select a smaller window (zoom in). The head waters are now dashed: they have no strength and play no role in the groundwater flow solution, because they have no water to infiltrate! Due to groundwater inflow along the stream network we see a gradually increasing line thickness from head waters to the confluence with the river in the channel deposits. Again, line sinks that are not networked as streams are not shown when plotting stream flow.

You may enter the line sink module, enter cursor mode, and select the stream network for writing one or more spread sheet files. This feature allows you to use all the power of your favorite spread sheet program to analyze the stream and groundwater interactions (plot water levels and heads, base flow, stream flow, etc.). When selecting streams for spread sheet printing it is advised to reset the plot option in the line sink module to "plot layout".

Model Calibration

The stream flow data generated by GFLOW1 may be compared to observed stream flow at USGS gaging stations and thus help in model calibration. Keep in mind, however, that the modeled stream flows are steady state (average over many years). The traditional method of model calibration is to compare modeled heads with observed heads. Again, the user is reminded that our solutions are steady state. Consequently, no exact match can be expected between modeled heads and those observed in piezometers or domestic wells, regardless whether observed at one particular date or at various dates. Therefore, instead of looking for a precise match we should be looking for trends. When observed heads are measured at random dates, the best solution is the one with some of the heads too high and some too low, preferably without any spatial pattern. To facilitate this form of model calibration GFLOW1 supports a graphical (visual) check on the differences between modeled and observed heads.

Go to the main menu in GFLOW1. If you did not already do so, reset the plot option in the line sink module to "layout" and again return to the main menu. Type `SW DEMOPZ <CR>`. A warning appears that input data will be added to current data. This is O.K., we do intend to add piezometer data to our model. Hit a key and press `<ESC>` after the data has been read in. Type `CH <CR>`, followed by `PL PI <CR>`. The menu shows that the differences between modeled and observed heads (at the specified piezometers) vary between `-7.269` and `3.582`. Use the filter command to scale the markers that will be plotted at the piezometer locations, e.g. `FI -5 5 <CR>`. It is advised to use the same "filter" parameters when plotting the results of different model runs. This way the markers have the same size for the same differences, making it easier to judge the relative improvements that you are making during the calibration process. Press `<F2>` twice. The red upward triangles indicate where modeled heads are too high, the white downward triangles indicate modeled heads that are too low. The size of the triangle is proportional to the head difference, see the scale above the plot. You may display the piezometer label and observed and calculated (modeled) heads by placing the mouse cursor at a piezometer triangle and pressing `<P>`; the information for the piezometer is printed at the top of the screen. Press `<CR>` to enter the graphics annotation environment. The `<L>` key should only be used if there is a contour plot! You can, of course, always add text to the graphics with the `<T>` key. Press `<ESC>` to replace the annotation menu by the scaling information. This is the proper point to press `<F7>` to make a hard copy (or perform a screen dump if you work with the educational version).

Contour Plots and Streamline Tracing

Return to the main menu (press `<CR>` twice). Let's generate a grid for contour plots. Type `GR <CR>` and press `<F2>`. The points which appear on the screen are the locations where calculated values for the grid are computed. Press `<ESC>` when the grid is ready. Leave the grid module and enter the trace module: type `TR <CR>`. There are a variety of options on the trace menu, but we will select only two: `CO ON <CR>` and `CU ON <CR>`, which selects the plotting of piezometric contours and selects the cursor for starting streamlines. Press `<F2>` three times. Notice the green status bar. We will accept all default settings and trace a streamline by moving the cursor somewhat away from a stream and press `<S>`. Near wells or near the aquifer top or bottom the streamline tracing may be slowed down due to smaller step size selections by the program. If the streamline "hits" a stagnation point it will stop and report a residence time of 0.0E21. Note, you may at any time abort a trace by pressing `<CTRL-BREAK>`, after which you can start a new streamline.

Let's zoom in on the well inside the channel deposits. Press `<ESC>` twice, type `CO OFF <CR>` to disable contour plotting (we will change the window!) and press `<F2>` and type `WI 16300 13100 21700 18150 <CR>`. Press `<F2>` again to proceed to the streamline tracing environment. Note: you cannot overlay a contour plot unless you first generate a new grid for the new window. Select the starting point of the streamline near the aquifer bottom: press `<H>` and type `210 <CR>` (aquifer base is at 200). Press the `SHIFT` < key to trace backwards in time, press `<T>` and enter `365 <CR>` (one year residence time intervals). Now move the cursor to the well and press `<W>`, accept the default 10 traces: press `<CR>`. Ten streamlines are traced backward from the well. White arrow heads indicate one year travel time (residence time) intervals. Note that the markers are close together outside the channel deposits where there is a low hydraulic conductivity. This back tracing by use of the `<W>` key is useful for *wellhead protection* studies.

There is also a different way of creating "time of travel capture zones". Return to the trace menu by pressing `<ESC>` twice. Type `TIME 3650 <CR>`, which will limit our streamline traces to 10 years. Press `<F2>` twice to return to the graphics screen. Press `<D>` to set the streamline traces to invisible (see status bar). Also reset the height of the streamline starting point to 210 (is automatically reset to the aquifer top any time you reenter the graphics screen). Move the cursor again to the well and press `<W>` followed by `25 <CR>`. No streamline traces are visible, but time markers are plotted which are alternately dashes and triangles. The orientation of the markers is perpendicular to the well, so that when markers are plotted for more than one well they can be separated by their orientation. The marker patterns are useful for drawing *isochrones* (lines of equal travel time). If the picture looks too cluttered you may zoom in further (using the window cursor option) and trace new (dark) streamlines by use of the `<W>` key. Press `<ESC>` in order to annotate your graph, e.g. mark the isochrones. Press `<F7>` for a hardcopy or perform a screen dump when working with the educational version. Note: screen dumps should be performed with "display monochrome" (set in the graphics menu prior to starting screen graphics).

Streamline elevations

Tic-marks may also be used for indicating streamline depth. Let's try. Leave the graphics screen (press `<ESC>` until back in the trace menu) and type "points" and "quit" to clear the points buffer. Type `TIME 100000000 <CR>`, to avoid too soon an end to streamline tracing. Press `<F2>` twice, press `<T>` and type `0 <CR>`, press `<Z>` and type `10 <CR>`, press `SHIFT >` to select forward tracing and `V` to make streamlines visible, then locate the mouse cursor at some point in the aquifer far from the well. Press `<S>` and note the tic-marks to the left of the streamline indicating subsequent drops in elevation of the streamline of 10 feet. When passing underneath a line sink the streamline jumps up or down for exfiltrating or infiltrating line sinks, respectively. Depending on the step size the tic-marks may bunch up near the line sink making it difficult to count them. You may increase the step size, or count tic-marks forward from the starting point and backwards from the endpoint to determine the jump underneath the line sink. Tic-marks to the left indicate a drop in streamline elevation, while tic-marks to the right indicate an upward moving streamline. When you are finished, press `<ESC> <ESC>` to return to the main menu, then `STOP <CR>` to leave GFLOW.

More Practice

This concludes the tutorial steps for operating GFLOW. You may also refer to the files `PROBLE*.DAT`, many of which contain additional tutorial information. Each GFLOW module has extensive on-line help (`<F1>`). The reference manual provides additional information on the use of the various modules and menu options.

Tutorial 2: Creating and Managing a GAEP Digital Map

Note

A digitizer MUST be connected and properly configured (see Appendix C) prior to performing the digital map tutorial. If this has not been done, please refer to Appendix C before proceeding.

Digital Maps

The GAEP program allows the user to digitize hydrologic features from maps and to use these hydrologic features to create analytic element files for use with GFLOW. The process is performed in two steps:

- Create a GAEP digital map of hydrography
- Use the GAEP digital map to create elements (see below)

GAEP digital map files can be saved and reused to make as many different sets of elements as desired. By the use of digital maps, GAEP makes the creation of models with GFLOW faster and more accurate. GAEP can also reread element files (in GFLOW format) which it has created. Be aware that GAEP can only reread files which it has created or which were written by the GFLOW1 `DATA` command after originally being created by GAEP.

GAEP was installed by the GFLOW installation procedure (see Chapter 1). To start, set your working directory to the `\GFLOW\DEMO` directory and start GAEP:

```
C:\> CD \GFLOW\DEMO <CR>
C:\GFLOW\DEMO> GAEP <CR>
```

The GAEP program will start, and the menu will appear. At any time, you may press `<F1>` for help.

Map preparation

Prior to digitizing, the modeler should highlight the important points and coordinates on each map to be used in the project. It is important that the user work in a cartesian coordinate system (that is, one in which distances in the *x* (horizontal) direction and *y* (vertical) directions are scaled the same. This tutorial works in Universal Transverse Mercator (UTM) coordinates, but state plane or any local coordinate system may be used. DO NOT ATTEMPT to use latitude-longitude coordinates in GAEP, except to convert them to UTM (see Chapter 5, GAEP Reference, for details about the UTM to Lat-Long conversion facility).

A fully prepared map for digitizing is provided (Figure 1). This map was prepared from the U.S. Geological Survey 1:100,000 scale map of the same region as your sample data set `DEMO.DAT`. Since this is a monochrome image, it may be rather difficult at times to distinguish detail. For the digitizing example, the user may wish to mark in surface water features with a colored pencil.

Before digitizing, it is always best to mark up the map, writing any important information which will aid while digitizing. For this example, we will be adding two stream features,

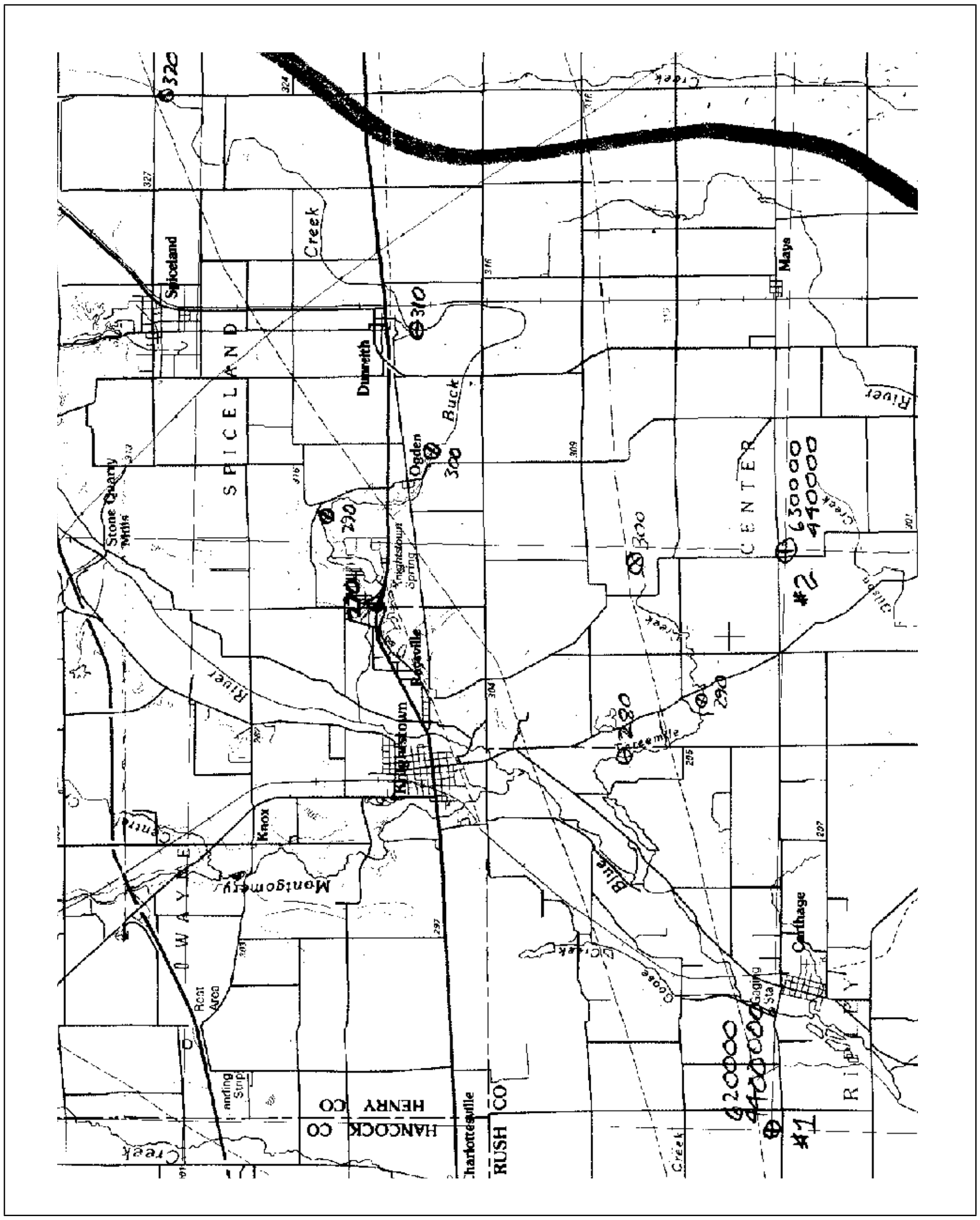

Figure 1. Sample map for digitizing example (near Knightstown, IN)

Buck Creek and Threemile Creek, and one background map curve, the perimeter of Knightstown, to `DEMO.DM`. The following have been marked on the map:

1. Places where contour lines cross the streams of interest[6] have been marked with a circled cross and the elevations are written nearby.
2. Two points for registering each map to the digitizer have been located (#1 and #2 in the lower portion of the map) and their world coordinates in UTM are written nearby. These points are to be used as "digitizer origins".
3. It is up to the user to mark the streams of interest and the Knightstown perimeter with colored pencil. (Note that Buck Creek enters the Big Blue River just north of Knightstown).
4. Note that all elevations in the demonstration data set are in meters.

Load the Existing Digital Map

Go to the GAEP Main Menu (press any key at the signon screen) and use the ReadDM command from the File Menu to read back DEMO.DM:

```
<F>
<R>
DEMO.DM <CR>
```

ensure that the digital map has been read successfully by going back to the Main Menu, then using the "Digitize/View" command:

```
<ESC>
<D>
<V>
```

Press `<ESC>` to return to the Digitize menu after viewing the original `DEMO.DM`. We may now begin to add features to the digital map.

Setting the digitizer origin

Before digitizing, points on the digitizer must be related to real-world coordinates. This is done by setting the "digitizer origin". Mount your map (Figure 1) on the digitizer and press `<O>` to begin. GAEP will request that the first origin point be selected. Place the digitizer puck at the first digitizer origin point (#1 at lower left) and press the first button on the puck (depending on your digitizer, this button may be labeled 1 or 0). You will be prompted for the world coordinates of the point. Enter the UTM coordinates from the map `620000 4400000 <CR>`. GAEP now requests the second origin point. Repeat the selection procedure for the second origin point (#2 at lower middle, UTM coordinates `630000 4400000`). GAEP is now ready to digitize map features!

---

[6] Brown contour lines scan poorly and cannot be seen in the figure.

Digitizing streams

We will use the Stream command to digitize the stream features. We'll begin with Buck Creek (GAEP will prompt for the name and abbreviation; use the default abbreviation that GAEP generates):

```
<S>
BUCK CREEK <CR>
<CR>
```

The graphic display we saw previously reappears, with the addition of a dotted white box which outlines the digitizer extent. Place the digitizer puck at one end of the stream and press the first puck button. Now move along the stream, pressing the first puck button at points along the stream. Use a sufficient number of points to describe the shape of the stream. Blue "plus signs" will be plotted on the screen as points are entered. If you make an error while digitizing, press the `<ESC>` key to abort. Press `<F3>` when you are finished digitizing the stream.

Now that Buck Creek is properly digitized, use the View command to again see the digital map,

```
<V>
```

note that Buck Creek appears in a dark blue color -- this color indicates that the stream had no heads associated with it, so line sink elements may not be created for it. Use the "Heads" command to add head values (those elevations marked on the map) to Buck Creek:

```
<H>
```

GAEP redisplays the map, and requests that a stream be selected. Move the mouse until Buck Creek is highlighted, then press the left mouse button. Now, the digitizing window reappears and the tablet is enabled. Place the puck at the labeled point for the elevation 320 m at upper right and press the first puck button. GAEP requests that the elevation be entered. Type

```
320 <CR>
```

Repeat the procedure for all the heads associated with Buck Creek. If you make a mistake, press `<ESC>`. When you are finished, press `<F3>`.

Now, repeat the digitizing process for Threemile Creek (just south of Knightstown). When you are finished, view the digital map again and admire your handiwork! The streams should appear in light blue when heads are added -- this means that you can use them to make analytic elements.

Digitizing Background Map Curves

As you might recall, we also wanted to add the perimeter of the city of Knightstown to the digital map. To do this, you will use the "Digitize/Curve" command in GAEP. GAEP allows you to make background map curves in four different colors: green, brown, cyan and magenta. GAEP does not distinguish between "closed" figures (for example, the perimeter of Knightstown) and "lines" (roads, for example), so you should mark a starting point somewhere on the perimeter of Knightstown so that you can digitize a completely closed figure.

We will use the Curve command to digitize Knightstown (GAEP will prompt for the name, abbreviationand color; use the default abbreviation that GAEP generates and the color magenta:

```
<C>
KNIGHTSTOWN <CR>
<CR>
MAGENTA <CR>
```

The graphic display reappears, with the dotted white box which outlines the digitizer extent. Place the digitizer puck at your desired starting point and press the first puck button. Now move along the perimeter of the town, pressing the first puck button at points along the stream. Use a sufficient number of points to describe the shape of the stream. Magenta "plus signs" will be plotted on the screen as points are entered. If you make an error while digitizing, press the `<ESC>` key to abort. Press `<F3>` when you are finished digitizing the town perimeter; remember to digitize the starting point you marked on the map as the last point!

Again, use the View command to examine the digital map. Feel free to digitize as many features as you like!

Other Data Types

In addition to the stream and curve data types which we have digitized, you may also wish to add some point features, though none are provided on the sample map. GAEP allows two types of point features, "Point Sets" and "Piezometers". Point sets may be used to add background "plus signs" to your GFLOW1 graphics (see the discussion of the "File/Map" command in Chapter 5 and the `MAp` module in Chapter 4). Piezometers may be used to create files of known water levels for model calibration (recall from the GFLOW1 Tutorial). This feature is described in Chapters 4 and 5 as well. Feel free to experiment -- draw a few points on your map and digitize them into your background map!

Background Map Editing

Suppose that you have made a mistake and included a digital map feature which you need to remove. Enter the "Edit" submenu from the Digitize menu (press `<E>`). YOu will see a set of editing features, Delete, Join and Rename. Refer to Chapter 5 of this manual for a further discussion of these commands. We assume that you've made no mistakes along the way, but again, experiment -- it's the best way to learn GAEP!

Write out the new Digital Map

It is now time to write the modified map to the disk. Leave the Digitize module and choose the "File/WriteDM" command to put the new digital map in `DEMO1.DM`:

```
<ESC>
```

```
<F>
<W>
DEMO1 <CR>
```

GAEP will create the new digital map file in your current working directory.

Final Remarks

As mentioned above, this is not a complete discussion of all of GAEP's features, but an introduction to the tools for digital map creation and modification. The best way to learn GAEP is to try it out. Each user will develop a style of working with GAEP which suits their own needs. Just remember to:

1. Carefully mark your maps before digitizing.
2. Save your work frequently.
3. Use a consistent scheme for naming features and abbreviations.

Tutorial 3: Creating a GFLOW1 Input File with GAEP

This section is not yet complete.

This page is intentionally left blank.