United States
Environmental Protection
Agency

Office of Research and
Development
Washington DC 20460

EPA/600/R-94/210
December 1994

EPA

W*h*AEM: Program Documentation for the Wellhead Analytic Element Model

EPA/600/R-94/210
December 1994

W*h*AEM:
PROGRAM DOCUMENTATION for the WELLHEAD ANALYTIC ELEMENT MODEL

by

Hendrik M. Haitjema
Jack Wittman
Vic Kelson
Nancy Bauch

School of Public and Environmental Affairs
Indiana University, Bloomington

CR 818029

Project Officer
Stephen R. Kraemer
Processes and Systems Research Division
Robert S. Kerr Environmental Research Laboratory
Ada, Oklahoma 74820

ROBERT S. KERR ENVIRONMENTAL RESEARCH LABORATORY
OFFICE OF RESEARCH AND DEVELOPMENT
U.S. ENVIRONMENTAL PROTECTION AGENCY
ADA, OKLAHOMA 74820

U.S. Environmental Protection Agency
Region 5, Library (PL-12J)
77 West Jackson Boulevard, 12th Floor
Chicago, IL 60604-3590

Printed on Recycled Paper

NOTICE

The information in this document has been funded wholly or in part by the United States Environmental Protection Agency (USEPA) under assistance agreement number CR-818029 to Indiana University. It has been subjected to the Agency's peer and administrative review, and it has been approved for publication as an USEPA document.

All research projects making conclusions or recommendations based on environmentally related measurements and funded by the USEPA are required to participate in the Agency Quality Assurance Program. This project did not involve environmentally related measurements and did not involve a Quality Assurance Plan.

The material introduced in this document should be fully understood prior to the application of the computer codes in W*h*AEM to field problems. Both the creation of the conceptual model and the interpretation of the program's output require an understanding of the Analytic Element Method and its implementation in W*h*AEM. Interpretation of the output generated by the programs is the sole responsibility of the user.

The software described in this document is available on an "as-is" basis without guarantee or warranty of any kind, expressed or implied. Neither the United States government (USEPA, Robert S. Kerr Environmental Research Laboratory), Indiana University, the University of Minnesota, nor any of the authors accept any liability resulting from the use of the codes.

Mention of trade names or commercial products does not constitute endorsement or recommendations for use.

FOREWORD

USEPA is charged by Congress to protect the Nation's land, air, and water systems. Under a mandate of national environmental laws focused on air and water quality, solid waste management, and the control of toxic substances, pesticides, noise, and radiation, the Agency strives to formulate and implement actions which lead to a compatible balance between human activities and the ability of natural systems to support and nurture life.

The Robert S. Kerr Environmental Research Laboratory is the Agency's center for expertise for investigation of the soil and subsurface environment. Personnel at the Laboratory are responsible for management of research programs to: (a) determine the fate, transport, and transformation rates of pollutants in the soil, the unsaturated zones of the subsurface environment; (b) define the processes to be used in characterizing the soil and subsurface environments as a receptor of pollutants; (c) develop techniques for predicting the effect of pollutants on ground water, soil, and indigenous organisms; and (d) define and demonstrate the applicability of using natural processes, indigenous to the soil and subsurface environment, for the protection of this resource.

Computer modeling has many uses in environmental research and development, including: representing our degree of understanding of subsurface processes in comparison to field and laboratory observations; and assisting in the design and predicting the performance of aquifer remediation and protection strategies. Occasionally, a new numerical solution technique comes along that is particularly well suited for environmental application. This research report explores the application of the *analytic element method* for the modeling of capture zones of pumping wells, and in particular in the design of wellhead protection areas. The Wellhead Analytic Element Model (W*h*AEM) should be considered a demonstration of a promising technology.

Clinton W. Hall

Clinton W. Hall, Director
Robert S. Kerr Environmental
Research Laboratory

ABSTRACT

The Wellhead Analytic Element Model (W*h*AEM) demonstrates a new technique for the definition of time-of-travel capture zones in relatively simple geohydrologic settings. The W*h*AEM package includes an analytic element model that uses superposition of (many) analytic solutions to generate a ground-water flow solution. W*h*AEM consists of two executables: the preprocessor GAEP, and the flow model CZAEM. W*h*AEM differs from existing analytical models in that it can handle fairly realistic boundary conditions such as streams, lakes, and aquifer recharge due to precipitation.

The preprocessor GAEP is designed to simplify input data preparation; specifically it facilitates the interactive process of ground-water flow modeling that precedes capture zone delineation. The flow model CZAEM is equipped with a novel algorithm to accurately define capture zone boundaries by first determining all stagnation points and dividing streamlines in the flow domain. No models currently in use for wellhead protection contain such an algorithm.

TABLE OF CONTENTS

ACKNOWLEDGMENTS

The authors express their appreciation to the volunteer group of "guinea pigs" who did the beta testing of the W*h*AEM codes. The codes and documentation were improved after anonymous technical review, as well as the many hours of review by graduate students at the Indiana University Groundwater Modeling Lab. Chursey Fountain of the Kerr Lab provided editorial comments and review.

CHAPTER 1. INTRODUCTION

W*h*AEM is a steady state ground-water flow modeling package designed to delineate capture zones and isochrones of ground-water residence times for the purpose of "wellhead protection." The package has been designed under USEPA assistance (CR # 818029) between the Robert S. Kerr Environmental Research Laboratory, Indiana University, and the University of Minnesota. The principal investigators were:

Hendrik M. Haitjema (Associate Professor)
School of Public and Environmental Affairs
Indiana University
Bloomington

Otto D. L. Strack (Professor)
Department of Civil and Mineral Engineering
University of Minnesota
Minneapolis

W*h*AEM consists of two independent executables:

- GAEP (**G**eographic **A**nalytic **E**lement **P**reprocessor) developed at Indiana University
- CZAEM (**C**apture **Z**one **A**nalytic **E**lement **M**odel) developed at the University of Minnesota

The foundation of the package is CZAEM, which solves the ground-water flow problem and generates the desired capture zones and isochrones of ground-water residence times. CZAEM is a single-layer analytic element model for steady state (regional) ground-water flow modeling (USEPA, 1994). The analytic element method employs superposition of elementary analytic solutions to ground-water flow features (Strack and Haitjema, 1981, Strack, 1989). The CZAEM program is an extension of the public domain code SLWL and is designed specifically to solve ground-water flow problems near well fields. New in CZAEM is the logic to automatically generate capture zones and isochrones of ground-water residence times (Bakker, Strack, in preparation), as well as the generation of isochrones for "contaminant fronts" using a new expanded transport equation (Strack, 1992).

The program GAEP handles data management for CZAEM (Kelson et al., 1993). Data preparation with GAEP is accomplished in two steps:

1) Create a digital map of hydrography using one or more U.S. Geological Survey (USGS) topographic maps. This procedure is greatly facilitated by use of a digitizer.

2) Create or edit an input data file for CZAEM using the digital map as a template for model building.

The purpose of GAEP is to ease the burden of input data preparation, in particular, to facilitate the editing of input data to improve CZAEM's ground-water flow solutions. **Figure 1** shows a cartoon of the W*h*AEM assisted modeling process. Creating and editing CZAEM input data files with GAEP is done graphically on screen by use of a mouse and appropriate key strokes. By making the process of modifying ground water models more efficient, GAEP allows the user to quickly evaluate a site several different ways. This iterative process of modeling is described in Chapter 2 of this document as the "hypothesis testing" approach to understanding ground-water flow.

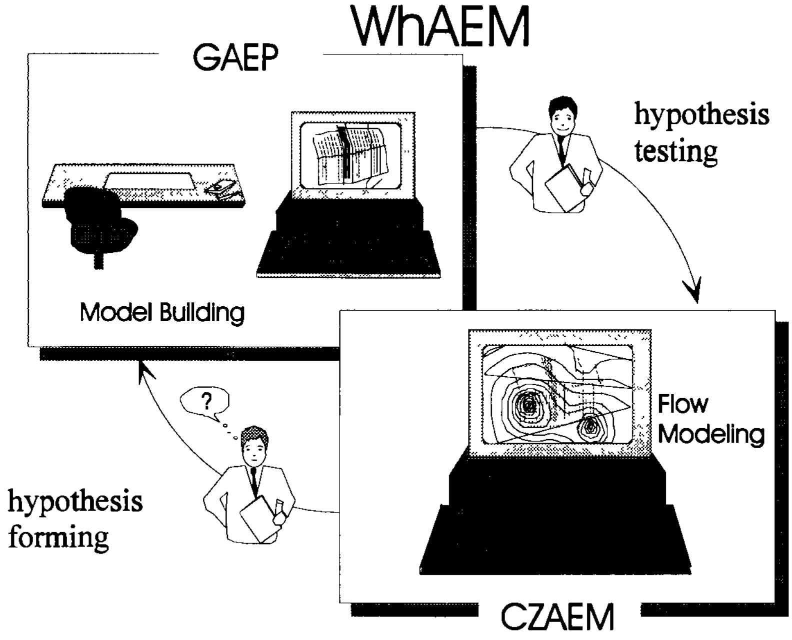

Figure 1 The modeling process using W*h*AEM.

System Requirements

The ground-water flow modeling system, W*h*AEM , has minimum hardware and software requirements described below. The ground-water flow model CZAEM has been written in Lahey Fortran[1] and runs on IBM-PC compatible personal computers in extended memory. The geographically-oriented preprocessor, GAEP, is written in Borland C/C++[2] and runs in conventional memory.

---

1 Lahey is a trademark of Lahey Computer Systems, Inc.

2 Borland is a trademark of Borland International

Hardware Requirements

- 80386- or 80486 based PC
- At least 1.5 megabytes of available extended memory
- At least 2 megabytes of available hard disk space
- Numeric Data Processor (except when using a 80486DX system)
- Microsoft, PS/2, or compatible mouse device
- Digitizer (optional)
- Printer (optional)

Software Requirements

- PC-DOS or MS-DOS Version 5.0 or higher
- DOS extender bound with `CZAEM.EXE` (provided)
- `ANSI.SYS` driver (supplied with DOS)
- Windows 3.1 (optional)

Installation Procedure

See the file README on the distribution diskette for the most current instructions.

Installing W*h*AEM on Your Hard Disk

The installation procedures for W*h*AEM are facilitated by the INSTALL.EXE program on the distribution disk. The installation program will locate potential target drives on the system, install the products in a target directory specified by the user and, when necessary, make any necessary changes to your CONFIG.SYS and AUTOEXEC.BAT files.

- To install, place the distribution disk in drive A: or B: and switch your logged drive to the floppy drive:

```
C:\> A: <Enter>
A:\>
```

or, if using drive B:,

```
C:\> B: <Enter>
B:\>
```

- Next, run the installation program:

  ```
  A:\> INSTALL <Enter>
  ```

  or

  ```
  B:\> INSTALL <Enter>
  ```

The INSTALL program will determine which available drives can be used for the installation and request an installation directory (default is C:\WHAEM). Once all the files are unpacked, you will have the opportunity to let INSTALL make changes to your AUTOEXEC.BAT and CONFIG.SYS files. The changes to these files can be made directly, in which case the old versions will be backed up, or changes may be saved to a new file and implemented in AUTOEXEC.BAT and CONFIG.SYS by the user. If changes are required, W*h*AEM will not operate properly until they are made and the system is rebooted with the changes made, in order for them to take effect.

Digitizer Configuration

Geographic data entry into ground water models is accomplished by transferring map data into digital form. This can be done manually, or through the assistance of a digitizing tablet or digitizer. GAEP facilitates both approaches. Configuration of the tablet is described in detail in Appendix C, "Tablet Configuration Guide." A utility program TABTEST is provided to simplify the process of setting up a digitizer to use with GAEP. The INSTALL program places a batch file TABSETUP.BAT in the \WHAEM installation directory. This file contains the DOS commands required to configure the digitizer. Program TABTEST allows the user to adjust the parameters for the digitizer driver allowing use of a standard digitizing tablet as the digitizing device. When the parameters are set properly, exit TABTEST to save the parameters to TABSETUP.BAT. In Appendix C, the supported tablet configurations are documented, including instructions for several popular digitizer models.

Printer Configuration

The printer is configured using a batch file. From the DOS prompt:

To create postscript files:

- Type: "`printer portrait`" This sets the CZAEM initialization up to send graphics to a postscript file (POST.PS) with a *portrait* orientation.

- Type: "`printer landscape`" This sets the CZAEM initialization up to send graphics to a postscript file (POST.PS) with a *landscape* orientation.

To print directly on an HP laser III compatible printer:

- Type: `"printer hplaser"` This sets CZAEM initialization up so that graphics will be printed on an HP laserjet III.

Another option for those without a Postscript printer is to use a public domain Postscript interpreter, such as Ghostscript, which is available by anonymous ftp over the Internet (for example, address: ftp.cica.indiana.edu[129.79.26.27], directory: pub/pc/win3/util/gs*.zip) .

W*h*AEM as a DOS application in Windows

Both programs, GAEP and CZAEM run as DOS applications under windows, with some rare, system-specific limitations. Run this way there is a familiar set of tools available to Windows users for capturing graphics screens, text, and other data generated while modeling.

CHAPTER 2. MODELING GROUND-WATER FLOW

When using W*h*AEM to delineate "time-of-travel capture zones" for well fields, you are engaging in ground-water flow modeling, which is an art just as much as it is a science. The opinions about the value of ground-water flow modeling are diverse: some profess an almost religious belief in model predictions; others call them a fraud. We believe that ground-water flow modeling can be a valuable tool to gain insight into the often complex subsurface flow processes which are otherwise hidden from the eye. The success of ground-water flow modeling, however, depends less on the "model" than on the professional skills and creativity of the hydrogeologist or engineer who performs the modeling. In designing W*h*AEM, we have tried to lower the barriers that can make ground-water flow modeling a slow, costly, and frustrating experience. We have not tried to create an "expert system" which claims to make human expertise obsolete.

There are several books that deal specifically with ground-water flow modeling (e.g. Bear and Verruijt, 1987 and Anderson and Woessner,1992). We assume that you are familiar with the basics of ground-water modeling in general, but not necessarily with the analytic element method employed by W*h*AEM. Therefore, we are providing a brief overview of some basic concepts that underlie the use of W*h*AEM.

Steady State Dupuit-Forchheimer Flow

Real world aquifer systems are usually far too complex to reproduce in a computer model. Consequently, several simplifications are made which are designed to make ground-water flow modeling feasible while maintaining the essential characteristics of the real world flow regime. Two of the most important simplifications are discussed below.

Steady State Flow

Although in reality the ground-water flow patterns vary in time, W*h*AEM deals only with steady state flow. Consequently, the modeling results reflect average conditions, which, depending on circumstances, may differ significantly from the actual conditions at any one time. Yet, there are important reasons to limit the initial study to steady state ground-water flow modeling, such as:

- Transient models require significantly more input data than steady state models (e.g. aquifer storativity, initial conditions).

- The transient modeling procedure is much more complex than steady state modeling. Consequently, it requires more expertise and more resources.

- Many more field data are required to properly calibrate a transient model as compared to a steady state model.

Instead of attempting transient ground-water flow modeling for delineating time-of-travel capture zones, it seems more practical to model average conditions. Deviations from average conditions, such as summer conditions and winter conditions, may be estimated by steady state modeling using summer and winter data, respectively. Such steady state representations of summer or winter conditions tend to overestimate the real transient solution; they bracket the actual transient flow patterns. In this manner, some insight is obtained into the relative importance of transient effects without engaging in a full blown transient modeling exercise.

Dupuit-Forchheimer Flow

In principle, ground-water flow is three-dimensional in nature. On a regional scale, however, horizontal flow is found to be far more important than vertical flow components (Dupuit,1863 and Forchheimer, 1886). In W*h*AEM, we employ the Dupuit-Forchheimer assumption, ignoring resistance to vertical flow. This assumption means that the piezometric head in the aquifer does not vary with depth, although vertical components of flow are still possible and may be estimated from continuity of flow (Strack, 1984). It appears that the Dupuit-Forchheimer assumption is acceptable as soon as boundary conditions in the aquifer are more than two aquifer thicknesses apart, or when areal recharge zones are more than two or three aquifer thicknesses in size (Haitjema, 1987). Otherwise, a fully three-dimensional analysis is warranted (Haitjema, 1985).

Limiting our modeling efforts to steady state Dupuit-Forchheimer flow greatly reduces the amount of field data required and enhances the efficiency of the modeling process. When done properly, most of the limitations that result from our simplifications can be overcome, at least in view of our objective: delineating time-of-travel capture zones. However, before discussing modeling procedures we will briefly introduce the analytic element method.

Analytic Element Method

The analytic element method was developed at the end of the 1970's by Otto Strack at the University of Minnesota (Strack and Haitjema, 1981). For a detailed description of the method refer to Groundwater Mechanics (Strack, 1989). A brief review of the method follows.

This new method avoids the discretization of a ground-water 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 as input data. 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 ground-water flow problem is then obtained by superposition of all 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 ground-water 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.

Hypothesis Testing

The ground-water flow solution obtained with W*h*AEM depends on many parameters, such as aquifer hydraulic conductivity, aquifer depth, areal recharge, and the interaction between the ground-water flow regime and ditches, streams, lakes, and wetlands. Many of these parameters are, in general, not very well known. In addition, there are real world complexities which cannot be included in our simplified model, such as local inhomogeneities, variations in recharge, aquifer stratification, etc. Instead of trying to remove uncertainties in parameterization through extensive data acquisition efforts prior to the modeling, we suggest a procedure of hypothesis testing.

The strategy is to model time-of-travel capture zones for various bounding values of the uncertain parameters, for instance, the lowest and highest hydraulic conductivities reported from pumping tests, or the low and high values on recharge, porosity, and pumping rates (summer and winter). The procedure accomplishes two things: (1) it will become apparent which parameters most affect the time-of-travel capture zones, and (2) a selection of time-of-travel capture zones for various extreme parameter choices may be overlain so that an envelope of time-of-travel capture zones can be constructed which incorporates uncertainties in the data.

The parameter sensitivity issue may be illustrated by the following example. Assume that the inclusion of a particular stream in the solution causes a major shift in the capture zone of the well field. In that case it may not be appropriate to include the stream in the envelope of time-of-travel capture zones. Instead, it is necessary to determine what the actual role of that stream is in the flow system. The model may provide clues in this case. For instance, if the stream is indicated on the USGS quad map as ephemeral (dash-dot line), while the model predicts that it loses large amounts of water to the aquifer, it is doubtful whether it is in contact with the aquifer. If the stream or ditch is known to carry water year round, it very well may be a boundary condition for flow in the regional aquifer. Hypothesis testing allows you to distinguish between those data which can be accepted and incorporated in the modeling results and those that require further study. In this manner, your modeling guides data acquisition and analysis activities, and not only vice versa.

The details of the step-wise approach to modeling with W*h*AEM, shown in **Figure 2**, will be introduced through a tutorial in the next chapter.

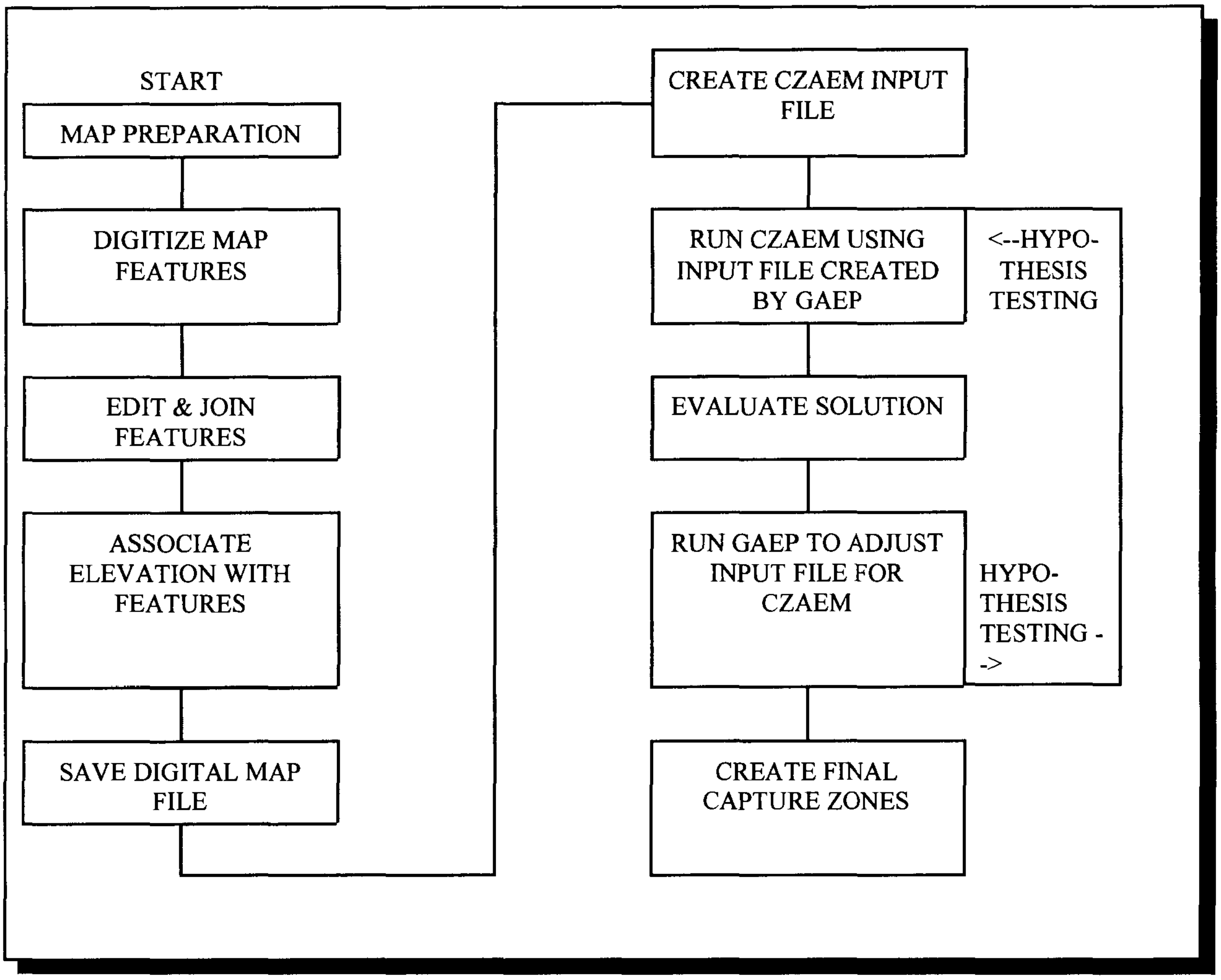

Figure 2 Hypothesis testing with W*h*AEM.

CHAPTER 3. W*h*AEM TUTORIAL

This chapter of the manual is a tutorial for the operation of the Geographic Analytic Element Preprocessor (GAEP) software package and the Capture Zone Analytic Element Model (CZAEM) software package for ground-water flow studies.

Data prepared for a wellhead protection demonstration project in the City of Vincennes, Indiana, will be used to illustrate the operation of W*h*AEM. The tutorial data set includes rivers, streams, and landmark features such as the well locations, municipal boundaries, and roads. Working through the following exercises should acquaint the new user with many of the key features of the programs and the basic concepts involved.

The purpose of the tutorial is to illustrate steps in delineating capture zones with W*h*AEM. These steps are:

1. Create a digital map file from a base map.
2. Use the digital map file to create an input data file for program CZAEM.
3. Read this file into CZAEM and solve the ground-water flow problem.
4. Generate graphical output and interpret modeling results.
5. Conduct hypothesis testing.
6. Delineate capture zones.

The reader is advised to work through this tutorial using a computer. We strongly suggest that as questions come up about specific commands you also take the time to read the CZAEM and GAEP reference guides in the appendices.

As discussed in Chapter 1, the two programs GAEP and CZAEM are in some important respects very different. The descriptions provided in this manual reflect that difference. Moving through the commands in GAEP is done either by clicking on the command with the mouse or typing in the first letter of the command from the keyboard. Throughout this chapter, the following convention is followed:

- Keys to be pressed are enclosed in angle brackets, e.g., <Enter> for the enter key.
- Text to be typed is shown in quotation marks.
- System prompts are in underlined bold face courier font, e.g., `C:\WHAEM>`.

CZAEM is a command line program which requires direct input of commands using the keyboard. General keyboard control procedures for moving around within the GAEP program are shown in **Figure 3**. We will start with the creation of a digital map file in GAEP.

Function-key control-

<F1> - context sensitive help.

<F3> - accept the data as entered.

<Esc>- quit this menu (or exit at the main menu).

Cursor control -

Be aware that the cursor is moved by *either* the digitizer puck or the mouse depending on the digitizer settings and the type of data being entered. Element creation and feature selection are always done with the mouse.

ZOOM in and out of any graphics image:

<Page Up> = ZOOM OUT <cursor arrows> = PAN
<Page Down> = ZOOM IN

Figure 3 General GAEP hints.

Load and Start the GAEP Program

To start GAEP, set your working directory to the C:\WHAEM[3] directory. Type "GAEP" and press the <Enter> key:

```
C:\>CD \whaem
C:\WHAEM> gaep
```

The GAEP program will start, and the introductory menu will appear. Press any key to enter the main GAEP menu. The main menu screen of GAEP is divided into two parts: an upper line listing the various modules of the program and the remainder of the screen, which indicates current settings.

3 Throughout this tutorial, we assume that you have installed W*h*AEM on the C: drive. If otherwise, replace C: by the proper drive letter.

The settings can each be changed in one of the modules of the program. Each module can be entered by one of two ways: 1) selecting the module with the mouse by clicking the word on the menu bar once with the left mouse button, or 2) pressing the key for the highlighted letter in the module name. Appendix A of this document provides a complete description of all GAEP menus, settings, and options. **Figure 4** shows the main menu screen with default settings.

```
File  Aquifer  Digitize  Element  Options  Utility  Quit


GAEP Release 1.0
Indiana University
SPEA Groundwater Modeling Laboratory   Current Directory:
                                       C:\WHAEM\DAT
WhAEM Version                          Current Map File:


                                       Current Element File:

                                       Memory available:   347248


                                       Option Settings:

                                       Unit Conversion:           M->FT
                                       Digitizer Mode:            MOUSE
                                       Video graphics mode:       COLOR

                   <f1> help  <esc> return to previous menu  07/Aug/94 01:00 PM
```

Figure 4 GAEP Main Menu screen.

When you run GAEP for the first time, prior to digitizer setup, the "Digitizer Mode:" will be set to "MOUSE." If a digitizer has been connected and configured for GAEP, as described in Appendix C, "Digitizer Mode:" will be set to "DIGITIZER." You can toggle this setting by selecting <O> (for Options), and pressing <D> twice (DigitizerMode, Digitizer). Press <Q> to return to the main menu, which should now look like the menu in **Figure 4**. Note: the "Memory Available" and the drive letter before the \WHAEM are, of course, dependent on your particular system.

An Example Digital Map

Before creating a digital map yourself, it may be helpful to take a look at an example. A digital map file has been prepared for this tutorial and was copied into your \WHAEM\DAT directory as part of the installation. The file VINCENNE.DM contains hydrologic features, roads and domestic wells (the file extension .DM is used on all GAEP digital map files). In this session you will read the digital map, view the contents, digitize a new stream in two separate sections, join the sections, and then create analytic elements for that stream. If you do not have a tablet, you can skip the exercises on digitizing.

At the main menu:

- Select the File module from the main menu, either with the mouse or the <F> key, and the File Submenu will appear (**Figure 5**).

```
New  ReadDM  WriteDM  LoadElem  SaveElem  Map  ChangeDir  setDrive  Quit


                                        Current Directory:
                                        C:\WHAEM\DAT
                                        Current Map File:

                                        Current Element File:

                                        Memory available:    347248


                                        Option Settings:

                                        Unit Conversion:          M->FT
                                        Digitizer Mode:           DIGITIZER
                                        Video graphics mode:      COLOR

                    <f1> help  <esc> return to previous menu  07/Aug/94 01:16 PM
```

Figure 5 GAEP File Submenu.

Read in the tutorial digital map file:

- Select <R> (for ReadDM). — This option allows the user to read files from disk. The program prompts for the file name.
- Type "VINCENNE" and press <Enter>. — The program displays the number of features read. The file-name extension .DM is assumed.

- Press the <Esc> key (or <Q> for Quit). Returns control to the main menu.

View the data:

- Press <D> in the main GAEP menu. Enter the DIGITIZE module (**Figure** 6).

```
Origin  Stream  Heads  Curve  Points  View  Edit  Quit



                                        Current Directory:
                                        C:\WHAEM\DAT
                                        Current Map File:


                                        Current Element File:

                                        Memory available:   347248


                                        Option Settings:

                                        Unit Conversion:           M->FT
                                        Digitizer Mode:            DIGITIZER
                                        Video graphics mode:       COLOR

                    <f1> help  <esc> return to previous menu  07/Aug/94 02:25 PM
```

Figure 6 GAEP Digitize Submenu.

- Press <V> to view.

The Vincennes digital map will appear on the screen, as shown in **Figure 7**. Individual features can be highlighted by moving the mouse; its color changes to yellow, and its name appears at the top of the screen. All lines are composed of a string of x,y coordinate pairs. Light blue lines are streams which have associated water table elevations along their reach. Dark blue lines are stream features which have no associated elevations (not in this file). The light blue lines through the middle of the study area are the banks of the Wabash River. Other map features are shown in red; roads are shown as straight and curved lines; production wells and observation well locations (piezometers) are marked by plus signs.

While viewing features in GAEP, you may change scale or shift the window on the screen as follows:

- Press <page down> Zoom in.

- Press <page up> Zoom out.
- Press the arrow keys Pan across the map.

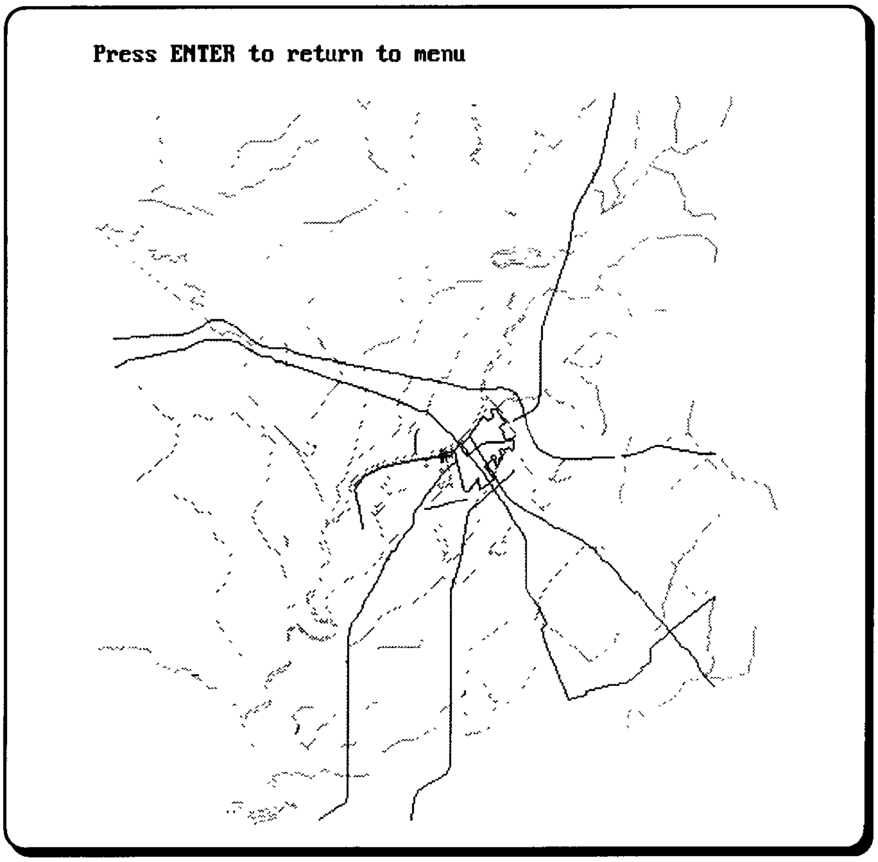

Figure 7. Prepared digital map of the Vincennes area.

When you are finished viewing, press <Esc> to return to the DIGITIZE menu and <Esc> again to return to the main GAEP menu. Then press <Q> (for Quit) to exit the GAEP program.

Prepare a Base Map for Digitizing

For this exercise you will be learning about the general operation of the program GAEP by building an example digital map file using a small portion of the USGS 7.5 minute Vincennes quad (included as Plate 1). The completed Vincennes digital map file will be used later in the tutorial to build a CZAEM input data file and to demonstrate CZAEM functions. Digitizing is greatly assisted through use of a digitizing tablet, but can be also be done "by-hand" (see Appendix C, keyboard entry protocol). If you would like to skip the digitizing discussion in the tutorial, proceed to section "Create an Input File for CZAEM."

Note:
The following exercises in the tutorial are designed to give the user practice in creating a GAEP digital map file. It is not necessary to create a complete file during this exercise; a digital map file VINCENNE.DM is available for later use.

Four Step Mark-Up

Digitizing is tedious but important work. The best rule to apply to the task is DO IT RIGHT THE FIRST TIME. One of the ways to assure success is to take the time to mark up the project maps before you digitize a single point.

1) *Mark the intersection of the topographic contour lines and the streams.*

Label each contour-stream intersection with the contour elevation on each of the streams on the map. For this particular exercise we will use Swan Pond Ditch, a small stream south of the City of Vincennes (located in the SE corner of the Vincennes, Indiana, quad sheet). In order to proceed, you should have all of the contours on this stream marked (contour levels 410- 440 ft),

2) *If there is no UTM grid on the map, draw and label this grid.*

GAEP has been designed to use the Universal Transverse Mercator (UTM) coordinate system. For more detailed information on UTMs, refer to the UTM Coordinates section of Appendix A, the GAEP Reference. UTM coordinates are marked along the margins of the topographic map. Unlike the other coordinates along the border of a USGS 7.5 minute map, UTMs appear in black type adjacent to light blue tick marks at 1000 meter intervals. Use these to draw straight lines across the map in a few locations to geo-reference the features digitized from the map. Be sure to include eastings 4276 000m, 4280 000m, and 4281 000m, and northings 450 000m and 456 000m. These will be used to define "digitizer origin points" within the digitizer surface area. An alternate way to get UTM coordinates is to convert from the latitude and longitude listed for a point on the USGS map using the GAEP UTM/LatLong conversion option in the Utility Menu (refer to Appendix A).

3) *Define a model origin for the project.*

The numeric values of UTM coordinates are large because the origin is at the equator (e.g., x-coordinates around 456,000 m and y-coordinates around 4,280,000 m near Vincennes). These large values may cause round-off errors in CZAEM. In order to minimize the dimensions of the model coordinates, GAEP shifts the origin from the equator to a spot within the area of interest. This is accomplished by defining a "Model Origin" (also referred to as the "Local Origin"). This model origin should be used for the duration of the project. For convenience, the model origin may be located west of the well field where UTM grid lines cross on the map. For instance, the point (450000, 4280000) may be used which is located just south of the Wabash River. The well field is located just south of the Wabash

River, west of town, near the sewage treatment area (no comments, please).

4) *Mark the locations and water levels of observed piezometric heads on the map from water levels in domestic wells, observation wells, or piezometers.*

These heads can be used to calibrate the model. For this tutorial we will not be entering these data.

Digitize Features from the Vincennes Quad

The next few sections will give you practice building a digital map file with GAEP digitizing utilities. It is assumed that a digitizer (minimum 12 inches by 12 inches) has been connected to the computer and tested, as described in Appendix C.

Run GAEP by typing "GAEP" at the C:\WHAEM> prompt. From the main menu enter the DIGITIZE module:

- Press <D> — The DIGITIZE menu will appear:

Origin Stream Heads Curve Points View Edit Quit

Associate Map Coordinates with Digitizer Coordinates

- Secure the map sheet onto the digitizer. — Position the lower right sections (T2N,R10W) on the active area of the tablet.

- Press <O> for Origin . — Defining an origin registers the map to the digitizer. You are prompted to move the puck to the first reference point.

- Move the puck to the UTM coordinates 450000, 4276000 and press button one. — A prompt will appear for the coordinates of the first reference point.

- Type in the UTM coordinates "450000,4276000", press <Enter>. — You will be prompted to move the puck to the second reference point.

- Move the puck the UTM coordinates 456000, 4281000 and press <Enter>. — A prompt will appear for the coordinates of the second reference point.

- Type in "456000, 4281000" and press <Enter>. The map coordinates are now entered in GAEP, and you will automatically be returned to the DIGITIZE menu, ready to digitize.

NOTE:
Referencing the map to the digitizer needs to be repeated when you move the map on the digitizer, or when digitizing a different map using new reference points.

Digitize Two Sections of a Single Stream

From the DIGITIZE Menu:

- Select Stream or type <S>.
- At the prompt type: "Swan Pond Ditch". Press <Enter>. The program responds with a default abbreviation for the feature (SPD). To accept this abbreviation press <Enter>.
- Place the puck at the south end of Swan Pond Ditch.
- Press button one on the puck. Repeatedly press button one as you move along the stream reach. Each point should appear on the screen as a dark blue plus sign. In general these points should be nearly the same distance apart along the stream. Stop digitizing at the intersection of Swan Pond Ditch and the railroad (where the stream turns sharply to the east), press <F3>.
- If you make an error while digitizing, press <Esc> to abort.

Go through the same steps as above for the remainder of the stream (from the railroad to the edge of the map). GAEP will label this part of the stream "SPD A."

NOTE:
We are digitizing Swan Pond Ditch in two sections for the purpose of this tutorial only. Later we will join these two sections into one stream. Normally this is necessary only when a feature continues on another map or extends beyond the active digitizer area on the current map.

Digitize the Elevations

You are now back in the DIGITIZE Menu after having entered two stream sections, labeled by

GAEP as "SPD ", and "SPD A", respectively. Next we will add surface water elevation data to these stream sections.

- Select Head or press <H>.

  Digitized features appear on the screen; in our case, only two sections of Swan Pond Ditch.

- Select the southern segment of the stream with the mouse.

  Move the mouse (not the digitizer puck), so that the first segment is highlighted. The segment will appear yellow and its name will appear at the top of the screen: [SPD]Swan Pond Ditch. You may want to use the <Page Down> and cursor commands to focus on the area of interest.

- Place the digitizing puck on the map where the 410 ft contour line crosses the stream (which you should have previously marked).

- Press puck button one.

  The following prompt will appear: "Enter elevation".

- Type the elevation (410) of the contour.

  Press <Enter>. The elevation will appear on the screen in yellow. To enter another elevation, place the puck at another contour line-stream intersection, press button one and type in the contour elevation.

NOTE:

Assume that the surface water elevation at the railroad intersection with the stream is 415 ft. The southern reach of Swan Pond Ditch, therefore, has only two associated elevation points: 410 at the map boundary and 415 at the railroad.

- If an error is made while entering elevations, press <Esc> to abort.
- When the surface water elevations along that reach have been digitized, press <F3> to save and exit the Head module.

Repeat the process for the other stream reach. Notice that when entering the graphics screen <V> the stream section with digitized water levels (heads) is now colored light blue.

Digitize Point Sets for Wells and Other Features

The locations of the wells are digitized from within the DIGITIZE menu:

- Select Point or press <P>.

  GAEP prompts you for the name of the feature to be entered.

- Type: "VINCENNES WELLFIELD".

  Press <Enter> twice. The digitizing screen will appear on the monitor.

- Place the digitizer puck on one of the well marks (the light blue circles along the river, west of the city).

- Press puck button one.

  Move the puck to another well and repeat until all five wells have been digitized. You will see purple plus signs on the screen for the points entered. <Page Down> to improve the viewing.

- If you make an error, press <Esc> to abort.

- Press <F3> when you are finished.

  You may repeat the process of digitizing points to represent the contamination sites and the known head locations (domestic wells, etc.).

- Contamination sites may be digitized as a single, named, point set. Known well/piezometers elevations may be digitized as another. Non-point sources can be outlined using the "curve" command discussed below.

Digitize Roads and City Limits

The curve command <C> is used to digitize a set of points that define a curve for non-hydrologic landmark features such as roads and city limits. From the Main Menu select Curve or press <C>. At the prompt enter the curve name:

- Type: "City Limits".

  Press <Enter> twice. The digitizing screen will appear on the monitor.

- Place digitizer puck on city limits.

  Press puck button one. Continue to press puck button one as you move along the city limits, selecting points farther apart where the boundary is straight, and closer together where it is more curved. Plus signs will appear on the screen for the points entered.

- If you make an error, press <Esc>, and start over.

- When finished digitizing, press <F3>.

- Repeat the procedure to digitize roads.

  Select a single road for the exercise.

View the Digitized Features

After digitizing Swan Pond Ditch, the city limits, a road, and the well field, select the view command by pressing <V>, in the DIGITIZE module to see the results. You may have to zoom out <Page Up> or shift the figure using the arrow keys to see all the features entered. Press <Esc> or click the mouse button to return to the DIGITIZE module.

File Operations in GAEP

The **F**ile module allows the user to save and read files from disk. To save your digital map (including the non-hydrologic map features):

- Press <Esc>. — Return to the GAEP Main Menu.
- Select File or press <F>.

New ReadDM WriteDM LoadElem SaveElem Map ChangeDir setDrive Quit

- Select **W**riteDM or press <W>. — The system prompts for a file name, e.g., your name (no extension needed). Make sure to give a name <u>other</u> than VINCENNE! We do not want to overwrite the existing file VINCENNE.DM.
- At the title prompt type: type a title, or press <Enter>.
- The file will be saved with the extension ".DM" .
- Press <Esc> to return to the main GAEP menu.

Edit Features in GAEP

You have digitized sections of the same stream from a single quad map. These two sections could have come from a stream which crossed from one map to another. In either case, you may wish to connect these two sections into one feature.

Using the EDIT menu, you can join these features into a single stream. Commands to edit digitized features are available in the EDIT menu of the DIGITIZE module.

From the main menu, select the **D**igitize option and the Edit command, press <D> followed by <E>.

Join View Delete Rename Quit

View the Data

Select the view command <V>. When the graphics screen appears, move the mouse around to highlight the various features. Note that the stream has two sections and each has a separate name. Our next step will be to join these features together under a single name; press <Esc> to return to the Edit menu.

Join Features

The Join command connects stream or curve segments. The feature that results will have the name of the segment selected first.

- Press <J> to enter the graphics screen.
- Highlight one of the two digitized segments by moving the mouse cursor.
- Press the left mouse button to select segment. The stream segment color will change from yellow to red.
- Highlight the other segment.
- Press the left mouse button. The color will again change to red. You will be prompted to confirm that these are the correct segments to join.
- Press <Y> and <Enter>. The entire stream reach will appear in red and is now one feature. A prompt will again appear to let you join another segment to these. Press <Esc> twice to return to the EDIT menu.
- If you make an error, press <Esc>. You will then be returned to the EDIT menu.
- Revised digital map can be viewed by pressing <V>.
- When you are finished, press <Esc>. This will to return to the DIGITIZE menu and <Esc> again to return to the main GAEP menu.

The same join procedure can also be used to join "curves." The revised digital map file can be saved on disk. From the Main Menu:

- Press <F> (**F**ile) and then <W> (**W**riteDM).
- At the prompt, type in a title or press <Enter>.

- Type in a file name (do not use vincenne!), and press <Enter> or accept the current file name by pressing <Enter>. When the file has been saved to disk, press <Esc> and then <Q> to exit.

Create an Input File for CZAEM

Up to this point, you have used GAEP to build and edit features of a digital map. The map, however, is far from complete; only a few features were entered for demonstration purposes. For the remainder of the tutorial, you will be working with a complete digital map of the Vincennes area: VINCENNE.DM, which is provided in the \WHAEM\DAT directory.

Relevant information about the aquifer, the well field, and local geology are needed for a ground-water modeling project. Because CZAEM is a steady state model, the input parameters should be thought of as average annual values. The following aquifer and well data are required:

- Pumping rates and radii for the wells in the well field
- Elevation of the aquifer base
- Average aquifer thickness
- Hydraulic conductivity (permeability)
- Porosity
- Average head in the study area (called reference head)
- Average areal recharge rate

Much of this information can be acquired from state and U.S. Geological Survey reports, well field data, and domestic well logs from state natural resource or environmental agencies.

GAEP allows the modeler to quickly build an initial model of an area covered by a digital map. It also makes modification of input data fast and efficient. These quick and easy changes in the CZAEM input file are important for hypothesis testing, as discussed in Chapter 2.

Creating Analytic Elements

The Element module in GAEP will be used to create and edit CZAEM input files using VINCENNE.DM. It is in this step of the process that hydrological features (streams, lakes, wells) are going to be represented by analytic elements used in CZAEM to solve the ground-water flow problem. This section of the tutorial outlines the necessary steps to build a basic CZAEM model containing line-sinks and wells.

First a word about data resolution. A decrease in model "resolution" is apparent moving outward from the area of interest (see **Figure 8**). The "near field" features are those in the immediate area of the well field. These features should be defined with the highest resolution (shortest) line-sinks.

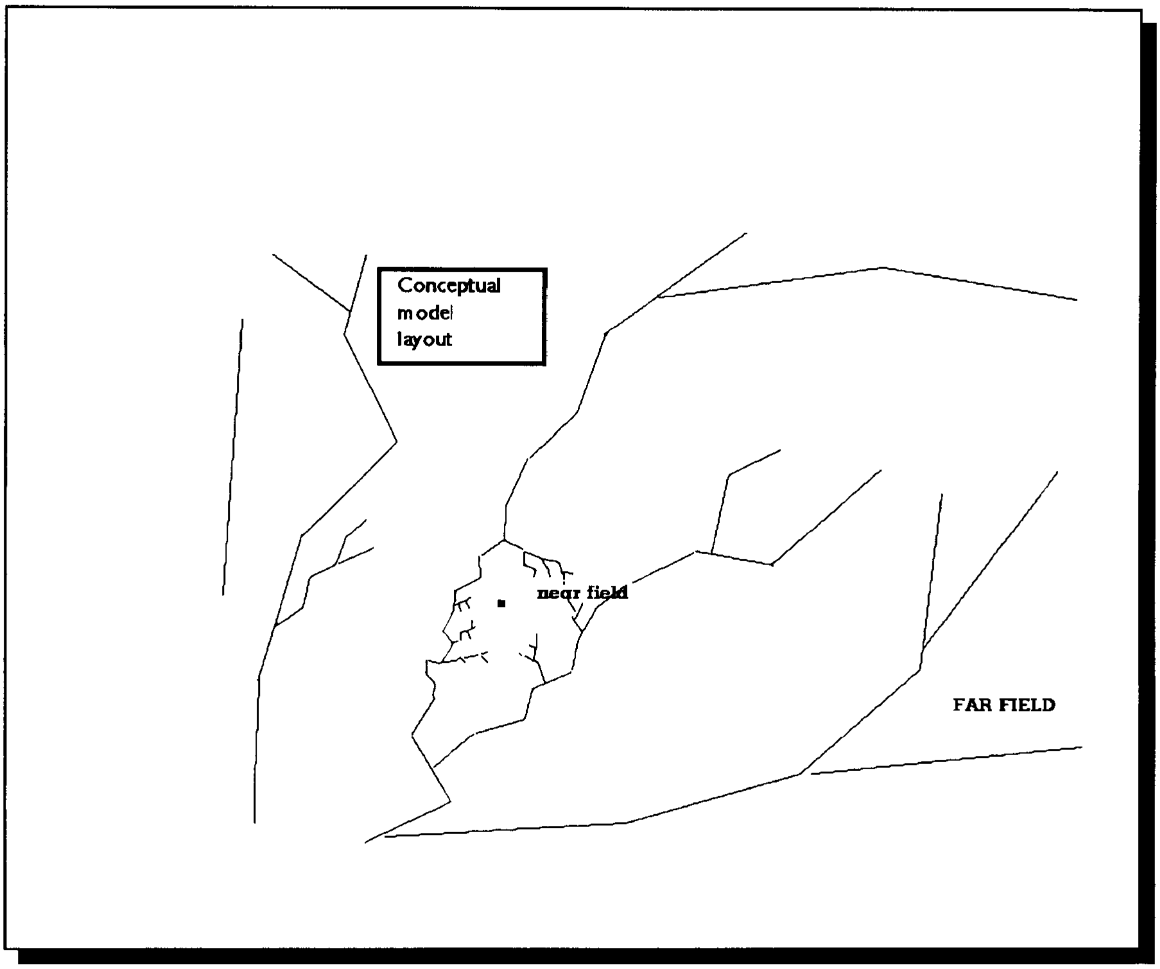

Figure 8 Relative positions of near-field and far-field features in an analytic element model. Notice the difference in resolution of the line-sinks.

Out in the "far field", hydrologic features can be represented much more coarsely. The purpose of these "far field" features is to control the ground-water flow into (or out of) the study area of the model.

Users unfamiliar with the analytic element method and the process of representing streams as discrete line segments frequently ask, "How many line-sinks are needed and how precisely should the segments match the actual streams?" Representing streams by an appropriate string of line-sinks requires experience and trial-and-error runs with CZAEM. The length of the line-sinks determines the effective resolution with which you can represent a stream. The fact that a stream may have a long straight section does not mean one long line-sink will adequately represent that section. The discharge along the section may vary significantly, in which case several (constant strength) line-sinks are required to properly model that discharge distribution. In general, shorter line-sinks are used for streams close to the well field (the area of interest). Line-sink length should gradually increase or decrease with changes in resolution because adjoining line-sinks of significantly different length may cause numerical inaccuracies. When creating line-sinks, you do not need to precisely match the stream location or shape, certainly not in the far field. The reader is referred to the CZAEM User's Guide for exercise in creating line-sink representation of streams (USEPA, 1994). The **E**lement module requires that streams and wells to be included in the model to already be digitized and read into GAEP. A complete digital map has been installed in the \WHAEM\DAT directory for this exercise.

NOTE:
The following section of the tutorial is designed to give the user practice in creating a CZAEM input data file. However, it is not necessary to create a complete file during this exercise. A completed input file (VINCENNE.DAT) has been included for later use in CZAEM.

Start GAEP at the DOS prompt:

`C:\WHAEM> gaep`

From the main menu:

- Select **F**ile or press <F>.
- Select **R**eadDM or press <R>.
- Type: "VINCENNE". Press <Enter> to import the file. Press <Esc> to return to the main menu.

Enter the **E**lement module:

- Select **E**lement or press <E>. The following set of choices will appear:

Linesink Well Delete Userwindow Image View Quit

Creating Line-sinks:

- Select the Linesink command <L>.

GAEP will display all presently defined map features. Only streams which have water levels defined (features displayed in light blue) can be used to generate line-sinks. In VINCENNE.DM all features are light blue.

For this exercise, we will create line-sinks for Kelso Creek, located east of the well field and the City of Vincennes, as shown in **Figure 9**. Prior to selecting the stream, adjust the figure so that the stream fits on the screen using the zoom <pageup> or <pagedown> and the arrow keys to adjust the viewing window.

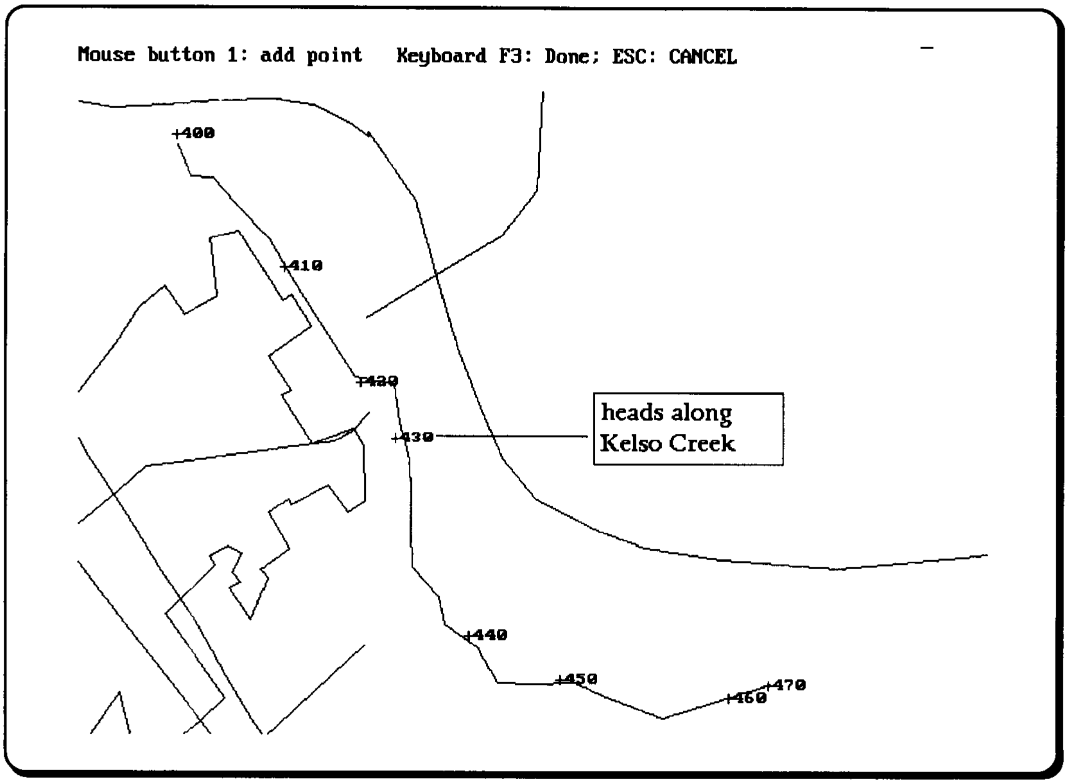

Figure 9 Kelso Creek with elevation marks where contours cross the stream.

- Select the stream, using the left mouse button.

The stream and the water levels at known points will be displayed on the screen in red.

To create the line-sinks along Kelso Creek, place the mouse pointer at the 400 foot elevation point and press the left mouse button. Move the mouse along the stream (the pointer on the screen will move) and press the left mouse button where end points are to be created. When subsequent points are selected, line-sinks appear in yellow with calculated heads printed at their centers. If an error is made while creating line-sinks, press <Esc> to abort. Press the <F3> key when you are finished creating line-sinks for Kelso Creek. The line-sink string appears in yellow with crosses to mark end points. **Figure 10** shows an example of line-sink representation of Kelso Creek.

When finished creating line-sinks, press <Esc> to return to the Element Menu.

NOTE:
CZAEM cannot handle more than 150 line-sinks. You are advised to keep the number of line-sinks much lower, particularly for initial modeling runs.

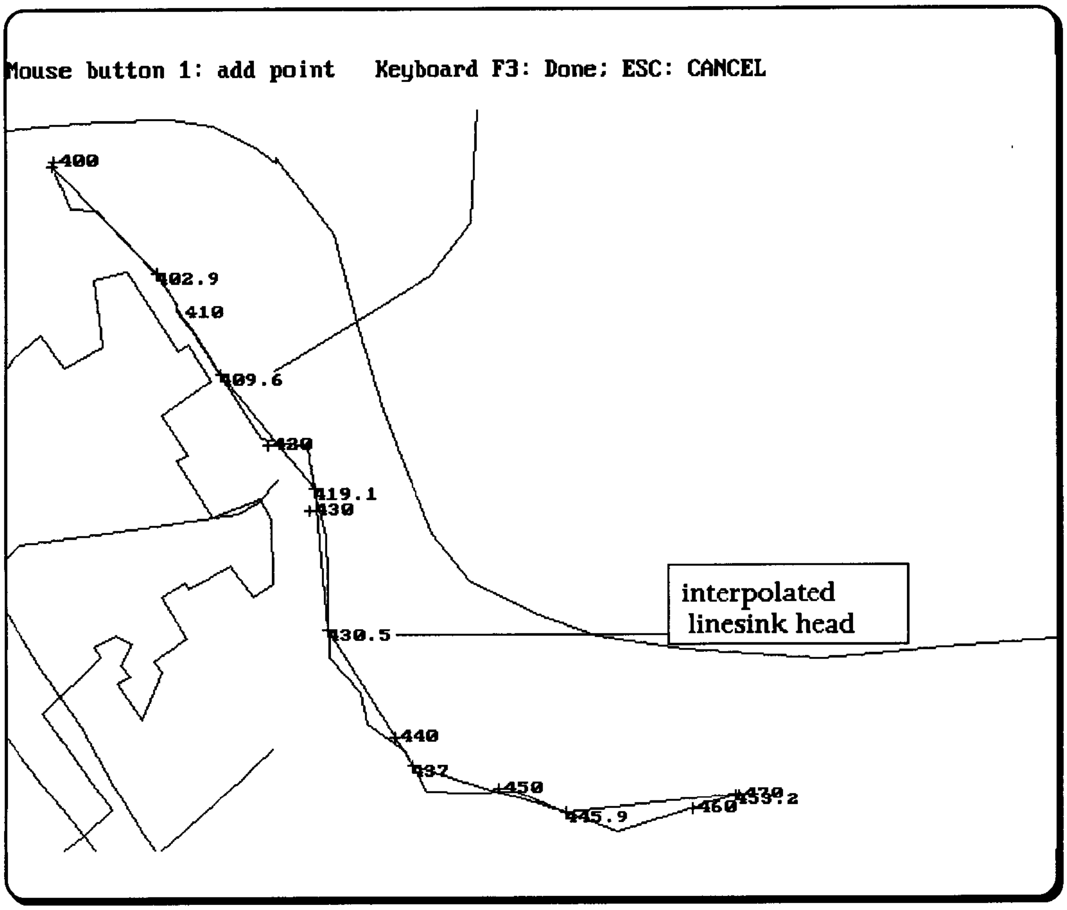

Figure 10 Line-sinks created along Kelso Creek.

Creating Wells

Well elements are created from predefined point sets. A well element in the middle of the five wells will be used to represent the combined pumping rate of the five wells.

- Press <W> or select **Well**. The screen will show all presently defined map features. The dark red plus signs are the points which can be selected for creation of well elements. <Page Down> to get see the individual wells of the wellfield.

- Move the mouse cursor toward the plus sign representing the Vincennes wellfield. The feature will be highlighted and its name will appear at the top of the screen.

- Press the left mouse button. You will be prompted to select the well position: "mouse button 1: well......."

- Place the cursor in the middle of the points and press the left mouse button. The well element will be added as a yellow plus sign. You will then be prompted for the well discharge.

- Type: "467000". A total pumping center discharge of 467,000 ft^3/day. A positive discharge takes water out of the aquifer.

- Press <Enter>. You will be prompted for the well radius.

- Type: "1" for 1 ft. Data can be entered in any units, but they must be consistent. Press <Enter> again.

- If an error is made, press <Esc>.

- Press <F3> when finished. You will be prompted for another well feature.

- Press <Esc> to return to the main GAEP menu.

Aquifer Module

The aquifer module is used to add aquifer parameters and a recharge rate to the CZAEM input data file. To access the AQUIFER module, select the aquifer command <A> in the main GAEP menu (**Figure 11**). Enter aquifer data by selecting the parameter in the AQUIFER menu and typing in the value. For example, to enter a base elevation of 330 ft, press <B>. You will be prompted for the elevation of the parameter. Type "330" and press <Enter>. GAEP will update the value on the screen. The units used for the aquifer parameters and the recharge must be consistent. The length unit used for the permeability (hydraulic conductivity) must be the same as that used when digitizing coordinates (usually feet). The time unit used to define the average areal recharge rate (usually days) must be consistent with that used to define permeability. This version of GAEP assumes "days" are used for the unit of time!

Right:	Permeability = 100	(feet/day)
	Recharge = 0.000913	(feet/day)

Wrong: Permeability = 3.527E-2 (cm/sec)
Recharge = 4.0 (inches/year)

```
Base Thick Permeability pOrosity UniformFlow reFerence Rain Quit


Aquifer Parameter Settings:                        Current Directory:

                                                   C:\WHAEM\DAT

Base Elevation:          0.0                       Current Map File:
Thickness:             100.0                       Vincenne.dm
Permeability:            100                       Vincennes
Porosity:                0.2                       Current Element File:
Uniform Flow:  Q0          0

               alpha   0.00                        Memory available:   368800

Reference point:           X    0.0
                           Y    0.0
                        Head    0.0

Rain element is not defined                Option Settings:

                                           Unit Conversion:               M->FT
                                           Digitizer Mode:                DIGITIZER
                                           Video graphics mode:           COLOR

                  <f1> help  <esc> return to previous menu
```

Figure 11 GAEP Aquifer Submodule.

Enter the aquifer data for Vincennes as defined in **Table 1**. Most of the parameters are self-explanatory. As a first approximation, the reference point is assigned the average head in the study area, and located far away from the study area. Coordinates for this point are entered as UTM values (meters); they will be transformed to coordinates with respect to the "model origin" when creating a CZAEM input file. See the CZAEM User's Guide (USEPA, 1994) for additional discussion of the reference point.

The thickness parameter can be used to indicate the presence of a confining layer and subsequently define the top of the aquifer. In an unconfined aquifer, the thickness parameter is often set artificially high to assure that flow is unconfined throughout the domain of interest. This does not affect the computations; see Strack (1989), section 8, for a full discussion. For the outwash aquifer near Vincennes, the outwash is less than 100 feet thick.

Table 1 Aquifer data for Vincennes case study.

Base Elevation	330
Thickness	100
Permeability (hydr.cond.)	350
Porosity	0.20
Reference Point X:	0
Y:	656160
Head:	410

The "rain element" is the only item in the aquifer module which is defined graphically. To define a "rain element" (circular recharge area):

- Press <R> at the AQUIFER menu. The Vincennes digital map will appear on the screen. You may have to zoom out and shift the figure to get all elements on the screen.

Use the mouse (which controls the cursor) to define the rain circle as follows:

- Place the cursor at the center of the near-field (City of Vincennes).
- Click the left mouse button.
- Move the cursor away from the center. The rain circle expands as the cursor moves.
- Click the left mouse button when all of the line-sinks are inside the rain circle. In our case only the line-sinks on Kelso Creek are present, but imagine that they cover other streams.
- Answer "yes" to accept. You are now prompted for the recharge rate.
- Type "0.0032" for a recharge of 0.0032 ft/day (14 in/yr). Press <Enter>. You will be returned to the AQUIFER menu. If an error is made (before pressing <Y> or <N>), press <Esc> to return to the AQUIFER menu.
- When finished press <Esc>. Return to the main GAEP menu.

Saving the CZAEM Input Data on Disk

The analytic elements and aquifer data created above need to be saved to disk as an input file for CZAEM. File operations take place within the file menu <F> of the main GAEP menu.

Once you are in the file menu:

- Select SaveElem to save the elements and aquifer data to disk. The analytic element file will contain line-sinks, well elements, aquifer properties, the recharge rate, and (optionally) map features such as roads.

- At the prompt, change the name of the file by typing in a new filename and press <Enter>. DO NOT use the name vincenne.dat! This would overwrite the existing data file.

- You will then be prompted for the UTM coordinates of the model origin.

- Type in "450000 4280000" and press <Enter>. You will be returned to the main menu in GAEP.

The coordinates of all elements saved to the file will be in feet from the model origin. If you want the coordinates to be in meters, select Options from the main menu. Then select <M> to change the units to metric. See the GAEP Reference Manual for more details on the Options menu. Exit the program.

Imaging Line-sinks Generated in GAEP

In some cases, the assumption of a homogeneous aquifer of infinite lateral extent is so far from reality that the modeler needs to consider other options. In W*h*AEM , the method of images can be used to model a no-flow boundary condition along an outwash valley wall (**Figure 12**). For a complete discussion of image theory, see Strack (1989), pages 27-33.

Note:
The following section is designed to give the user practice in creating an image data file; however, it is not necessary to complete the exercise. A complete file VINIMAGE.DAT is included on your disk.

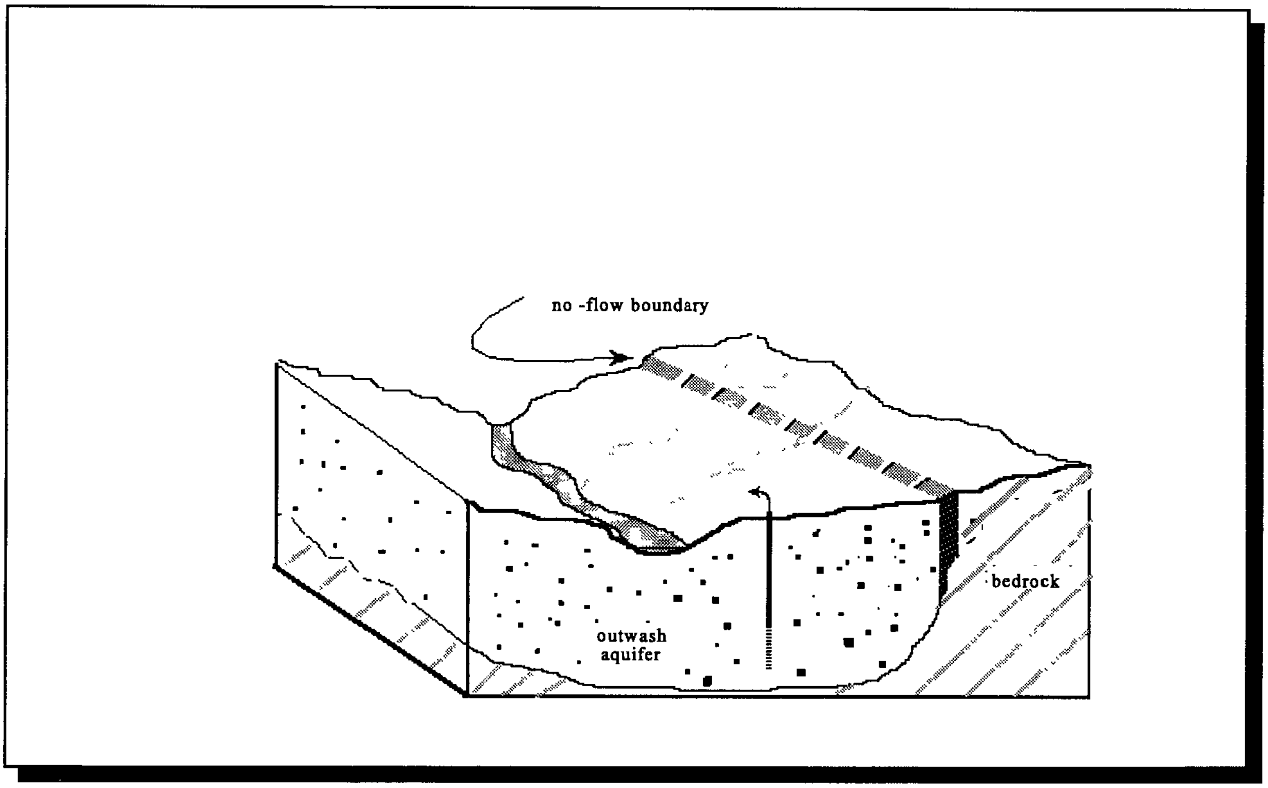

Figure 12 Geologic setting of an outwash aquifer.

Near the city of Vincennes, this linear no-flow feature can be used to model a sandstone bedrock/outwash interface that occurs to the south and east of town. In GAEP, the method of images is implemented by defining an image line through two points on the no-flow boundary of the aquifer, as shown in **Figure 13**. The program then images the existing line-sinks across that boundary and repositions the rain circle so that it is centered on the image line. Any line-sinks occuring on the east side (the image side) of the line are removed prior to imaging. The application of this imaging technique (introducing a no flow boundary) is illustrated by the following steps, from the main menu in GAEP:

- Select **F**ile or press <F>. — We will first load the completed CZAEM input data file (without images) -VINCENNE.DAT.
- Select **R**ead**N**ew or press <R>.
- Type: "VINCENNE".
- Press <Esc>. — This will bring you back to the Main Menu of GAEP.

- Select **Element** or press <E>.

- Go into **View**<V> and identify Mantle Ditch on the southeast side ot the city.

The bedrock/outwash boundary is nearly parallel to this stream just east of the ditch. We will create an image line along this stream by the following steps:

- **Select Image** or press <I>.

  The screen will shift to a graphics image of the element file (VINCENNE.DAT). The following prompt will be displayed at the top of the screen:

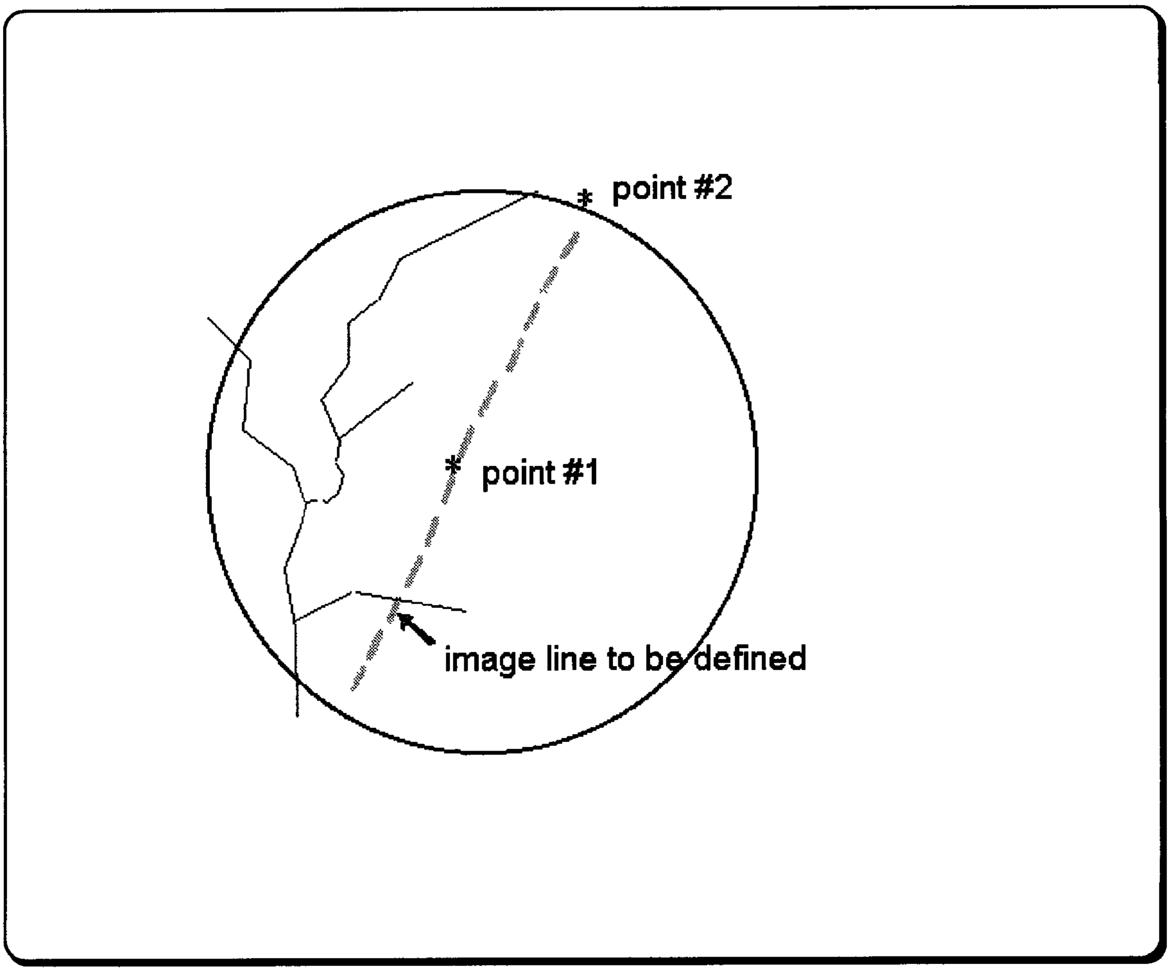

Figure 13 Outwash boundary within regional layout.

<u>`Select image origin and press mouse button`</u>

- Move the cursor south of town, on Mantle Ditch. Press the left mouse button to select the first image line point. The following prompt will appear at the top of the screen:

Select the second point on image line and press left mouse button.

- Move mouse to the NE and notice the GAEP image line .
- Rotate the line so that it reasonably approximates the position of the rock outcrop (roughly parallel to Mantle Ditch and Wabash River).
- Click the mouse button to define the image line, as shown in **Figure 14**.

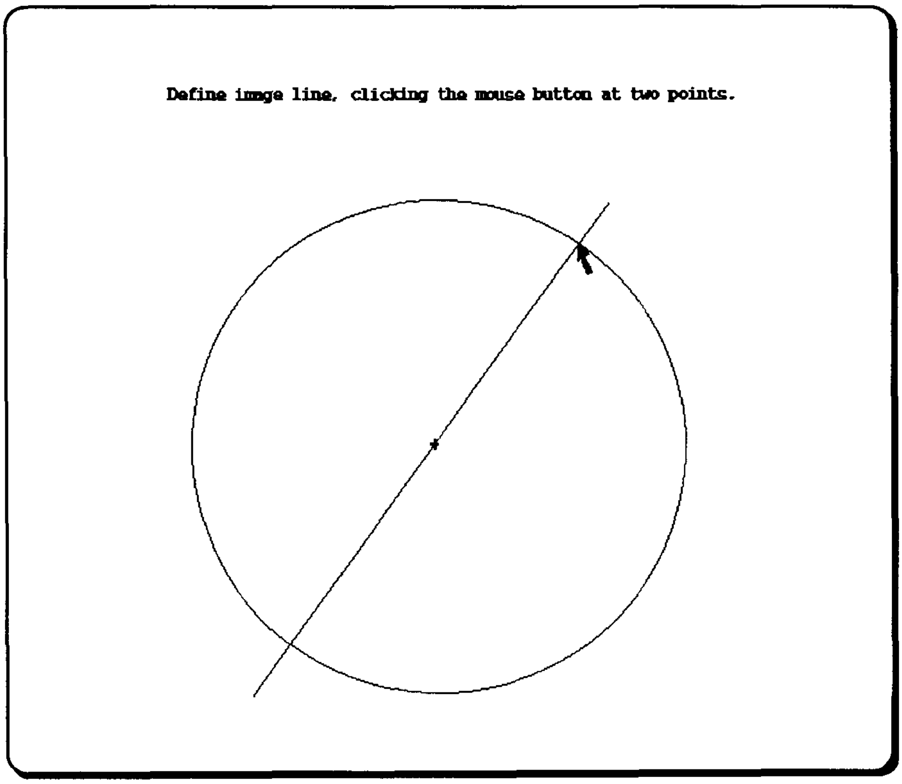

Figure 14 Defining the image line with mouse control.

Image line OK?

- Type: "yes". Next you will see a new rain circle in red, centered at the first image line point.
- Use the mouse to open the circle so that it covers the image domain and the "real" linesinks to the left of the line, as shown in **Figure 15**.

- Click left mouse button.

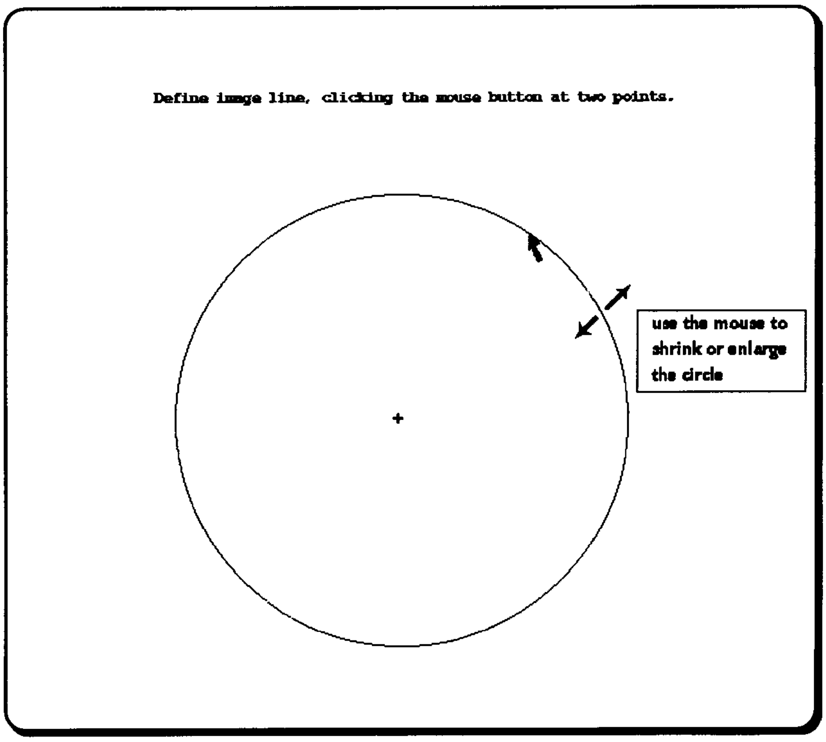

Figure 15 Defining rain circle with mouse control.

Rain circle OK?

- Type: "yes".

Enter Recharge (same units as permeability):

- Use same recharge rate as before: (0.0032)

Enter reference head:

- As before, use average heads in modeled area: (405)

Now you are back in the Element menu. Press <V> to View the "image" and "real" linesinks. The image linesinks are plotted in light gray. From here you may return to the main GAEP menu, press <F> (File) and save the data in a new input file. We will omit these steps, because a complete file with images has already been provided for use in the \WHAEM\DAT subdirectory named VINIMAGE.DAT. Do not replace this file by saving new data to this file name. Exit the program.

Model the Site with CZAEM

You are now ready to run the Capture Zone Analytic Element Model (CZAEM) to determine the capture zones for the Vincennes City well field. The tutorial will take you through basic operations of using CZAEM for wellhead protection. A more complete discussion is found in the CZAEM User's Guide (USEPA, 1994). A reference guide to CZAEM commands can be found in Appendix B.

There are eight steps to performing a wellhead protection capture zone analysis with CZAEM:

1. Read the input file: VINCENNE.DAT.
2. Visually check the input file.
3. Solve the ground-water flow problem.
4. Generate a grid for contouring piezometric heads.
5. Evaluate the ground-water solution.
6. Create capture zones.
7. Save solution to disk.
8. Test hypotheses by adjusting input data and start over.

To start the CZAEM program, change your working directory to the \WHAEM directory:

```
C:\>CD \WHAEM
C:\WHAEM>CZ
```

At the introductory CZAEM information screen, press <Enter> to enter the main module of CZAEM. When the main module command line appears on the screen, you are ready to read in an input file produced with GAEP. Like GAEP, CZAEM is a modular program. The most important operational difference is the user interface. CZAEM is a command line program and requires direct keyboard input. CZAEM commands can be abbreviated to the first few letters (enough to make them unique). This W*h*AEM tutorial is only a brief introduction to the

General CZAEM Hints

All commands may be abbreviated to the first few letters.

< ? > context sensitive help.

"**return**" previous menu, exit graphic screen.

<F3> previous command.

Cursor control -

The mouse is the default cursor control device.

The mouse buttons are <u>not</u> active.

<backspace> will delete commands when not in graphics mode.

< (the left angle bracket key) > will delete commands while in graphics mode.

program. For more information about the use of CZAEM, the reader is referred to the more extensive discussion in the separate CZAEM Users Guide (USEPA, 1994).

Read CZAEM Input File: VINCENNE.DAT

The CZAEM input file created by GAEP is an ASCII file which contains the data describing the flow problem as well as the CZAEM commands needed to process the data. The file is read into CZAEM through the SWITCH module. GAEP has added the instructions to the VINCENNE.DAT file to solve the ground-water flow problem, create a grid for contouring the piezometric surface, and to define some parameters for capture zone delineation.

Files created by GAEP are stored in the c:\WHAEM\DAT\ subdirectory.. To read in the tutorial CZAEM input file:

- Type: "SWI VINCENNE.DAT "<Enter>.

As the file is being read in, the data will scroll across the screen quickly. When the end of the file is reached control will be returned to the keyboard (console) and the Main Command Menu (**Figure 16**) will appear on the screen.

```
\\\  Module=MAIN MENU           Level=0    Routine=INPUT            ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>        <MAP>                                        <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                     <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>     <PLOT>                                       <PAUSE>
<SOLVE>        <TRACE>                                      <RESET>
<CHECK>        <CURSOR>                                     <PSET>
                                                            <STOP>
```

Figure 16 CZAEM Main Menu.

Check the Input File

At this point we will inspect the input data file through on-screen graphics. The window size (domain to be plotted on the screen) will be whatever was set as a User Window in GAEP (see Appendix A, GAEP reference manual). We will store the current window, expand the window to include all elements in the file, and look at this model layout using the following sequence of commands.

- Type: "WINDOW PUSH"<Enter>. This stores the current window settings into the buffer.

- Type: "WINDOW ALL"<Enter>. This defines a new window including all elements.

- Type: "LAY "<Enter>.

The user can examine the distribution of linesinks, as well as the road and well field locations for the area being modeled. The screen image should look like **Figure 17**.

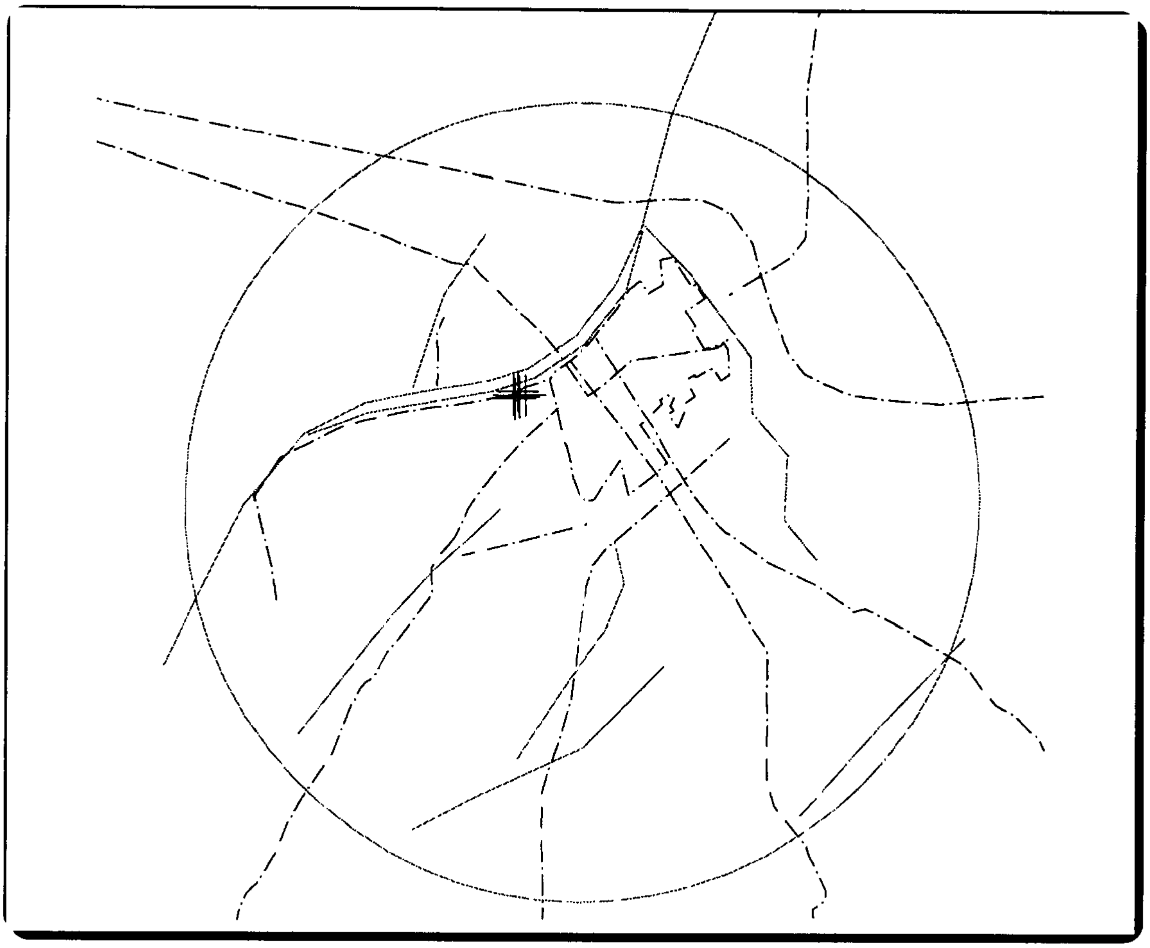

Figure 17 Layout of the entire VINCENNE.DAT data file - window all. (Large overlapping plus signs indicate the well field; they will appear smaller on the screen).

The solid white lines and the small white square are the linesinks and the well element defined earlier in GAEP. The dark red dashed-dotted lines are roads and the city boundary. <Enter> to return to command mode.

Turning map on and off

You may want to turn off the plotting of the roads and wells to reduce visual clutter on the screen. To do this, at the MAIN MENU:

- Type: "MAP"<Enter>. — This will place you in the MAP module.
- Type: "PLOT OFF "<Enter>. — Plotting of the road and well file is turned off.
- Type: "RET "<Enter>. — You are back at the MAIN MENU.
- Type: "LAY "<Enter>. — Again, check the image. To turn the map plotting routine back on, type "PLOT ON" in the MAP module.

Changing the Window

The "window" is the area displayed on the screen and is defined by two coordinate pairs: the lower left and the upper right corner of the area to be displayed. A new window may be defined by typing: "WIN x1, y1, x2, y2" where the first coordinate pair is the lower left corner, and the second coordinate pair is the upper right hand corner of the domain.

The WINDOW PUSH command will save the current window. The WINDOW POP command will retrieve the last window pushed into memory. It is not saved! Only the windows pushed into memory can be retrieved (popped). We advise the user to select a set of windows for the project and take the time to write these coordinates down.

The initial window can be restored with GAEP's User Window utility:

- Type: "WIN POP "<Enter>. — This retrieves the last window in memory (selected in GAEP).

You will be restoring a window with lower left coordinates of (4752, -9327) and upper right coordinates of (21625, 5545).

- Type: "WIN"<Enter>. — The current window coordinates are printed on the screen.
- Type: "LAY "<Enter>. — To see a close-up of the well field, as in **Figure 18**.
- Press <Enter> again. — You will be back at the main menu.

In order to be complete, this tutorial includes instructions for interactively issuing the **solve** command (even though this command has been included in the Vincennes file) as well as instructions and explanation of the **grid** routine. Because this has been automated, the reader may elect to skip these topics. You should be aware that any time you change the number of elements, or aquifer properties, or the reference point, you will need to re-solve and re-grid. Any time you change the window, or desire a different resolution on a contour plot, you need to re-grid.

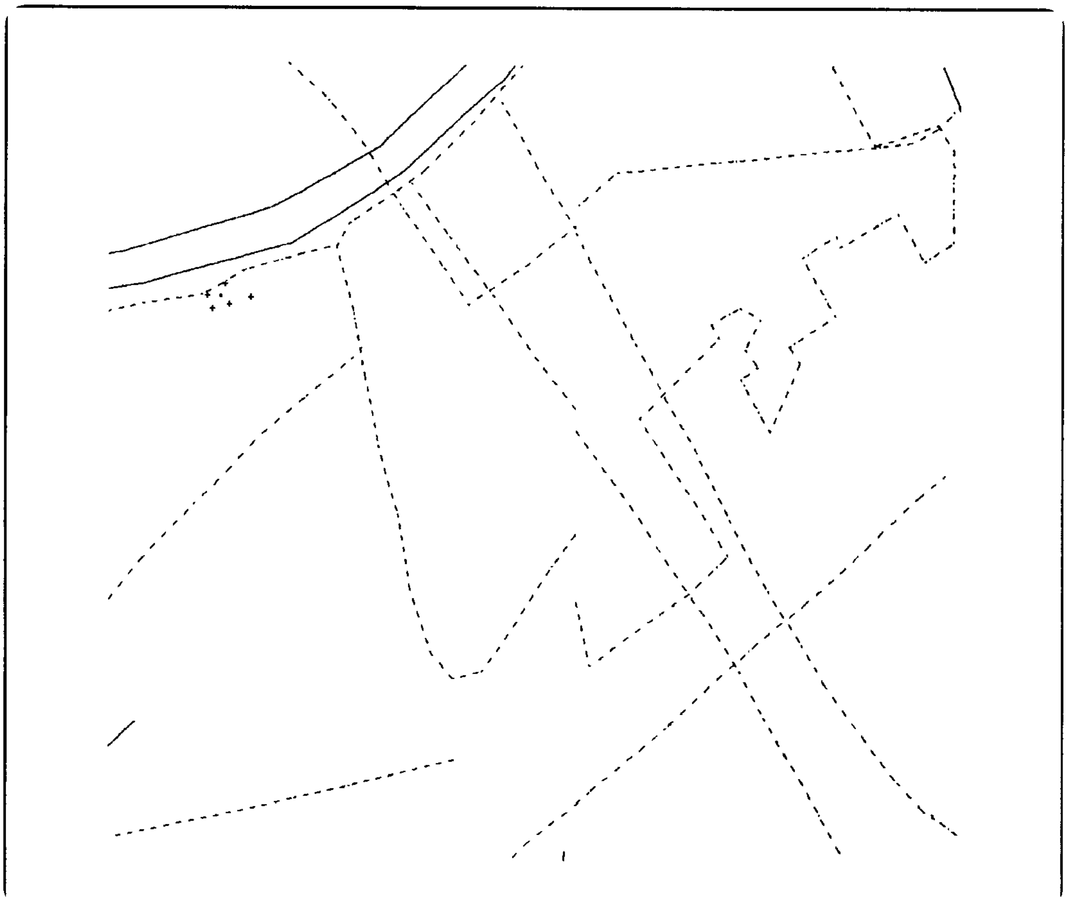

Figure 18 Zoomed-in view of layout of VINCENNE.DAT file, including map features.

Solve the Ground-Water Flow Problem

(Optional: already done in vincenne.dat!)

Once the input data appears correct, you proceed to solve the ground-water flow problem. At the MAIN MENU:

- Type: "SOLVE "<Enter>. When SOLVE is complete, the MAIN MENU will appear on the screen.

Generate Grid for Contouring Heads

(Optional: already done in vincenne.dat)

In order to generate a contour plot of piezometric heads, we will calculate the head at a grid of evenly spaced points in the current window (domain seen in layout). At the MAIN MENU:

- Type: "GRID 30 "<Enter>.

Important:
Anytime you change the window, you will have to repeat the grid procedure if you want to view contours inside the new window. This can be done by typing "GRID 30" at the MAIN MENU after the new window has been defined. The parameter "30" specifies 30 grid points along the horizontal axis of the plot and an appropriate number of points along the vertical axis to obtain an even grid spacing. You may create higher resolution plots by increasing this number, but this will take a little more time.

View Piezometric Contours in PLOT Module

To inspect the computed piezometric head surface, at the MAIN MENU:

- Type: "PLOT "<Enter>. This will place you in the PLOT module. You will see the following prompt:

```
 <D>EFAULT [NUMBER OF LEVELS] <L>AYOUT
 (MIN LEVEL [INCREMENT {>0}][MAX LEVEL]
 (MAX LEVEL [DECREMENT {<0}][MIN LEVEL]
MIN. LEVEL=   3.963030E+02 MAX. LEVEL=  4.448840E+02
```

- The "min. level" and "max. level" values on your screen may differ somewhat from those shown here.

- You can either type, "D" <Enter>, to accept the default contour interval or adjust the settings by supplying the minimum contour level to be plotted, the desired contour interval (optional), and the maximum contour level to be plotted (also optional). For example:

- Type: "400 5 "<Enter>. The number of contours will appear on the screen, along with the following message: THERE ARE 9 LEVELS: PRESS ENTER.

- Press <Enter>. The screen will display a contour plot of piezometric heads plotted from the lowest to highest head.

Use Pathline Tracing in TRACE Module

While potentiometric contours are the most familiar image of a ground-water flow solution, an alternative way to evaluate flow is to use the pathline tracing functions of CZAEM to see how ground-water moves throughout the model domain. This is done in the TRACE module, which allows the modeler to view a layout of the study area (with or without piezometric contours) and trace pathlines from any point in the user window. From the main menu:

- Type: "TRACE "<Enter>. The TRACE menu appears on the screen.
- Type: "LAY "<Enter>. A "layout" will appear of the current window without the piezometric contours. The linesinks, well, and cursor will be displayed on the screen along with menu selections at the top.

To draw pathlines place the cursor near the upper end of the Mantle Ditch linesinks (the lower left white line on the screen):

- Type: "TRACE 380 "<Enter>. A pathline will be drawn in purple, with the pathline starting at the cursor point (x,y) defined by the cursor location, and elevation equal to 380 feet. Markers cross the pathline at one year intervals.

Place the cursor between City Ditch (the white line straight south of the well) and the well field.

- Type: "TRACE "<Enter>. A pathline will again be drawn, this time starting at the default value of the top of the aquifer at the cursor location x,y. If you wish to do so, continue to move the cursor around the domain, starting pathlines in various locations to "get a feel" for how ground water is moving. Type "RET" to return to the main module.

NOTE:
The tic marks on the streamlines indicate ONE YEAR ground-water travel time intervals (defined earlier in VINCENNE.DAT).

Initial Capture Zone Analysis

It is recommended to use the option WGEN (Well Generate) in the TRACE module to perform initial analysis of capture zones. The WGEN routine is relatively fast and can be used to get a clear picture of ground-water flow patterns near the well. After modeling the site and selecting the most important scenarios, final capture zones can be prepared using the SUBZONE and TIMEZONE commands in CZAEM. These commands are explained later in this section.

At the MAIN MENU:

- Type: "TRACE" <Enter>. You will be placed in the TRACE module.
- Type: "SET" <Enter>. This moves you into the SET submodule of TRACE and allows you to alter the tolerance and other settings..
- Type: "BACK ON" <Enter>. This changes the setting of the tracing routine to trace against the direction of flow. A message will tell you that the tracing direction is set to backward.
- Type:"MAXSTEP 100" <Enter>. Sets the step size along a pathline to 100 days.
- Type: "MARK TIME 730:<Enter>. Changes the time markers on the streamlines from one year to two years (730 days).
- Type: "TERM TIME 3650" <Enter>. This will terminate the pathline trace after 3650 days or 10 years.
- Type: "RET" <Enter>. You will be returned to the main menu of the TRACE module.
- Type: "LAY" <Enter>. Menu choices and the layout of the element features will appear on the screen, with the cursor located near the middle of the window. Move the cursor to the well.
- Type: "WGEN 16" <Enter>. This will trace 16 pathlines from the well and will define the shape of the capture zone. The pathlines will show up in purple on the screen. While the lines are being traced the cursor will disappear. When all of the lines are complete the cursor reappears. See **Figure 17.**

Note:

The well receives part of its water from the Wabash River. The tick marks indicate 2-year increments. This set of stream line and tick marks provide an approximation of a more detailed 10-year time-of-travel capture zone that will be generated later with the SUBZONE and TIMEZONE commands.

- Type: "MENU" <Enter>. You will be returned to the TRACE module screen.

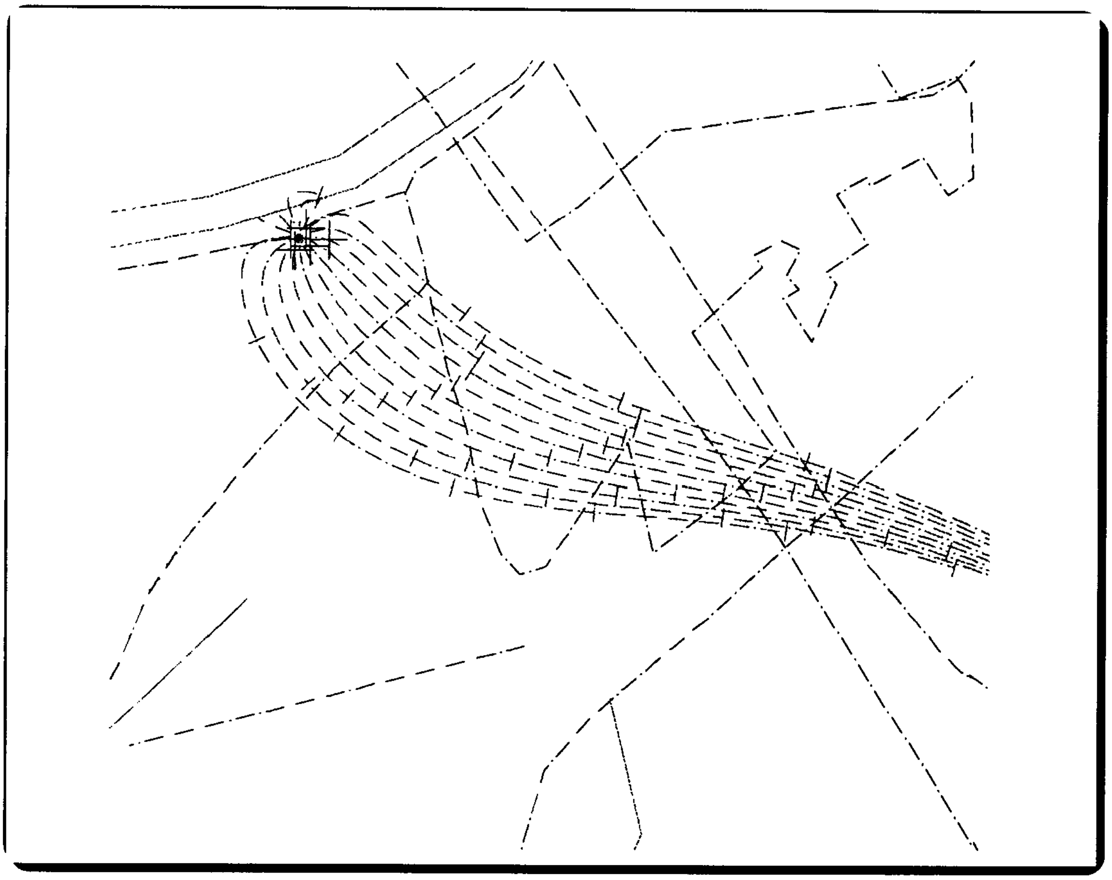

Figure 19 Steady state pathlines backward traced from the Vincennes wellfield (WGEN 16 command in TRACE module; tic marks every two years).

Hypothesis Testing

During the initial stages of modeling the user is comparing modeled piezometric heads with heads observed in the field. Evaluating the differences between modeled and observed heads is the first step in model calibration and is one way to judge the adequacy of the ground-water flow model. This procedure requires good hydrologic insight and substantial modeling skills. Developing these skills is outside the scope of this tutorial.

The current CZAEM model of the Vincennes area (VINCENNE.DAT) predicts heads to be higher than have been observed. In addition, it lacks the necessary degree of realism because it does not include the transition in hydraulic conductivity from the highly permeable channel deposits along the Wabash River to the sandstone bedrock outcrop east of town. Previous USGS modeling efforts in this area dealt with this transition as a no-flow boundary (Shedlock, 1980). As discussed earlier in this tutorial, GAEP allows the modeler to introduce no-flow boundaries by applying the method of images. The file VINIMAGE.DAT includes an image line that approximates this rock outcrop. The solution to VINIMAGE.DAT is in much better agreement with field data than VINCENNE.DAT which ignores this feature.

We will continue our tutorial using the file VINIMAGE.DAT.

Read in New File: VINIMAGE.DAT

- Return to the main menu by typing "RET"<Enter>.
- Type: "RESET" <Enter>.
- Answer "Y" <Enter>.
- At the main menu type: "SWI VINIMAGE.DAT" <Enter>.

Create Final Capture Zones

When a set of satisfactory models have been obtained for the ground-water flow around the Vincennes well field, you can generate capture zones. For this example we will use both of the two different commands to delineate a capture zone. The first command, "subzone," delineates the complete capture zone of a well. The second command, "timezone," will draw isochrones within the capture zone. The water in the area bounded by the isochrone is captured by a well within a specified time period. Each command can be used to identify a wellhead protection area for a well field. A solution to the ground-water flow problem must be in memory before creating capture zones.

The first step in creating a capture zone is to make sure that the well generates at least one stagnation point within the current window. The subzone routine that is used to create a capture zone searches the window for a stagnation point generated by the well. Use the WGEN command to verify that stagnation point(s) are inside the current window. For this purpose repeat the steps under the heading "Initial capture zone analysis" (See **Figure 20**).

Compare new streamline patterns with the previous WGEN without the image line.

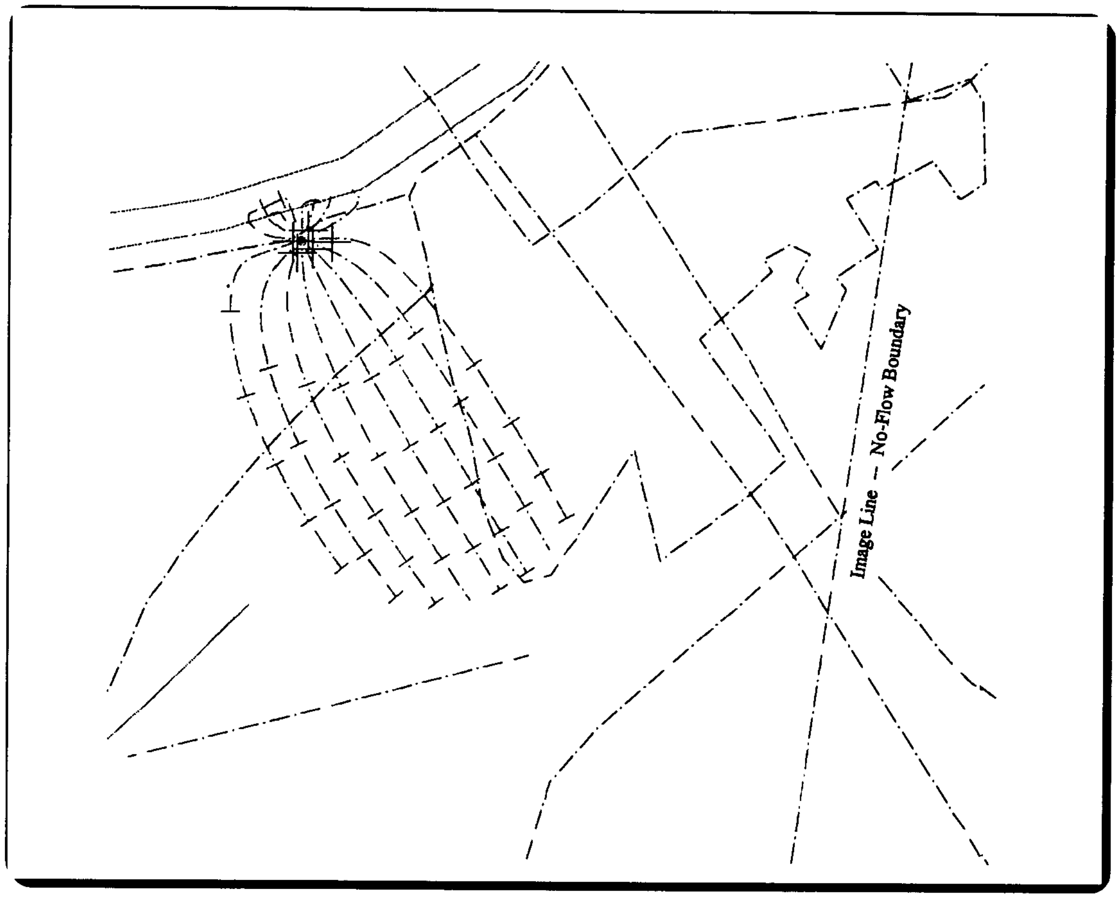

Figure 20 Modeled steady state path lines from the Vincennes wellfield (WGEN 16 command and the input data VINIMAGE.DAT).

Generate a Subzone Around the Well

For our case the window is suitable (includes all stagnation points for the well) so that we may proceed delineating the capture zone. This will be accomplished by first entering the CAPZONE submodule of TRACE and then typing SUBZONE. In the TRACE module:

- Type: "CAPZONE" <Enter>. — Information about piezometric head levels will appear on the screen.
- Type: "400 5" <Enter>. — This will plot contour levels in 5 foot intervals beginning at the 400 foot level. You will be told how many contours will be plotted.
- Again, press <Enter>. — An image of the elements and contour lines will appear on the

- Position the cursor on the well.

  screen along with the menu choices for the CAPZONE submodule.

- Type: "SUBZONE" <Enter>.

  The program goes through a three phase calculation procedure. This takes a few minutes. .

If problems occur at this point (for example, no stagnation point is found), messages will appear on the screen indicating the nature of the problem. If you receive an error message, type:" COM" and press <Enter> to get the menu for the CAPZONE submodule. If the subzone is ready, it will be plotted on the screen. The subzone you have created for the Vincennes well field should look like **Figure 21** .

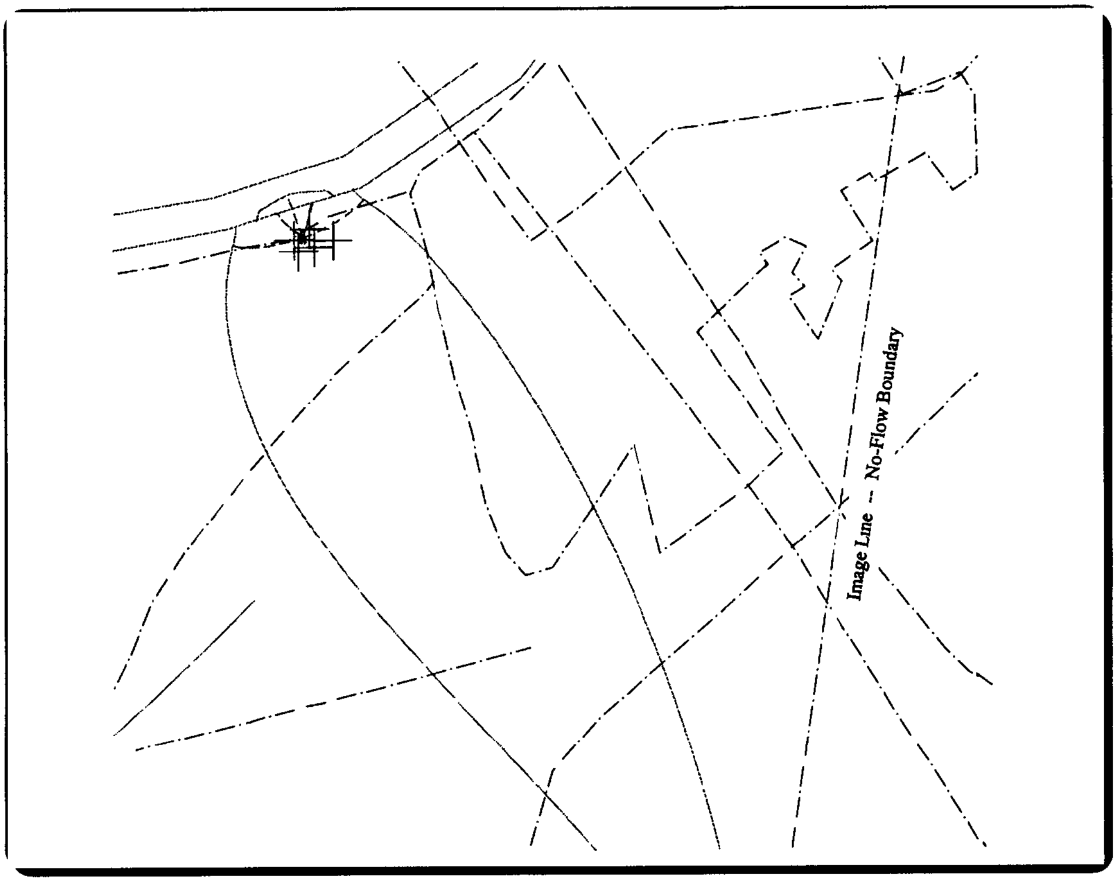

Figure 21 Modeled subzone for VINIMAGE.DAT showing source areas for the wellfield.

Generate Isochrones Around theWell

After the total capture zone has been delineated with the SUBZONE routine, you can compute steady state time-of-travel capture zones (TIMEZONES) for the well. Place the cursor on the well.

- Type: "TIME" <Enter>. You will be prompted to enter a minimum time, time step, and maximum time, or redraw last time zone, calculate the default time zone, or exit.

- Type: "730 730 3650" <Enter>. Start at 2 years, step 2 years, stop at 10 years.

The time zone calculations may take a while to compute. When the solution is complete, the timezones will appear on the screen. The steady state time-of-travel (TOT) capture zones for the well field are illustrated **Figure 22**.

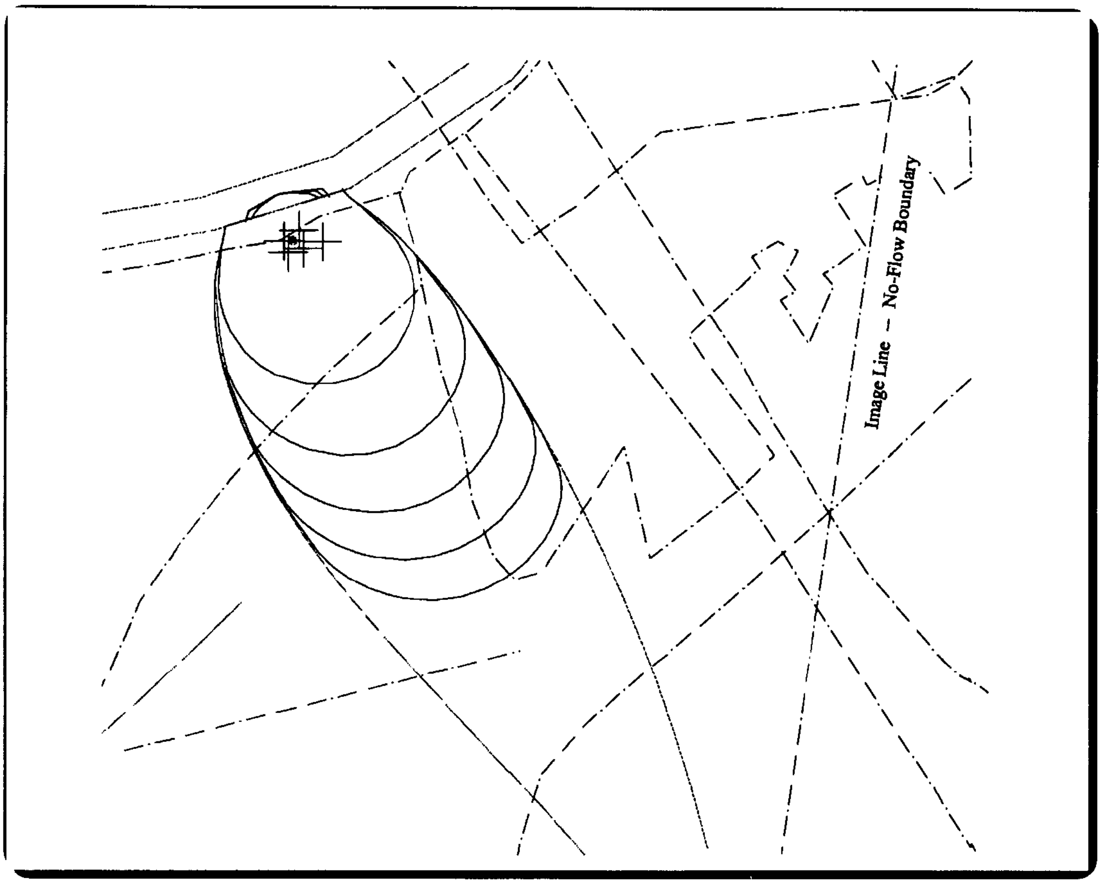

Figure 22 Steady state time-of-travel capture zones (2-yr intervals) including no-flow boundary.

- Type: "RET" <Enter>. To clear the screen and return to the TRACE MENU.
- Type: "RET" <Enter>. To return to the MAIN MENU.

Sending Graphics to the Printer

In Chapter 1, it was explained that a small batch file needs to be run outside of the program (from the DOS prompt) which sets the system up for producing hard-copy output. Before generating the capture zone printouts, be sure that the map file shows all of the features that you need for interpretation of the results. (Be sure that the PRINTER.BAT file has been run during set-up.)

After using WGEN to identify the solutions which are most useful, you should be ready to generate pictures of the time zones, subzones or both. There are two ways of doing this in CZAEM:

1) Recalculate the time zones and subzones with graphics output routed to the printer (or a print file).
2) Save the time zone buffers to disk (binary file) and read them back when you are ready to generate graphics on the printer.

The following discussion outlines the steps needed to generate images on the printer using each approach. We assume that you are currently in the CAPZONE module.

To regenerate capture zones and then send graphics to the printer:

• Type:"`RETURN`" <Enter>.	This will put you at the TRACE module.
• Type:"`PSET`" <Enter>.	The PSET module allows you to control various settings relating to the graphics displayed by CZAEM.
• Type:"`PRINTER`" <Enter>.	This redirects graphics output to the printer.
• Type:"`RETURN`" <Enter>.	You are now back in the TRACE module.
• Type:"`CURSOR OFF`"<Enter>.	This is very important! This is required in order to go through the next steps.
• Type:"`CAPZONE`" <Enter>.	
• Type the contour levels: "`400 5`".	
• Press <Enter> twice.	No graphic appears. The program waits for the coordinates of the well followed by the "TIMEZONE" command.

- Type:"8549 2234 TIMEZONE". The well is centered at the point (8549,2234). Wait for the calculations to be complete, then enter the minimum, increment, and maximum time:
- Type:"730 730 3650" Wait for the calculations to be completed.
- Type:"**RETURN" <Enter>** This closes the Postscript file (PLOT.PS) which may be copied to any Postscript printing device or it sends the file to the HP Laser Jet III printer.

Alternately, if you would like to save the image buffers and then read them back later, you would have to do the following: (This assumes that you are looking at a graphics screen in the CAPZONE module.)

- Type: "CSAVE"<Enter>. To save the subzone and timezone buffers to disk. (This command is not documented on the menu.)
- Enter a filename, (e.g., "vincenne.cap" <Enter>).
- Type: "RETURN"<Enter>. To get to the TRACE module.
- Type: "PSET"<Enter>. To re-route the graphics to the printer.
- Type: "PRINTER"<Enter>.
- Type: "RETURN"<Enter>. You are now back at the trace module.
- Type: "CURSOR OFF"<Enter>. Do not forget this!!!
- Type: "CAPZONE"<Enter>. At the prompt enter the contour levels.
- Type: "400 5" (or <L>). To either print the contours or simply the layout.
- Type: "CREAD"<Enter>.
- To re-load the buffers enter the filename at the prompt (e.g., "vincenne.cap") (This command is not documented on the menu.)
- Type: "8549 2234 TIMEZONE".
- Type: "730 730 3650". To contour from the two to the ten year time-of-travel capture zones in two year intervals.
- Type: "RETURN"<Enter>. To plot to file PLOT.PS or to send the graphics directly to your HP Laser Jet printer, depending on

your setup.

Exit CZAEM

When you are finished with your work in CZAEM, return to the MAIN MENU.

- Type: "STOP" <Enter>.

REFERENCES

Anderson, M.P., and W.W. Woessner, 1992. Applied Groundwater Modeling: Simulation of Flow and Advective Transport, Academic Press, San Diego, CA.

Bakker, M., and O.D.L. Strack, in preparation. Capture zone delineation with analytic element models, *Water Resources Res.*

Bear, J. and A. Verruijt, 1987. Modeling Groundwater Flow and Pollution, Reidel, Dordrecht.

Dupuit, J., 1863. *Études Thèoriques et Practiques sur le Mouvement des Eaux dans les Canaux Decouverts et à Travers les Terrains Permèables.* Dunod, Paris, 2nd edition.

Forchheimer, P., 1886. Ueber die ergiebegdeit von brunnen-anlagen und sickerschlitzen. *Architekt. Ing. Ver. Hannover*, 32:539-563.

Haitjema, H.M., 1985. Modeling three-dimensional flow in confined aquifers by superposition of both two- and three-dimensional analytic functions. *Water Resour. Res.*, 21(10):1557-1566.

Haitjema, H.M., 1987. Comparing a three-dimensional and a Dupuit-Forchheimer solution for a circular recharge area in a confined aquifer. *J. Hydrology*, 91:83-101.

Haitjema, H.M., in preparation. Groundwater Modeling Using the Analytic Element Method, Academic Press.

Kelson, V.A., H.M. Haitjema, and S.R. Kraemer, 1993. GAEP: A geographic pre-processor for groundwater flow modeling. *Hydrological Science and Technology, 8(1-4):74-83.*

Shedlock, R.J., 1980. Saline water at the base of glacial outwash aquifer near Vincennes, Knox County, Indiana, USGS Water Resources Investigation 80-65, 52pp.

Strack, O.D.L., 1984. Three-dimensional streamlines in Dupuit-Forchheimer models. *Water Resour. Res.*, 20(7):812-822.

Strack, O.D.L., 1989. Groundwater Mechanics. Prentice Hall, Englewood Cliffs, New Jersey.

Strack, O.D.L., 1992. A mathematical model for dispersion with a moving front in groundwater, *Water Resources Res.*, 28(11):2973-2980.

Strack, O.D.L., and H.M. Haitjema, 1981a. Modeling double aquifer flow using a comprehensive potential and distributed singularities. 1. Solution for homogeneous permeabilities. *Water Resour. Res.*, 17(5):1535-1549.

Strack, O.D.L., and H.M. Haitjema, 1981b. Modeling double aquifer flow using a comprehensive potential and distributed singularities 2. Solution for inhomogeneous permeabilities. *Water Resour. Res.*, 17(5)1551-1560.

USEPA, 1994. CZAEM User's Guide: Modeling Capture Zones of Ground-Water Wells Using Analytic Elements, by O.D.L. Strack, E.I. Anderson, M. Bakker, W.C. Olsen, J.C. Panda, R.W. Pennings, D.R. Steward, Robert S. Kerr Environmental Research Lab Research Report, EPA/600/R-94/174.

APPENDIX A. GAEP REFERENCE

Written by:
Victor A. Kelson
Groundwater Modeling Laboratory
School of Public & Environmental Affairs
Indiana University, Bloomington
Copyright (c) 1993,1994

This Chapter of the W*h*AEM Manual describes the operation of the GAEP program with a number of different hardware configuration systems. The GAEP program was developed by Vic Kelson at the SPEA Groundwater Modeling Laboratory, Indiana University, Bloomington. The author acknowledges Phil DiLavore for his work on the initial design of GAEP. Thanks also to Jack Wittman of IU and Dr. Stephen R. Kraemer of the USEPA/RSKERL-Ada for assistance and guidance with this work.

CalComp is a trademark of CalComp Inc.
Summagraphics, SummaSketch, MM1812, and Bit Pad Plus are trademarks of Summagraphics Corporation.
Microsoft and MS-DOS are trademarks of Microsoft Corporation
80386 and 80486 are trademarks of Intel, Inc.

Concepts

This Appendix describes the program GAEP (as implemented for the W*h*AEM product). It is not intended as a complete tutorial for GAEP, but as a user reference once the printed tutorial has been studied. GAEP (Geographic Analytic Element Preprocessor) is a program which greatly speeds and simplifies the process of developing ground-water flow models using the analytic element method. The purpose of GAEP is not to supplant the use of GIS tools for ground-water management, but to provide a specific set of functions which streamline the modeling process.

GAEP allows the modeler to create and manage a digital map of the hydrography of his study region, irrespective of any planned modeling work. The digital map area used should be large enough to cover any intended modeling work in a region. Once a digital map is prepared, it can be used on a variety of modeling projects, with the modeler creating his model from the digital map using the mouse on his computer. Other considerations, such as aquifer properties, can also be managed with GAEP. Once the modeler has defined his elements and aquifer properties, a data file for the CZAEM modeling program can be written, freeing the modeler from the process of manually editing CZAEM input files.

Files

GAEP manages two different types of data files, "Digital Map" files (*.DM) which are specifically defined for GAEP's use, and "Analytic Element" files (*.DAT), which are in a format compatible with CZAEM, but include additional information required for GAEP's interpretation.

Digital Map Files

GAEP creates and manages digital maps in a simple ASCII file format. These "Digital Map" files contain the locations of streams, lakes and background map features and the elevations of surface hydrologic features for use in creating analytic elements for CZAEM.

DIGITAL MAP FILES SHOULD NOT BE MODIFIED BY THE USER EXCEPT BY THE USE OF THE GAEP PROGRAM. USE OF OTHER METHODS FOR EDITING THESE FILES MAY RESULT IN LOSS OF DATA OR SEVERE MODELING ERRORS.

Analytic Element Files

Once the modeler has completed the definition of analytic elements and aquifer properties for a model run, GAEP saves the elements in a CZAEM-compatible "Analytic Element Data" file. Included in the analytic element file are a set of commands which solve the model, create a grid of heads for contour plotting and create a background map of geographic features. A view window selected in GAEP by the user is optionally set in the analytic element data file, as well.

ANALYTIC ELEMENT FILES SHOULD NOT BE MODIFIED BY THE USER EXCEPT BY THE USE OF THE GAEP PROGRAM. USE OF OTHER METHODS FOR EDITING THESE FILES MAY RESULT IN LOSS OF DATA OR SEVERE MODELING ERRORS.

Digital Map Features

GAEP allows the user to create a variety of digital map features which represent hydrologic features (which can then be used to create line-sink analytic elements) and other geographic features such as roads and political boundaries.

Stream Features

A stream feature represents a body of surface water, either a river or the perimeter of a lake. Stream features are used as the basis for the creation of LINESINK analytic elements. Stream features are entered as a set of points which define the geographic extent of the feature and a set of points where the water elevation is known (usually where a topographic contour crosses the feature). It is important for the user to determine which stream reaches should be included in the digital map.

Curve Features

A curve feature represents a road, geographic boundary, geological feature, contaminant site or other linear feature which will ultimately be used in CZAEM to orient the viewer. Typically, only major roads will be digitized, to ease the interpretation of the model without making the CZAEM screen too cluttered.

Point Set Features

A point set represents a set of wells, homes, locations of known water levels or other point features which will ultimately be used in CZAEM to orient the viewer. Point sets are also used as the basis for the creation of WELL analytic elements. The user should create a point set for each wellfield, set of known heads or other set of features.

Measurement Units

CZAEM Units

CZAEM works in a "dimensionless coordinate space"; that is, the modeler is responsible for maintaining a consistent set of units throughout a project. Before beginning a project, you should determine which unit of length to use (GAEP supports feet and meters) and which time unit to use (GAEP assumes days). Once the set of units has been decided upon, the modeler must maintain them in all his work.

UTM Coordinates

GAEP, however requires that geographic input data be entered in standard georeferenced coordinates. For convenience and consistency, the developers of the W*h*AEM package have selected the UTM coordinate system for use in GAEP. The UTM system breaks the globe into "zones," each of which has a central meridian. Within each zone, a set of coordinates is assigned to each point, measured (in meters) relative to the central meridian (UTM X value at the meridian is 500000) and to the equator (UTM Y value at the equator is 0). Over a relatively small geographic area, the coordinates can be considered to be Cartesian. This provides a simple X-Y coordinate system in data units for modeling work.

Conversion of Latitude-Longitude to UTM Coordinates

For users' convenience, a facility is included in GAEP for converting between UTM coordinates and latitude-longitude coordinates, given the number of the UTM zone. This feature simplifies the process of locating digitizer origin points.

Users who are unfamiliar with the UTM coordinate system may wish to investigate this topic further. A good reference into geographic coordinate systems should be available in your local library.

Coordinate Origins

The GAEP user is required to set two different types of coordinate origins when managing a modeling project with GAEP. The first, the "Model Origin," is consistent throughout all GAEP and CZAEM operations and should be set at the beginning of the project. The second, the "Digitizer Origin," is set whenever a map is mounted on the digitizing tablet. It is important that the modeler not confuse the meanings of these two terms.

As discussed above, CZAEM works in a dimensionless coordinate space. GAEP, however, assumes that its data are measured in world coordinates which may have a base length unit of meters or feet. A conversion may be performed by GAEP to change the X-Y coordinates from meters to feet when creating an analytic element data file if the modeler desires.

Digitizer Origin

Whenever a map is mounted on the digitizer, the GAEP user must tell GAEP how to convert digitizer coordinates (typically measured in inches or millimeters internally) to "real-world" coordinates. This task is performed by the use of "Digitizer Origins", which are points marked on the map for which the world coordinates are known. GAEP requires that the user locate these points with the digitizer and then enter the world coordinates from the keyboard (Origin command in the Digitize menu).

A pair of digitizer origin points are required each time the map is mounted on the digitizer and must fit on the digitizer surface (of course). This means that for a small digitizer, several sets of digitizer origins may be required on each topographic map. A convenient way to enter these is by the use of the UTM conversion utility (see above), converting the latitude-longitude points on the edges of the map, and writing the corresponding UTM coordinates in the map margin.

Model Origin

The numeric values of world coordinates are often so large (particularly in the Y direction for UTM coordinates) that numerical errors can occur in CZAEM if the geographic coordinates are simply used directly from GAEP. To prevent this, GAEP requires that the modeler enter a "Model Origin" in world coordinates that will be the "zero point" for CZAEM's computations. The model origin should be maintained throughout a particular modeling project, and is included in the CZAEM input data files prepared by GAEP. To select a Model Origin, simply choose a point near the model study region and record its world coordinates. It is particularly convenient to mark and label this point on your maps as well. When prompted by GAEP for a Model Origin, enter the appropriate world coordinates. GAEP will convert the X-Y coordinates of all element features to either feet or meters (depending on the "metric output files" option setting) from the model origin.

Digitizer Origin

Whenever a map is mounted on the digitizer, the GAEP user must tell GAEP how to convert digitizer coordinates (typically measured in inches or millimeters internally) to "real-world" coordinates. This task is performed by the use of "Digitizer Origins", which are points marked on the map for which the world coordinates are known. GAEP requires that the user locate these points with the digitizer and then enter the world coordinates from the keyboard (Origin command in the Digitize menu).

A pair of digitizer origin points are required each time the map is mounted on the digitizer and must fit on the digitizer surface (of course). This means that for a small digitizer, several sets of digitizer origins may be required on each topographic map. A convenient way to enter these is by the use of the UTM conversion utility (see above), converting the latitude-longitude points on the edges of the map, and writing the corresponding UTM coordinates in the map margin.

Construction of No-Flow Boundaries

Implementation of aquifer heterogeneity is considered to be beyond the scope of CZAEM. GAEP, however, allows the modeler to use the "method of images" to create a single, linear no-flow boundary in a model. This feature provides a simple, foolproof technique for generating a "worst case" simulation of this specific case of aquifer heterogeneity. The intent of this feature is to allow the modeler to determine whether additional modeling with a model code which supports heterogeneous aquifers is necessary. If the image result is substantially different from the "no image" result, additional modeling work with a more powerful program is warranted. For a mathematical discussion of the Method of Images, see Groundwater Mechanics (Strack , 1989) pages 28-29.

The image line can be used to simulate the aquifer interface between a highly permeable alluvial channel or glacial outwash zone and a much less permeable aquifer outside the channel or outwash. The user is cautioned that this feature allows only an analysis of two extremes; one with no aquifer heterogeneity, and another with a no-flow boundary. The analysis of image-based, no-flow boundaries should be used only to evaluate the potential effect of a nearby aquifer inhomogeneity in order to determine whether further modeling with a more sophisticated modeling program is warranted.

The steps in creating an image region were illustrate earlier in the W*h*AEM tutorial. The user first selects the "image origin" --- the center of the "image axis." This center also is the center of the rain circle for imaged solutions. The "left-hand rule" is used to determine the region in which elements are to be imaged, or mirrored across the image line. This mirroring of elements across the image line generates a mathematical no-flow boundary along the image line.

Once the user has defined the image line, GAEP prompts for a point on the perimeter of the rain circle, the recharge rate in the rain circle and the head at the reference point. GAEP will automatically locate the reference point far from the study region along the image axis.
This method for creating no-flow boundaries requires that a regional solution with recharge be used. Use of uniform flow in an imaged solution is not allowed.

GAEP is designed to be very easy to understand and use. Commands are set out in a logical set of menus, accessible either from the keyboard or with the mouse. To simplify the discussion of the various GAEP commands, this manual will first describe the use of the menu system, and the use of certain "special keys," which are used consistently throughout GAEP.

Menus

The basic GAEP screen is shown in **Figure 23**. The screen menu shows the current settings of GAEP options and a list of available commands are arrayed across the top of the screen. GAEP commands are accessed by either pressing the "hot" key for the command (shown on the menu in upper case - in red on color monitors) or by placing the mouse cursor on the desired command and pressing the left mouse button. For example, the F (file), A (aquifer), D (digitize), E (element), U (utilities), O (options) and Q (quit) "hot" keys are available either by pressing the appropriate letter on the keyboard or by using the mouse.

File Aquifer Digitize Element Options Utility Quit

GAEP Release 1.0
25 January 1994
Indiana University
SPEA Groundwater Modeling Laboratory

WhAEM Version

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Metric Output Files:	NO
Use digitizer:	YES
Video graphics mode:	COLOR

<f1> help <Esc> return to previous menu 25/Jan/94 12:45 PM

Figure 23 GAEP Main Menu.

Special Keys

In addition to the "hot" keys available from the GAEP menu system, some additional special keys are used:

F1 Key - Help!

Whenever you are at a GAEP menu, press the F1 key to display a help screen which describes the commands available from that menu. This feature is available from menus only, not while entering data or while digitizing.

Esc Key - Go Back!

At all times while using GAEP, the Esc key (upper left on most keyboards) aborts the current command and returns to the previous menu. This key is active at all times while using GAEP to abort the current command. The Esc key is commonly known as the "escape" key.

F3 Key - I'm Done!

While performing certain functions, the F3 key is used to tell GAEP that an operation is complete (examples are completing entry of a stream with the digitizer or creating line-sink elements). This key is used consistently as the "Successful Completion" command while performing data entry.

Page Up / Page Down / Home - Zoom In / Zoom Out

Whenever graphics screen is active, the Page Up and Page Down keys zoom in or out on the area around the center of the screen. To zoom in, press Page Down. To zoom out, press Page Up. The Home key zooms out to a window which shows the entire extent of the digital map data presently loaded. GAEP remembers where you have set the view window for future commands.

Arrow Keys - Scrolling

Whenever graphics screen is active, the left, right, up and down arrow keys scroll the graphics image. GAEP remembers where you have set the view window for future commands.

Data Entry Considerations - Free-Form Input

Whenever more than one value is requested by GAEP, the user may use either spaces, commas or other common punctuation characters (except decimal points and minus signs, of course). All punctuation marks are ignored. For example, if the latitude of a point 37 degrees, 30 minutes, 15 seconds is requested, the user may enter :

37 30 15 or 37,30,15

Procedure for Using GAEP

The user should apply the following procedure when using GAEP on a modeling project:

Create the Digital Map (Digitize menu)

This step involves the entry of hydrologic features using the digitizer, including the points of known head, and the maintenance of the digital map using GAEP's editing features.

Save the Digital Map (File menu)

The digital map is saved to a Digital Map File (see above). It is STRONGLY recommended that the user perform save operations regularly during map creation and editing to prevent loss of data. Always back up your digital map files to floppy disks for safekeeping!

Create Analytic Elements (Element menu)

Once a digital map is complete, the modeler uses GAEP to create analytic elements (line-sinks and wells) and to set the various aquifer properties.

Save the Analytic Element Data File (File menu)

After element creation is finished, the user saves the analytic element file for use in CZAEM. GAEP also allows the modeler to re-load the analytic element file into GAEP for editing and modifications.

Detailed GAEP Command Descriptions

The remainder of this document describes the commands available from each menu in the GAEP program in detail, organized by menu. Each GAEP command menu is described separately.

File [F] - File operations
Selects the "File" menu (see **Figure 24**).

Aquifer [A] - Set aquifer properties
Selects the "Aquifer" menu (see **Figure 25**).

Digitize [D] - Digital map creation/editing
Selects the "Digitize" menu (see **Figure 26**).

Element [E] - Analytic element creation/editing
Selects the "Element" menu (see **Figure 28**).

Options [O] - Set GAEP options
Selects the "Options" menu (see **Figure 29**).

Utility [U] - Run GAEP utilities
Selects the "Utilities" menu (see **Figure 30**).

Quit [Q] - Exit program
Exits GAEP. If changes to the digital map file or analytic element data file have been made but not saved to disk, GAEP will ask if the user really wishes to leave the program. Answer "YES" to exit GAEP without saving to disk.

File Menu

This menu provides access to the various file management facilities in GAEP.

New [N] - Clear program memory
Clears all GAEP's memory, both the digital map and any elements which have been created. This is functionally equivalent to leaving GAEP and re-entering the program.

ReadDM [R] - Read a digital map
Reads a digital map file from the current directory (defaults to x:\WHAEM\DAT after installation, where x: is the drive where you installed W*h*AEM). GAEP will prompt for the name of the file to be read from disk. Type the name of the file, followed by the <enter> key, or press <esc> to abort the command. If a digital map is already loaded, this command will add the newly read map to the already loaded map. To remove the previously loaded map, either re-start GAEP or use the New command (see above). Note: A digital map file MUST be loaded before certain functions can be performed. It is impossible to create or to view analytic elements unless a digital map is loaded.

New ReadDM WriteDM LoadElem SaveElem Map ChangeDir setDrive Quit

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode: COLOR

<f1> help <esc> return to previous menu 25/Jan/94 12:56 PM

Figure 24 GAEP File Menu.

WriteDM [W] - Write a digital map
Writes the current digital map to a digital map file in the current directory (defaults to x : \WHAEM\DAT after installation, where x: is the drive where you installed W*h*AEM). GAEP will prompt for the name of the file to be written. The current version of GAEP does not enforce file extensions, so any filename and extension are allowed, but it is useful to choose a consistent naming convention. The "unofficial" standard for digital map file names is to use the extension .DM. Type the name of the file, followed by the <enter> key, or press <esc> to abort the command.

LoadElem [L] - Read an analytic element data file
Reads an analytic element data file from the current directory (defaults to x:\WHAEM\DAT after installation, where x: is the drive where you installed W*h*AEM). GAEP will prompt for the name of the file to be read from disk. Type the name of the file, followed by the <enter> key, or press <esc> to abort the command.

SaveElem [S] - Write an analytic element data file
Writes the currently defined analytic elements and aquifer properties to an analytic element data file in the current directory (defaults to x:\WHAEM\DAT after installation, where x: is the drive where you installed W*h*AEM). GAEP will prompt you for the Model Origin for the project (see above discussion of the Model Origin). GAEP will then prompt for the name of the file to be written. The current version of GAEP does not enforce file extensions, so any DOS filename and extension is allowed, but it is useful to choose a consistent naming convention. The "unofficial" standard for analytic element data file names is to use the extension .DAT. Type the name of the file, followed by the <enter> key, or press <esc> to abort the command.

Note: If you have defined any GAEP digital map features for "background" maps in CZAEM (that is, curves or point sets), GAEP will automatically put the background map into your CZAEM-compatible analytic element data file. You will be asked if you wish to leave out the map features (this is discouraged). If so, answer "YES" to the question, "Exclude background map?"

Map [M] - Create a background map file
Writes any currently defined "background map features" (that is, curves and point sets) to a CZAEM-compatible data file in the current directory (defaults to x:\WHAEM\DAT after installation, where x: is the drive where you installed W*h*AEM). GAEP will prompt you for the Model Origin for the project (see above discussion of the Model Origin). GAEP will then prompt for the name of the file to be written. The current version of GAEP does not enforce file extensions, so any DOS filename and extension is allowed, but it is useful to choose a consistent naming convention. The "unofficial" standard for analytic element data file names is to use the extension .MAP.

ChangeDir [C] - Change the current directory
Changes the default working directory for GAEP. GAEP will prompt for the new working directory, which will be the "permanent" directory for future GAEP sessions until changed again. The default working directory is x:\WHAEM\DAT where x: is the drive where you installed W*h*AEM, after installation. Since this command does not change the INITAEM.DAT file (see CZAEM documentation), use of this command is discouraged. If you do wish to change the default directory, please remember to edit INITAEM.DAT so that the DATA directory prefix matches the new directory.

Quit [Q] - Return to the main menu

Aquifer Menu

This menu allows the specification of aquifer properties.

Base [B] - Set the aquifer base.
Sets the elevation of the base of the aquifer. GAEP will request the elevation, in units consistent with the desired project units. Enter the value and press <Enter>. Press <Esc> to abort.

Base Thick Permeability pOrosity UniformFlow reFerence Rain Quit

Aquifer Parameter Settings:

Base Elevation:	0.0
Thickness:	100.0
Permeability:	100
Porosity:	0.2
Uniform flow: Q0	0
alpha:	0.00
Reference point: X	0.0
Y	0.0
Head	0.0

Rain element is not defined

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 297632

Option Settings:

Unit Conversion:	FT->M
Digitizer Mode:	DIGITIZER
Video graphics mode:	COLOR

<f1> help <esc> return to previous menu 25/Jan/94 06:39 PM

Figure 25 GAEP Aquifer Menu.

Thick [T] - Set the aquifer thickness.
Sets the thickness of the aquifer. GAEP will request the thickness, in units consistent with the desired project units. Enter the value and press <Enter>. Press <Esc> to abort.

Permeability [P] - Set the aquifer hydraulic conductivity.
Sets the permeability (hydraulic conductivity) of the aquifer. GAEP will request the permeability, in units consistent with the desired project units (feet per day or meters per day are commonly used). Enter the value and press <Enter>. Press <Esc> to abort.

Porosity [O] - Set the aquifer porosity.
Sets the porosity of the aquifer. GAEP will request the porosity, as a fraction between 0.0 and 1.0. Enter the value and press <Enter>. Press <Esc> to abort.

UniformFlow [U] - Define a uniform flow field.
Allows the user to set up uniform flow in the model domain (NOT ALLOWED IF IMAGING IS IN USE). GAEP requests the aquifer discharge magnitude in units consistent with the desired

project units. Enter the value and press <Enter>. Next the orientation of the discharge vector (in degrees) is requested. Enter the value and press <Enter>. Press <Esc> to abort.

NOTE: When uniform flow is used, the uniform flow discharge rate is <u>not</u> recomputed when the user changes the aquifer geometry or permeability. It is up to the user to recompute the uniform flow discharge when aquifer properties or geometries are adjusted.

<u>Reference</u> [F] - Set the reference point.
Allows the user to define a reference point for the model (see the modeling discussion in the main W*h*AEM manual). GAEP will request the location of the reference point in UTM coordinates. Enter the location and press <Enter>. GAEP will then request the reference head. Enter the value and press <Enter>. Press <Esc> to abort at any point.

NOTE: When using imaging, the location is determined by GAEP; only the head will be requested.

<u>Rain Circle</u> [R] - Define a rain recharge circle.
Allows the user to define the rain (areal recharge) element (see the modeling discussion in the W*h*AEM tutorial). GAEP will display the digital map and request the center of the rain circle. Select the center for the rain circle and press the left mouse button. GAEP then requests that a point on the perimeter of the circle be selected. Select the point and press the left mouse button. GAEP then requests that a recharge rate be entered, in the same units as the permeability. Enter the value and press <Enter>. Press <Esc> at any time to abort.

NOTE: When using imaging, the center location is determined by GAEP; only the edge and recharge rate will be requested.

<u>Quit</u> [Q] - Return to the main menu

Digitize Menu

This menu provides all the digital map feature creation and editing facilities. Whenever the user digitizing a digital map feature, the current UTM coordinates of the puck are shown in the upper right corner of the screen. During the setting of origins, the digitizer coordinates (in inches) are shown. When a digitizing step is begun, the digitizer cursor (white "plus" sign) will appear and the computer will sound a "beep" tone. Wait for the beep before entering points with the puck. It is up to the user to determine the number of points to digitize along a feature according to the detail desired. It is also up to the user to determine what stream extents to include, and whether ephemeral streams are to be digitized.

<u>Origin</u> [O] - Set the digitzer origins
Sets up the conversion between digitizer coordinates and real-world coordinates on your map. GAEP will request that the user input one of the digitizer origins marked on the map (see above). Place the digitizer puck on the first point and press the first button on the puck. GAEP will prompt

Origin Stream Heads Curve Points View Edit Quit

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode:COLOR

<f1> help <esc> return to previous menu 25/Jan/94 12:58 PM

Figure 26 GAEP Digitize Menu.

for the UTM coordinates of the first point. Next, GAEP will repeat the process for the second origin point. Once the digitizer origin is set, it is possible to digitize hydrologic and background map features. Note: If the digitizer option is set to "mouse markup" mode, you may digitize onscreen at any time after a digital map is loaded. See the "Options" menu discussion of the digizer setting.

Stream [S] - Digitize a stream
Allows the user to digitize the location of a stream. GAEP will prompt for the name of the feature; enter the name and press <Enter>. GAEP will now design an abbreviation for the name and allow you to change it by entering a new abbreviation (up to 9 characters), or just press <enter> to use the one GAEP designs. Now, the graphics screen will show the extent of the digitizing area. Select points on the stream from the map and press the first digitizer button to enter the stream's location. When complete, press <F3>, or press <Esc> to abort digitizing the feature.
While digitizing, GAEP will display the points entered as dark blue "plus" signs. The points are connected into a stream feature after the user presses the F3 key. Once entered, the stream will be displayed as a continuous curve.

Heads [H] - Digitize heads on a stream
Allows the user to digitize the locations of known elevations on a stream. GAEP will show the digital map onscreen. Select the feature to have heads added and press the left mouse button. Now, the graphics screen will show the extent of the digitizer. Select points where elevation contours cross the stream from the map and press the first digitizer button. GAEP will request the elevation. Enter the elevation in units consistent with the desired model units (feet or meters). Repeat the process for all possible elevations. When complete, press <F3>, or press <Esc> to abort digitizing the heads. If an elevation is entered incorrectly, you should press <Esc> and start again.

Curve [C] - Digitize a background map curve
Allows the user to digitize the location of a background map curve. A background map curve is a digital map feature which denotes a road, property boundary, political boundary, or other linear feature. These curves become "background map" features in CZAEM and are used to orient the reader when interpreting CZAEM output. GAEP will prompt for the name of the feature; enter the name and press <enter>. GAEP will now design an abbreviation for the name and allow you to change it by entering a new abbreviation (up to 9 characters), or just press <enter> to use the one GAEP designs. Now, the graphics screen will show the extent of the digitizer. Select points on the curve from the map and press the first digitizer button to enter the curve's location. When complete, press <F3>, or press <Esc> to abort digitizing the feature.

Points [P] - Digitize a background map point set
Allows the user to digitize the location of a background map point set. A background map point set is a digital map feature which denotes a set of wells, houses or other "point" feature. These points become "background map" features in CZAEM and are used to orient the reader when interpreting CZAEM output. They are also used as the base points for the creation of WELL analytic elements (see below). GAEP will prompt for the name of the feature; enter the name and press <enter>. GAEP will now design an abbreviation for the name and allow you to change it by entering a new abbreviation (up to 9 characters), or just press <enter> to use the one GAEP designs. Now, the graphics screen will show the extent of the digitizer. Select points from the map and press the first digitizer button to enter each point's location. When complete, press <F3>, or press <Esc> to abort digitizing the feature.

View [V] - View the digital map
Shows the current digital map on the screen. Streams which have heads associated will be in bright blue (or white on monochrome systems). Streams without heads are dark blue (or dashed). Background map curves and points are dark purple (or dotted).

Edit [E] - Edit the background map
Enters the Digitize/Edit submenu (see **Figure 27**).

Quit [Q] - Return to the main menu

Digitize/Edit Submenu

Join View Delete Rename Quit

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode: COLOR

<f1> help <esc> return to previous menu 25/Jan/94 12:59 PM

Figure 27 GAEP Digitize/Edit Submenu.

Join [J] - Join two map features into a single feature
Allows two segments of a curve or stream to be digitized separately and joined together. GAEP requests that the first segment be selected from the graphics screen. Select the feature with the mouse and press the left mouse button. GAEP then requests the second feature; select with the mouse and press the left button. GAEP will ask "Are you sure?" Press Y to join the features. The joined feature will keep the name of the first feature selected.

View [V] - View the digital map
Shows the current digital map on the screen. Streams which have heads associated will be in bright blue (or white on monochrome systems). Streams without heads are dark blue (or dashed). Background map curves and points are dark purple (or dotted).

Rename [R] - Rename a feature
Allows a feature's name and abbreviation to be changed. GAEP requests that the feature be selected from the graphics screen. Select the feature with the mouse and press the left mouse

button. GAEP then requests a new name; enter the new name or simply hit <enter> to keep the old name. GAEP then requests a new abbreviation; enter the new abbreviation or simply hit <enter> to keep the old abbreviation.

Delete [D] - Delete a feature
Allows a digital map feature to be deleted. GAEP requests that the feature be selected from the graphics screen. Select the feature with the mouse and press the left mouse button. GAEP asks, "Are you sure?" Press Y to delete the feature.

Quit [Q] - Return to the Digitize menu

Element Menu

This menu allows for creation of analytic elements.

Linesink [L] - Create line-sink elements
Allows the user to create line-sinks for a stream. GAEP requests that the stream be selected from the graphics screen. Select the stream with the mouse and press the left mouse button. GAEP will then show a screen which shows only the feature selected. Select the end points of line-sink elements, pressing the left mouse button at each point. Line-sink elements will appear with heads computed by GAEP. Press <F3> to complete the creation of line-sinks or <Esc> to abort. To create additional line-sinks for the s tream, repeat the procedure.

Note: the first line-sink endpoint entered will not appear on the screen until a second point is entered. It does not matter to CZAEM whether line-sinks are entered "heading upstream" or "heading downstream," but the user might wish to work in a consistent manner, creating line-sinks in the same direction for all features.

Well [W] - Create well elements
Allows the user to create wells for a point set. GAEP requests that the "background map point set" (see the "Digitize/Point" command discussion above) be selected from the graphics screen. Select the point set with the mouse and press the left mouse button. GAEP will then show a screen which shows only the feature selected. Select the location of a well and press the left mouse button. GAEP will prompt for the discharge (in units consistent with the ones determined by the user) and radius of the well. Repeat the procedure for each well. Well elements will appear onscreen. Press <F3> to complete the creation of wells or <Esc> to abort.

Delete [D] - Delete elements associated with a digital map feature
Allows elements to be deleted. GAEP requests that the feature be selected from the graphics screen. Select the feature with the mouse and press the left mouse button. GAEP asks, "Are you sure?" Press Y to delete all elements associated with the digital map feature.

Linesink Well Delete Userwindow Image View Quit

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode: COLOR

<f1> help <esc> return to previous menu 25/Jan/94 01:00 PM

Figure 28 GAEP Element Menu.

UserWindow [U] - Set the user's window for CZAEM
Allows the user to tell CZAEM the extent of the CZAEM display window. GAEP will show the digital map and request the lower left corner of the window. Locate the lower left corner with the mouse and press the left mouse button. GAEP will request the upper right corner; select it with the mouse and press the left mouse button. GAEP will then ask if the selection is correct. Press Y to accept the window.

Image [I] - Create a no-flow boundary by imaging
Allows the user to locate a single, linear no-flow boundary. GAEP will display the digital map and request that the image "origin" be selected. Choose the point which is at the center of the desired rain circle (on the image line) and press the left mouse button. GAEP will request that the second image point be entered; select a point which is on the "positive" axis of the image line. That is, the features which will be imaged will be determined by the "left hand rule," based upon the two points on the line. GAEP will ask if the entered line is correct. Press Y to select the image line. GAEP next requests that a point on the perimeter of the rain circle be entered; select and press the left mouse button. GAEP then requests the recharge rate for the rain circle. Enter the rate of

recharge (in the same units as permeability) and press <enter>. Finally, the head at the reference point will be requested. Enter the reference head and press <enter>. To abort the imaging procedure, press <Esc> at any time in the procedure.
Please see the discussion of imaging in the introductory portion of this manual.

View [V] - View the digital and element map
Shows the current digital map on the screen, with all elements. Streams which have heads associated will be in bright blue (or white on monochrome systems). Streams without heads are dark blue (or dashed). Background map curves and points are dark purple (or dot-dashed). Line-sink elements which have been created by the user will appear in green, those which were created by imaging (if any) in white. The image line (if any), rain circle and user window will be shown in green.

Quit [Q] - Return to the main menu

Options Menu

All of the commands listed here set program option settings and perform no direct action on the digital maps or analytic elements in use. Once a setting is modified, it is saved to the disk and remains set until changed, regardless of whether GAEP is re-started.

UnitConv [U] - Set the unit conversion mode
Allows the user to change the way unit conversions are handled when reading or writing analytic element files. Three modes are supported; "feet-to-meters," "none" and "meters-to-feet." If your digital map was digitized in UTM, you would use the "meters-to-feet" setting to make analytic element files in feet or the "none" setting for analytic element files in meters.
When this command is selected, GAEP cycles through the unit conversion options. Continue until the desired mode is shown on the screen.

Digitizer [D] - Select the digitizer mode
The digitizer mode can have any of three states: "Digitizer", "Mouse Markup", and "Direct." In "Digitzer Mode," all coordinate input is performed using the presently configured digitizer. In "Mouse Markup Mode," the mouse may be used to enter points directly on a displayed digital map. It is NOT RECOMMENDED for general purpose data entry. "Direct" mode allows the user to enter the coordinate locations (in world coordinates) from the keyboard, such as for wells and piezometers of known location.

NOTE: Previous versions of GAEP used this command to support the "keyboard digitizer" for users who had no digitizers. This function is now handled by the digitizer driver as the "keyboard" protocol; see the Appendix C Tablet Installation Guide for details.

UnitConv Digitizer VideoMode Quit

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 309920

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode: COLOR

<f1> help <esc> return to previous menu 25/Jan/94 01:01 PM

Figure 29 GAEP Options Menu.

VideoMode [V] - Switch video modes
Switches the video from color to monochrome or back.

Quit [Q] - Return to the main menu
Returns to the main menu. All option settings will be maintained for future GAEP sessions until changed again.

Utility Menu

This menu provides several useful utility functions.

UTM/LatLong [U] - Enter the UTM/Latitude-Longitude conversion utility
Enters the UTM/LatLong submenu (see **Figure 31**).

Dos [D] - Run MS-DOS
Starts the MS-DOS shell specified in the COMSPEC environment variable (see MSDOS manual).

To return to GAEP, type EXIT<enter> at the DOS prompt.

Quit [Q] - Return to the main menu

```
Utm/latlong  Dos  Quit

                                        Current Directory:
                                        C:\WHAEM\DAT
                                        Current Map File:


                                        Current Element File:

                                        Memory available:  309920


                                        Option Settings:

                                        Unit Conversion:      FT->M
                                        Digitizer Mode:       DIGITIZER
                                        Video graphics mode:  COLOR

          <f1> help  <esc> return to previous menu  25/Jan/94 01:02 PM
```

Figure 30 GAEP Utility Menu.

UTM / Latitude-Longitude Utility

This menu allows the user to convert UTM coordinates to latitude-longitude coordinates or latitude-longitude coordinates to UTM coordinates, given a known UTM zone.

Zone [Z] - Enter the UTM zone
Allows the user to define the UTM zone for conversions. GAEP will request that the zone number be entered. Enter the desired zone (look in the lower left corner of the topographic map for the zone number for you map) and press <Enter>. Press <Esc> to abort.

Zone laTitdue loNgitdue Utm Quit

Geographic Location:

UTM Zone: 0

Latitude: 00 d 00 m 00 s
Longitude: 00 d 00 m 00 s

UTM coordinates: X 0.0
Y 0.0

Current Directory:
C:\WHAEM\DAT
Current Map File:

Current Element File:

Memory available: 297632

Option Settings:

Unit Conversion: FT->M
Digitizer Mode: DIGITIZER
Video graphics mode: COLOR

<f1> help <esc> return to previous menu 25/Jan/94 06:56 PM

Figure 31 GAEP Lattidude-Longiture/UTM Utility Menu.

Latitude [T] - Enter the latitude, convert to UTM
GAEP requests the latitude of the point to be converted. Enter the degrees, minutes and seconds of latitude of the point and press <Enter> (do not use any text labels such as "degrees" in the data entry, enter only the numerical values in the specified order). Press <Esc> to abort. GAEP will report the UTM coordinates of the point entered in latitude-longitude coordinates.

Longitude [O] - Enter the longitude, convert to UTM
GAEP requests the longitude of the point to be converted. Enter the degrees, minutes and seconds of longitude of the point and press <Enter> (do not use any text labels such as "degrees" in the data entry, enter only the numerical values in the specified order). Press <Esc> to abort. GAEP will report the UTM coordinates of the point entered in latitude-longitude coordinates.

UTM [U] - Enter the UTM coodinates, convert to latitude-longitude
GAEP requests the UTM coordinates of the point to be converted. Enter the UTM coordinates and press <Enter>. Press <Esc> to abort. GAEP will report the latitude-longitude coordinates of the point, based on the UTM zone specified.

Quit [Q] - Return to the utility menu.

APPENDIX B. CZAEM REFERENCE

There are two levels of help in CZAEM: 1) brief, context-sensitive help available by typing a command followed by a question mark (e.g., "trace ?"); and 2) more extensive help available by typing "help" while in one of the modules. This chapter of the W*h*AEM manual is a printout of these more detailed help files located in the \WHAEM\CZAEM\HELP directory.

CZAEM Main Module

```
\\\  Module=MAIN MENU           Level=0   Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

If you type help at the main menu,

help<Enter>

the following help file will scroll across the screen:

NOTATION: <COMMAND> (PARAMETERS) [OPTIONAL] {explanation}

ANALYTIC ELEMENT MODULES: <AQUIFER><GIVEN><REFERENCE><WELL><LINESINK>

These modules allow for the entry of aquifer parameters and boundary conditions.

SOLVING: <SOLVE><COINCIDE>[TOL]

<SOLVE>

Initiate the process of solving the system of equations. If iteration is necessary, this will be done with a relative accuracy of 0.01; no maximum number of iterations is set.

The program will abort the solving process whenever it detects that control points are too close together. It will tell you which control points on which elements are too close. You can then modify your input accordingly.

<COINCIDE>(TOL)

Display of all control points that are within TOL from one another, if the conditions applied at these points are in conflict. The value of TOL will be used by the solve routine. Thus, you can force the program to solve problems with control points that are very close together (but do not coincide) by setting TOL to zero in this command. Whenever control points have been detected that are too close, they will be marked on the layout. If the optional argument TOL is not entered, then the program will display the current value prior to checking control points.

CHECK MODULE: <CHECK>

This module is used for retrieval of data in numeric form.

TRACE MODULE: <TRACE>

This module is used to generate pathlines. This module also allows access to the capture zone routines.

CONTOURING MODULE: <GRID>

This module is used for grid scalar values for contouring.

STAND-ALONE COMMANDS: <GRID>(NX)

Contour rectangular grid of NX intervals horizontally. The default of the function to be contoured is piezometric head.

<LAYOUT>

Will cause a layout to be displayed.

<PAGE>

Will cause all screens to be cleared. On printers the previous page will be ejected.

<PAUSE>

Suspends execution of the program. Availability of commands depends upon the memory available with the program installed. The extended memory versions do not support a meaningful use of the PAUSE command.

<PLOT>

Initiates contour plotting. The program will display the maximum and minimum levels encountered in the grid to be displayed, and will ask you to supply starting levels and a contour increment.

<TITLE>

After you enter this command, the machine will prompt you for a title of less than 17 characters. If you press ENTER without text, the current title is displayed. The title will be displayed on the plots and on printed output.

<VIEWPORT>(FACTOR)[I1,I2]

This command allows you to reduce the plots in size and to move it to a new position. FACTOR must be <=1. The values of I1 and I2 are also <=1, and represent the distance as a fraction of the display width and height over which the plot is to be moved right and up, respectively.

<WINDOW>[(X1,Y1,X2,Y2)]/<ALL>]

<WINDOW> sets the window for subsequent plotting. X1,Y1 and X2,Y2 represent the coordinates of lower left and upper right corners of the window, respectively. Typing WINDOW ALL will change the window size to be large enough to include all elements in the model. IT IS IMPORTANT THAT YOU ENTER THE WINDOW BEFORE ENTERING AQUIFER DATA, BECAUSE MANY DEFAULT VALUES ARE SET AS A FRACTION OF THE WIDTH OF THE CURRENT WINDOW.

<RESET>

Resets all values in the program to what they are when the program is loaded into memory.

<SWITCH>(FILENAME)

Will cause the program to read further input from the file FILENAME; the last command in the file must be SWITCH CON.

<STOP>

Stops the program.

SERVICE MODULES: <SAVE><READ><CURSOR><SWITCH><PSET><MAP><FIPLOT>

<SAVE>/<READ>

To store or retrieve solutions and grids in binary form; to subtract grids.

<CURSOR>

Data retrieval by means of the cursor. Changing of well and line-sink data.

<SWITCH>

Input and output re-direction.

<PSET>

To direct graphics output, and to set plotting attributes.

<MAP>

To generate a background map and to activate plotting of the background map.

<FIPLOT>

To record plots in binary form.

AQUIFER

```
\\\  Module=MAIN MENU            Level=0    Routine=INPUT               ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>        <MAP>                                        <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                     <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>     <PLOT>                                       <PAUSE>
<SOLVE>        <TRACE>                                      <RESET>
<CHECK>        <CURSOR>                                     <PSET>
                                                            <STOP>
```

aquifer <Enter>

```
\\\  Module=AQUIFER              Level=1    Routine=INPUT               ///
<PERMEABILITY>(PERM)<THICKNESS>(THICK)<BASE>(ELEVATION)<POROSITY>(POROSITY)
<RESET><HELP><RETURN>
```

This module allows input of aquifer parameters.

The command words are:

<PERMEABILITY>(PERMEABILITY)

Specify the hydraulic conductivity in units of L/T.

<THICKNESS>(THICKNESS)

Specify the aquifer thickness. If the aquifer is unconfined, choose a value for THICKNESS large enough to ensure that the aquifer is everywhere unconfined. Do not choose unnecessarily large values to prevent loss of accuracy. Transition from confined to unconfined conditions is automatically taken care of by the program.

<BASE>(ELEVATION)

Specify the base of the aquifer. The program will refer all values of head with respect to this base. For example, if ELEVATION equals 10, a head of 20 will correspond to a head of 10 with respect to the base of the aquifer system. Unnecessarily large values for base may lead to loss of accuracy.

<POROSITY>(POROSITY)

Set the porosity in the aquifer.

<TFAC>(FACTOR)

Set the factor by which time entries must be multiplied in order to obtain units as used in the coefficient of permeability and rainfall, e.g., if permeability is in m/s and times are given in days, then FACTOR=3600*24=86400.

<RESET>

Reset all parameters in the aquifer module to their default values.

<RETURN>

Return control to the main menu.

GIVEN

The word "given" refers to the already known or "given" average extraction or infiltration rate of a hydrologic feature such as wells, line-sinks, or ponds.

```
\\\  Module=MAIN MENU            Level=0    Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

given <Enter>

```
\\\  Module=GIVEN                Level=1    Routine=INPUT              ///
<UNIFLOW>(DISCHARGE)[ANGLE]<RAIN>(X,Y,RADIUS,RATE)<RESET><HELP><RETURN>
```

This module allows input of the following given functions: uniform flow, and constant infiltration over a circular region of the aquifer.

The command words are:

<UNIFLOW>(DISCHARGE RATE)[ANGLE OF FLOW IN DEGREES]

Specify the discharge rate and the angle of flow (optional) in degrees between the direction of flow and the x-axis for uniform flow. The discharge rate equals the amount of flow per unit width, measured over the entire thickness of the aquifer.

<RAIN>(X,Y,RADIUS,INFILTRATION RATE)

Specify the rainfall entering the top of the aquifer. INFILTRATION RATE is positive for water entering the aquifer and is measured in volume per unit area (L/T). Only one region of infiltration due to rainfall may be specified.

<RESET>

Reset all parameters in the module GIVEN to their default values.

<RETURN>

Return control to main menu.

REFERENCE

The reference point is a point in the domain where the head is "known" or assumed. It is usually treated like the average head in the study area placed at some arbitrary point outside the domain of interest. It is entered directly from the main command line.

```
\\\  Module=MAIN MENU           Level=0     Routine=INPUT             ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>        <MAP>                                        <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                     <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>     <PLOT>                                       <PAUSE>
<SOLVE>        <TRACE>                                      <RESET>
<CHECK>        <CURSOR>                                     <PSET>
                                                            <STOP>
```

ref
(X,Y,REFERENCE HEAD)

X,Y,REFERENCE HEAD : 0.00000E+00 0.00000E+00 0.00000E+00

```
\\\  Module=MAIN MENU           Level=0    Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

well <Enter>

```
\\\  Module=WELL                Level=1    Routine=INPUT              ///
<GIVEN><RESET><HELP><RETURN>
```

This module allows input of wells. The program currently supports two types of wells: wells with given discharge and wells with the head specified at the well boundary.

The command words are:

<GIVEN>

Enter wells of given discharge. The user will be prompted for well coordinates and discharge as follows:

(XW,YW,Q)[RADIUS][[LABEL]]<COMMAND>

where XW,YW represent the x and y coordinates of the center of the well, and Q the discharge. RADIUS is the radius of the well and is optional. The radius has a default value of .001 in absolute units. The value of RADIUS is used in the pathline tracing routine and in the contouring routine. An optional label may be entered between brackets. The program will expect continued entry of discharge specified wells until a command word is entered. You can return to the well input menu by entering COMMAND.

<HEAD>

Enter wells of given head. The user will be prompted for well coordinates, radius, and head at the well radius as follows:

(XW,YW,HEAD,RADIUS)[[LABEL]]<COMMAND>

where XW,YW represent the x and y coordinates of the center of the well, HEAD represents the head of the well at the well radius, and RADIUS is the radius of the well. An optional label may be entered between brackets. The program will expect continued entry of head specified wells until a command word is entered. You can return to the well input menu by entering COMMAND.

<FACTOR>(NUMERICAL VALUE)

Specifying a factor by which each discharge value which is as typed in is to be multiplied to convert it to the units used in the program; e.g., to convert from GPM to cubic feet/min.

<RESET>

Cause all parameters in the module WELL to be reset to their default values.

<RETURN>

Return control to main menu.

LINESINK

```
\\\  Module=MAIN MENU            Level=0     Routine=INPUT               ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]    <HELP>
<GIVEN>        <MAP>                                         <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                      <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                      <READ>
<LINESINK>     <PLOT>                                        <PAUSE>
<SOLVE>        <TRACE>                                       <RESET>
<CHECK>        <CURSOR>                                      <PSET>
                                                             <STOP>
```

linesink <Enter>

```
\\\  Module=LINE-SINK            Level=1     Routine=INPUT               ///
<GIVEN><HEAD><STRING>[<ON>/<OFF>]<TOLERANCE>[TOL]<RESET><HELP><RETURN>
```

This module makes it possible to enter line-sinks. The line-sinks may be used to model narrow creeks that may be either above the groundwater table or intersecting it, as well as closed boundaries.

The line-sinks may be chained together. Either the rate of extraction or the head at the center of the linesink may be specified. The rate of extraction is positive for removal of water from the aquifer, and is measured in discharge per unit length of line-sink, i.e., in units of L^2/T. If the line-sinks are used to model creeks in direct contact with the aquifer, i.e., in cases that the heads are specified at the centers of the line-sinks, the user should verify, after solving the problem, that indeed the heads below the linesink are at or above the streambed level. Otherwise, a run should be done with these line-sinks entered as elements of given strength. In general, boundaries of specified head will be modeled more accurately when more elements are used to discretize these boundaries.

The command words are:

<GIVEN> Enter line-sinks of constant rate of extraction. The program prompts for input of coordinates and extraction rate as follows:

(X1,Y1, X2,Y2, EXTRACTION RATE)[[label]]<COMMAND>

The program will expect such lines of input to continue until another command word is entered. An optional label may be entered between brackets. You may return to the linesink menu by entering COMMAND.

<HEAD>

Enter line-sinks of constant head. The program prompts for input of coordinates and head as follows:

(X1,Y1, X2,Y2, HEAD)[[label]]<COMMAND>

The program will expect such lines of input to continue until another command word is entered. An optional label may be entered between brackets. You may return to the linesink menu by entering COMMAND.

<FCHANGE>[EL.NR.]

Change the location and head of the head-specified linesink element number EL.NR.

<STRING>[<ON>/<OFF>]

This command allows the user to specify whether linesink elements will be linked together into strings of elements. When line-sinks are linked into strings, the start nodes of linesink elements on the same string will be moved to the location of the closest end node on the string. Typing any command or specifying a start node which is at least the TOLERANCE distance away from all end nodes on the string will cause a new string of linesink elements to be started.

<TOLERANCE>[TOL]

This command allows the tolerance for connecting linesink elements into strings of elements to be displayed and/or specified. If the start node of the linesink element that is being entered is within the TOLERANCE distance of the end node of any other linesink element on the string, then the element will be linked to the closest end node on the string.

<RESET>

This command causes all parameters in the linesink module to be reset to their default values.

<RETURN>

Returns control to the main menu.

CHECK

```
\\\  Module=MAIN MENU            Level=0    Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

check <Enter>

```
\\\  Module=CHECK                Level=1    Routine=INPUT              ///
<AQUIFER><GIVEN><REFERENCE><WELL><LINESINK>
<HEAD>(X,Y)<DISCHARGE>(X,Y)<CONTROL><SUMMARY><HELP><RETURN>
```

The check routine is intended to provide access to data such as well locations and discharges, aquifer thickness and hydraulic conductivity. It also provides a means to check that the solution meets the conditions specified at the control points. It is important to note that errors caused by loss of significant digits can always lead to a solution that is inaccurate at certain control points. Check gives access to data in each of the modules, all of which have individual check routines and relevant help screens via the <HELP> command.

The following commands give access to modules:

<REFERENCE><AQUIFER><GIVEN><WELL><LINESINK>

Each of the modules has a command <CONTROL> which displays the conditions at the control points along with the computed values. For example, the command <REFERENCE> followed by the command <CONTROL> displays the coordinates at the reference point, the value of the head specified, and the value of the head computed.

The RANGE command, common to nearly all of the check routines, is explained here. This command sets the type and range of elements to which all subsequent data refers.

The range command words are:

<RANGE>(TYPE,NR1)[NR2]

TYPE is a word signifying the type of element to be considered (the types are for WELL: GIVEN, HEAD, or TIME; for LINESINK: GIVEN or HEAD). The value NR1 is the starting number of the element to be considered (this number is determined by the sequence in which the elements of TYPE were entered).

For example, if 10 wells are entered as type=GIVEN, the command:

RANGE GIVEN 4,8

specifies that subsequent data should pertain to the wells entered as 4,5,6,7, and 8. If NR2 is omitted, the default is to set NR2 equal to NR1.

Other command words are:

<CONTROL>

Display the conditions at the control points for each of the modules. In general, the numbers in the last two or three columns should be identical. Differences in these values indicate inaccuracies for a given problem. Module specific information for this command is available by invoking <HELP> in the appropriate module.

<HEAD>(X,Y)[MEASURED HEAD]

Display the head computed at point (X,Y). If an optional value for MEASURED HEAD is entered, the program will also display the measured head and the difference between the measured and computed heads. This command is useful for comparing observed and predicted heads at observation points. An input file with this command, followed the X,Y location and the associated measured head is recommended for model calibration. The input file is subsequently read using the command <INPUT> in the module SWITCH.

<OMEGA>(X,Y)

Display the complex potential, i.e. the value of the potential and the value of the stream function, at point (X,Y).

<DISCHARGE>(X,Y)[DELTA]

Display the components Qx and Qy of the discharge vector (units of L^2/T) for a point (X,Y). If the optional DELTA is entered, numerical differentiations are performed over an interval DELTA in the x and y directions. The numerically computed discharge components are displayed below those obtained analytically.

<TIME>[TIME]

Display the program time in user units and the value of the time factor TFAC. Entering the optional parameter TIME will cause the time to be set to TIME.

<SUMMARY>

Display general information about all of the modules, including the current and maximum numbers of specific elements. Press the ENTER key after the summary of each module has been displayed, in order to display the next summary.

GRID

The grid module can be executed from the main command line.

```
\\\  Module=MAIN MENU          Level=0    Routine=INPUT           ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

grid <Enter>

```
\\\  Module=GRID               Level=1    Routine=INPUT           ///
<TITLE><HEAD><POTENTIAL><PSI><LEAKAGE><SURFACE><BOTTOM><GRID>(NX)
<WINDOW>[X1,Y1,X2,Y2]<HELP><WRITE><SWRITE><RETURN>
```

This module will set the function to be gridded before contouring.

The command words are:

<HEAD>

Set the piezometric head as the function to be used for future GRID commands.

<POTENTIAL>

Set the potential as the function to be used for future GRID commands.

<PSI>

Set the stream function as the function to be used for future GRID commands.

Note that the contours of the stream function may show branch cuts generated by features that remove water from the aquifer.

The routine will issue a warning if the window has not been set.

<SURFACE>

Set the phreatic surface as the function to be used for future GRID commands.

<BOTTOM>

Set the aquifer bottom as the function to be used for future GRID.

<WINDOW>(X1,Y1,X2,Y2)

Set the window boundary, where X1,Y1 are the coordinates of the lower left corner and X2,Y2 those of the upper right corner.

<GRID>(NUMBER OF INCREMENTS)

Set the number of increments in the X direction of the grid to be contoured.

<TITLE>

Use this command to specify a title of your problem. The program will prompt you to enter a title of less than 17 characters, or to press ENTER to display the current title.

<WRITE>

Use this command to have the program write the grid information to a file in the form: x,y,item, where item is whatever has been gridded the last time. This file can then be used later in a contouring

<RETURN>

package, or other program for display. The program will prompt for the name of the file to be written to.

Returns control to main menu.

PLOT

The plot module is entered AFTER a grid has been generated in the GRID module. In the plot module contours are generated based on the grid data.

```
\\\  Module=MAIN MENU            Level=0     Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]    <HELP>
<GIVEN>        <MAP>                                         <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                      <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                      <READ>
<LINESINK>     <PLOT>                                        <PAUSE>
<SOLVE>        <TRACE>                                       <RESET>
<CHECK>        <CURSOR>                                      <PSET>
                                                             <STOP>
```

plot <Enter>

TRACE

```
\\\  Module=MAIN MENU            Level=0     Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]    <HELP>
<GIVEN>        <MAP>                                         <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                      <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                      <READ>
<LINESINK>     <PLOT>                                        <PAUSE>
<SOLVE>        <TRACE>                                       <RESET>
<CHECK>        <CURSOR>                                      <PSET>
                                                             <STOP>
```

trace <Enter>

```
\\\  Module=TRACE;                 Level=1     Routine=INPUT              ///
<WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]<TOL>[TOLERANCE]<CURSOR>(<ON>/<OFF>)
<SWITCH><SET><PLOT><LAYOUT><CAPZONE><HELP><RETURN>
```

This module allows tracing of pathlines and it allows access to the capture zone module.

The command words are:

<SWITCH>

Set device for numerical output. If OUTPUT=CON, then numerical data will be printed on graphics screen.

<SET>

Provide access to the routine for setting tracing parameters.

<TIME>[TIME]

Set the time, given in user units, to TIME. Entering the command without argument will cause the current time to be displayed both in user and program units. Note that TIME must be positive.

<LAYOUT>

Layout all elements prior to the appearance of the cursor.

<PLOT>

Layout all elements and draw contour plots prior to the appearance of the cursor.

<CAPZONE>

Allow capture zones to be created.

<CURSOR>(<ON>/<OFF>)

When entering the portion of the program for input of starting points of pathlines with CURSOR ON, coordinates will be entered using the cursor. If the CURSOR is OFF, coordinates must be entered by typing in values. These values must then precede the command word. You would set CURSOR OFF, for example, if it is desired to produce the graphical output on the plotter (specified by entering the commands: PSET and PLOTTER). This facility also makes it possible to start pathlines at points with specified coordinates, either typed in or read from a file, using the SWITCH facility.

<WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]

Sets the window. Typing WINDOW ALL will change the window size to be large enough to include all elements in the model. Specifying WINDOW PUSH will save the window setting to a stack in memory. Specifying the WINDOW POP command will retrieve the window settings that were saved via the PUSH option, in the order in which the window settings were saved. Typing

WINDOW without any arguments will display the current window settings.

<TOLERANCE>[TOL]

Sets the tolerance. Entering the command without argument will cause the current value to be displayed. The default value is automatically scaled to the current window.

<SAVE>

Will save a plan view of pathlines on a file. The program will prompt for a filename. If this command is not given, then upon entering the routine TRACE, any plot of pathlines produced will be saved on a file with the name CZDFLT.PTH. Because the tracing routine uses the same memory locations as those reserved for storing grids, the current grid will be saved upon entering trace under the filename SLDFLT.GRD.

<READ>

Will read a plan view of pathlines and produce the plot when either the command PLOT or the command LAYOUT is entered. The program prompts for a filename (which may be CZDFLT.PTH)

++
CAUTION: This will occur, for example, if the program attempts to read a grid file for a pathline plot.
To help avoid this, it is suggested that the files be given easily distinguishable extensions, such as GRD, and PTH.
++

<PSET>

Provides access to the routine for setting plot options.

<SWITCH>

Provides access to the routine for re-directing I/O.

<RETURN>

Return to the main menu.

CURSOR ACTIVITY

Entering one of the commands PLOT, LAYOUT, or START, will cause the cursor to appear on the graphics screen. Once the cursor appears, move it to the desired position and enter any one of the commands listed below. Appearance of the cursor may be suppressed, while retaining the graphics capabilities of the TRACE module. This is done by entering the command CURSOR OFF from the main TRACE INPUT menu. In this case, enter manually a pair of coordinates before giving the appropriate command.

<BASE>

Print the base elevation.

<SURFACE>

Print the aquifer upper boundary elevation. For unconfined flow, this boundary is the phreatic surface elevation; for confined flow it is the elevation of the confining layer.

<POTENTIAL>

Print the values of both the potential and the stream function.

<COORDINATES>

Print coordinates of cursor location.

<WLL>

Record set of coordinates for lower left corner of the new WINDOW. Move the cursor to the upper right corner of the window and press enter. The screen will be cleared and the new WINDOW will be activated.

<TRACE>[ELEV]

Start tracing of pathline, beginning at cursor location at elevation ELEV. If ELEV is omitted, the pathline starts at either aquifer top or phreatic surface.

<TOLERANCE>

Set the tolerance. Entering this command followed by moving the cursor will cause a temporary line to be drawn from the position of the cursor at the time the command was given to the current cursor position. Pressing the enter key will cause the tolerance to be stored as the length of the displayed line. This value will be displayed on the screen. If you press ENTER without moving the cursor, the current tolerance will be displayed. Note that the value set for the tolerance using the cursor is temporary; it is valid only while in cursor mode. To re-set the tolerance permanently, enter the value after issuing the command MENU.

<WGENERATE>(# LINES)[ELEV]

Generate pathlines starting at the radius of the well at which the cursor is located. The pathlines will be generated by backwards tracing at equally spaced intervals around the well. The elevation at which the pathlines are started may be specified, otherwise the pathlines will be started at the bottom of the aquifer.

<BACKWARD>(<ON>/<OFF>)

Enables or disables backward tracing. Once set, backward tracing will remain into effect until disabled by this command. Note that the clock will run backward while backward tracing occurs.

<COMMAND>

Causes the command line to be displayed on the screen.

<MENU>

Terminate cursor activity while remaining in the module.

<RETURN>

Returns control to main menu.

CAPZONE in <TRACE>

```
\\\  Module=CAPTURE ZONE;                    Level=1    Routine=INPUT      ///
<COORDINATE><BASE><SURFACE><WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]<WLL>
<SUBZONE><TIMEZONE><SOURCE><NLINE>(LINES)<PAGE><HELP><COMMAND><RETURN>
<FRONT>[<ON>[VELOCITY FACTOR]/<OFF>]<WGENERATE>(# LINES)<COLOR>[COLOR1][2][3]
<BSAVE><BREAD>{TO BACKSPACE, PRESS < }
```

This module allows the creation of capture zones. There are two different types of capture zones which may be created: subzones and time zones.

Subzones are capture zones which delineate the source of all water which enters a well. The capture zone envelope is drawn for this diagram to show what water will enter the well. Dividing streamlines are also drawn from the well to the capture zone envelope to show how much of the water comes from different sources.

Time zones are capture zones which delineate how long it takes the water to reach the well. Contours are drawn which indicate how long it will take water from different locations in the aquifer to flow to the well. The capture zone envelope is drawn for this diagram and it is also equal to the time zone for very large times. The dividing streamlines are not drawn in this diagram; however, if it is required, then both time zones and subzones may be drawn for the same well.

The capture zones are created by moving the cursor close to any discharge well and then typing in either SUBZONE or TIMEZONE.

The capture zone routines use a set of internal buffers to create the various capture zone diagrams. These buffers are set large enough to handle all practical cases. If the buffers are filled during the creation of capture zones, messages will be displayed to suggest how the capture zones may be created. The following variables may be changed in order to create the capture zones properly:

- the minimum step, maximum step, and neighborhood in the tracing module,
- the window size,
- the number of time zones being created,
- the initial number of pathlines used to define the capture zones (NLINE).

The command words are:

<SUBZONE>

Create subzones for the well at which the cursor resides.

<TIMEZONE>

Create time zones for the well at which the cursor resides. The user will be queried for the following information:

```
ENTER <MINIMUM TIME>[TIME STEP][MAXIMUM TIME], OR
   <R>EDRAW LAST TIME ZONES, OR <D>EFAULT TIME ZONES, OR<E>XIT
   MINIMUM AND MAXIMUM TIMES FOR CAPTURE ZONE: (min time) (max time)
```

Specify the times for which time zones are to be created. Entering 'R' will redraw the last time zones that were created. The minimum and maximum time it takes to reach the boundary of the time zone is also displayed as an indication of what times to enter.

<SOURCE>

Display the percentage of water going to the well from all the different subzone sources for the last well at which subzones were calculated. The subzones are displayed in the counter-clockwise direction, starting with the first subzone which begins at an angle greater than zero degrees.

<NLINE>(LINES)

Specify the initial number of pathlines used to define the capture zones. There must be enough pathlines specified so that water from two different sources will not both flow between any two adjacent pathlines. The default number of lines is adequate for most cases.

<COORDINATES>

Print coordinates of cursor location.

<BASE>

Print the elevation of the base of the aquifer at the cursor location.

<SURFACE>

Print the surface elevation at the cursor location.

<WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]

Change the window size. When the window size is changed, the capture zones which were in the previous window are redisplayed. Typing WINDOW ALL will change the window size to be large enough to include all elements in the model. Specifying WINDOW PUSH will save the window setting to a stack in memory. Specifying the WINDOW POP command will retrieve the window settings that were saved via the PUSH option, in the order in which the window settings were saved. Typing WINDOW without any arguments will display the current window settings.

<WLL>

Change the window size by specifying the new window coordinates via the cursor. This command records the set of coordinates for lower left corner of the new WINDOW. Move the cursor to the upper right corner of the new window and press enter. Note that this command is not allowed when CURSOR is OFF.

<SWITCH>

Allows access to the switch routines.

<PAGE>

Clear the screen and layout the elements. This command erases the BSAVE or BREAD file which was currently being used.

<COMMAND>

Causes the command line to be redisplayed.

<HELP>

Causes this help file to be displayed.

<RETURN>

Returns control to the tracing routines.

<FRONT>[<ON>[VELOCITY FACTOR]/<OFF>]

Specify whether to create timezones for the mean velocity of the contaminant, or for the front of the contaminant. Specifying <OFF> will create timezones for the mean travel time of the contaminant. Specifying <ON> along with the VELOCITY FACTOR will create timezones for the front of the contaminant using the specified velocity factor. The velocity factor specifies how much faster the front of the contaminant is traveling than the mean travel time; e.g. specifying a VELOCITY FACTOR of 2.0 will mean that the front of the contaminant travels twice as fast as the average ground-water flow. Note that pathlines in the tracing module are created using these velocities also.

<WGENERATE>(# LINE)

Generate pathlines which go backwards in time and start at the radius of the well at which the cursor is located. The pathlines will be generated by tracing backwards at equally spaced intervals around the well at the bottom of the aquifer.

<COLOR>[COLOR1][2][3]

Specify the color numbers for the different line types:

- COLOR1 is the color of the subzone dividing streamlines
- COLOR2 is the color of the timezones
- COLOR3 is the color of the subzone outer envelopes

<BSAVE>(FILE)

Specify a file to which all subsequent capture zone boundaries will be saved. This file will be closed when any of the following occurs: a new file is specified to save capture zones, a capture zone file is read in via the BREAD command, the PAGE command is specified, or the RETURN command is specified. The default file to which capture zones are saved is CZDFLT.CZ.

<BREAD>(FILE)

Read and plot the capture zone boundaries which were previously saved to a file via the BSAVE command.

<CSAVE>(FILE)

Specify a file to which capture zone buffers will be saved. This command will save all internal buffers which are used by this capture zone program to create capture zones. The buffers will only be saved for the last well for which capture zones were created, prior to issuing the CSAVE command. This command is to be used in conjunction with the CREAD command to redraw capture zones without having to recalculate the capture zone buffers.

<CREAD>(FILE)

Read the capture zone buffers for a well that was previously saved via the CSAVE command. After reading these buffers, capture zones may be redrawn for the well.

Note: If any changes to the model of the aquifer was made between when the capture zone buffers were saved via CSAVE and when they were read via CREAD, then the capture zones will not be an accurate representation of the true capture zones for the aquifer. The subzones and time zones should be recalculated whenever changes are made to the model of the aquifer. Likewise, the buffers were created for a particular window, and hence the window setting which is used during the CREAD command must be the same window setting which was used during the CSAVE command.

Note: The capture zone buffers which are read via the CREAD command were created using either the mean velocity of ground-water or using the front velocity of the contaminant. Therefore, when buffers are read via the CREAD command, the variables specified by the FRONT command will also be changed to correspond to the values which were used when the capture zone buffers were created.

NOTE1:
The time zone and subzone capture zone routines all use the same buffers, and these buffers are filled for the well whose capture zone is currently being drawn. Therefore, it is faster to create all subzone and time zone contours for the given well before creating capture zones for the next well.

NOTE2:
If the buffers are getting full while creating time zones, then try creating just one time zone instead of multiple time zones at once. Changing the window size may also result in being able to create time zones more efficiently. Typing in the PAGE command or creating a capture zone at a new well will clear the internal buffers so that new time zones may be created at the well.

NOTE3:
When a capture zone file is read via the BREAD command, the file remains open and any further capture zones which are created will also be saved in the same file.

NOTE4:
Calling the PAGE command will erase the capture zone file that is currently being filled.

NOTE5:
The window size is not saved in the capture zone file. When the file is read it uses the current window size.

NOTE6:
The layout of the elements is not saved in the capture zone files.

NOTE7:
The capture zones will not be created correctly if there is a stagnation point too close to the well. If the capture zone doesn't look correct, then the WGENERATE command may be used to get an approximate idea of where the stagnation points should be for the well. If the stagnation point is too close to the well, then either the window size may be changed or the tracing maximum step size may be changed to correctly create the subzones.

CURSOR

```
\\\  Module=MAIN MENU            Level=0    Routine=INPUT              ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]   <HELP>
<GIVEN>       <MAP>                                        <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                     <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                     <READ>
<LINESINK>    <PLOT>                                       <PAUSE>
<SOLVE>       <TRACE>                                      <RESET>
<CHECK>       <CURSOR>                                     <PSET>
                                                           <STOP>
```

cur <Enter>

```
\\\  Module=CURSOR               Level=1    Routine=INPUT              ///
<TOLERANCE>(TOL)<SWITCH><PLOT><LAYOUT><HELP><RETURN>
```

This module allows access to cursor activity.

The command words are:

<SWITCH>

Allow setting device for numerical OUTPUT. If OUTPUT=CON, then numerical data will be printed on graphics screen

<LAYOUT>

Display the layout prior to appearance of the cursor.

<PLOT>

Display both the layout and contour plots prior to the appearance of the cursor.

<PAGE>

Clear both the text and graphics screens.

<START>

Will cause the cursor to be displayed.

<TOL>(TOL)

Set the tolerance for subsequent MOVE commands (see below) to TOL.

<RETURN>

Return to the main menu.

Once in cursor mode, enter any of the following commands.

<HEAD>

Print the head.

<POTENTIAL>

Print both the potential and stream function. (Stream function has meaning only if the infiltration rate at position of cursor is zero).

<DISCHARGE>

Print the two components of the discharge.

<COORDINATES>

Print coordinates of the cursor location. A line is drawn from the previous point to the current point. You may prevent this line from being drawn by pressing enter twice. You should do this prior to entering any other command.

<WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]

Change the window size. When the window size is changed, the capture zones which were in the previous window are redisplayed.

Typing WINDOW ALL will change the window size to be large enough to include all elements in the model. Specifying WINDOW PUSH will save the window setting to a stack in memory. Specifying the WINDOW POP command will retrieve the window settings that were saved via the

PUSH option, in the order in which the window settings were saved. Typing WINDOW without any arguments will display the current window settings.

<WLL>

Record set of coordinates for lower left corner of the new window. Move the cursor to the upper right corner of the window and press enter. You may abandon by entering any key other than enter; the cursor will then reappear ready for the next command. The screen will be cleared and the new window will be activated.

<MENU>

Exit from graphics mode; return to main cursor menu.

<TOLERANCE>

Set the tolerance for MOVE commands; e.g., commands that will allow the user to modify endpoints of line-sinks. Entering this command followed by moving the cursor will cause a temporary line to be drawn from the position of the cursor at the time the command was given to the current cursor position. Pressing the enter key will cause the tolerance to be stored as the length of the displayed line. This value will be displayed on the screen. If you press ENTER without moving the cursor, the current tolerance will be displayed.

<SIZE>

Set the maximum size for the display of discharge vectors to the distance between the previous cursor location and the current one.

<NDISCHARGE>

Make it possible to plot the discharge normal to any given line. The discharge is computed at the midpoint of the line between the current cursor location and the next one, defined by entering a carriage return. A rubberband line is displayed if the command is entered properly. Discharge in this context is the component of the discharge vector normal to the line. Points and discharges up to a maximum of 50 are recorded. Plotting is deferred until the command ENDISCHARGE is given. A scale factor is then computed such that the largest vector is equal to SIZE. Both tangential and normal components of the discharge vector are recorded on the OUTPUT device, as well as the coordinates of the points in question and the scale factor. The normal component of discharge is taken positive if pointing to the left with respect to an arrow pointing from the first to the second point entered.

<ENDISCHARGE>

Terminate the entry of points for plotting normal components of discharge.

<LSMOVE>[STRENGTH]

Make it possible to change the locations of endpoints of linesinks, and to change their strength. Move the cursor within a distance of TOL from an endpoint, then move the cursor to the desired location and press ENTER. If you don't move the cursor, the current linesink attributes will be displayed. You may change the strength of the linesink by entering the strength as an optional parameter. If you move the cursor, a line will appear connecting the old and new locations of the endpoint. To facilitate updating of input files, both the old and new locations of the endpoint as well as the linesink number (according to its position in the input file) are printed on the OUTPUT device. It is recommended to echo OUTPUT to a file by the use of the SWITCH command.

<WLMOVE> [DISCHARGE]

Use this command to move wells. It works in the same way as LSMOVE above.

<MARK>

Mark a point on the screen with a marker; this marker is temporary and will not appear on subsequent plots.

<RETURN>

Return to the main menu.

```
\\\  Module=MAIN MENU           Level=0    Routine=INPUT            ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>      <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]  <HELP>
<GIVEN>        <MAP>                                       <SWITCH>[FILE]
<REFERENCE>    <LAYOUT>                                    <SAVE>
<WELL>         <GRID>(NUMBER OF POINTS)                    <READ>
<LINESINK>     <PLOT>                                      <PAUSE>
<SOLVE>        <TRACE>                                     <RESET>
<CHECK>        <CURSOR>                                    <PSET>
                                                           <STOP>
```

pset <Enter>

```
\\\  ROUTINE SET PLOT MODE                                           ///
<PRINTER><SCREEN><DRIVER><PALETTE>(NUMBER)<MOUSE>(<ON>/<OFF>)<HELP><RETURN>
```

This module will set the program for various types of hardware configurations.

<PALETTE>(CODE)

Selects one out of 4 color palettes; CODE may be 1,2,3, or 4.

<PRINTER>

Causes all subsequent plots to be sent to the printer. Note that the resolution on most printers will be much higher for plots generated in this way than by using screen dumps.

<SCREEN>

Causes all subsequent plots to be sent to the screen.

<DRIVER>

Please see read.me file.

<RETURN>

Transfers control back to main program unit.

```
\\\  Module=MAIN MENU             Level=0    Routine=INPUT            ///
ENTER COMMAND WORD FOLLOWED BY ? FOR BRIEF HELP FROM ANY MENU
<AQUIFER>     <WINDOW>[(X1,Y1,X2,Y2)/<ALL>/<PUSH>/<POP>]    <HELP>
<GIVEN>       <MAP>                                         <SWITCH>[FILE]
<REFERENCE>   <LAYOUT>                                      <SAVE>
<WELL>        <GRID>(NUMBER OF POINTS)                      <READ>
<LINESINK>    <PLOT>                                        <PAUSE>
<SOLVE>       <TRACE>                                       <RESET>
<CHECK>       <CURSOR>                                      <PSET>
                                                            <STOP>
```

stop <Enter>

APPENDIX C. TABLET CONFIGURATION GUIDE

This appendix contains a description of the digitizing tablet driver provided as part of GAEP and details about configuration. A description of the program TABTEST, a tool for testing digitizer installation is included. A number of digitizers have been tested with GAEP. The configurations for those tablets are also included here.

Introduction

This document describes the configuration procedure for the program GAEP (as implemented for the W*h*AEM product). It also describes the configuration of the digitizer driver, the devices supported, and gives setup instructions for several digitizers. It is intended that this manual will be expanded as additional digitizers are tested with GAEP.

Installation of GAEP

GAEP installation was performed as part of the install procedure for the W*h*AEM product. A default GAEP configuration based upon no digitizer being connected to the system was installed. If you have not yet installed GAEP, refer to the installation information in the W*h*AEM documentation. This manual presumes that you have already installed GAEP on your system.

Digitizer Configuration

The tablet driver included as part of the W*h*AEM product supports four common digitizing tablet protocols:

- Formatted ASCII protocol (digitizer writes digitizer inches)
- SummaGraphics MM ASCII protocol
- SummaGraphics MM Binary protocol
- SummaGraphics Bit Pad Plus protocol

It is expected that one or more of these protocols will work with nearly any digitizer on the market. Both binary mode and ASCII mode protocols are available. A support program, TABTEST, is provided to assist with digitizer configuration and testing. It is easy to configure GAEP for any of the protocols by modifying the \WHAEM\TABSETUP.BAT file. Protocol selection for GAEP is done by setting the environment variable TABLET and appropriately setting the port parameters in the \WHAEM\TABSETUP.BAT batch file. For example,

```
SET TABLET=ASCII COM1 12 12 4
MODE COM1:9600,E,7,2
```

will select a formatted ASCII mode tablet connected to port COM1 at 9600 bps, even parity, 7 data bits and 2 stop bits. The tablet is 12" x 12" and has a 4-button puck. TABTEST will modify the file TABSETUP.BAT once digitizer configuration is complete. The following several pages document the available protocols.

How Do I Configure My Digitizer for GAEP?

General

Digitizer configuration can be a difficult and frustrating process. GAEP uses a digitizer driver which was written particularly for its use. As part of the digitizer driver, a support program (TABTEST) is provided which may be used to ensure that the digitizer and the software are communicating properly.

This section outlines the basic steps which the user needs to execute to configure the GAEP digitizer driver for a particular digitizer.

Note:

If your system does not have a digitizer, you may wish to configure GAEP to use a Microsoft (or compatible) mouse or to request direct keyboard entry of coordinates. The digitizer driver supports these also; no hardware testing is required. See the appropriate protocol discussion in the "Digitizer Protocols" section of this appendix. TABTEST does not test these protocols.

Step-By-Step

To configure and test your digitizer with the GAEP digitizer driver, the following step- by-step process should be performed. Detailed discussions of the options are to be found in the "Digitizer Protocols," "Program TABTEST" and "Tested Configurations" sections of this appendix.

- If necessary, unpack and install your digitizer and cable it to your computer. Place the digitizer puck on the active digitizing surface.

- Locate your digitizer's reference manual and have it handy before beginning the configuration process.

- Examine your digitizer manual and the "Digitizer Protocols" section of this appendix. Select a protocol to be used. Our experience has shown that the ASCII protocols are easier to test, because the tablet transmits readable characters. In some cases, however, only binary protocols are available. If you have a SummaGraphics Bit Pad Plus, only the Bit Pad Plus protocol may be used.

- Select a transmission baud rate, parity and character format for serial communications. Recommended settings are:
  - ASCII Protocols: 9600 bps, even parity, 7 data bits, 2 stop bits
  - Binary Protocols: 9600 bps, no parity, 8 data bits, 1 stop bit

- Configure your digitizer for the desired protocol. This will require that you follow the instructions in your digitizer manual carefully. This may include setting of hardware switches in your digitizer or running a DOS-based configuration program, or both.

Note: Software-based digitizer configuration

Depending on your digitizer model, you may need to run a program from DOS to set up the digitizer protocol. Be aware that many applications which use your digitizer may transmit configuration information prior to their execution. If a DOS command is needed to configure your digitizer, you will need to manually modify the file TABSETUP.BAT in the W*h*AEM installation directory to execute the proper configuration command, once the digitizer communications have been tested.

Note: Tested Digitizers

Some digitizers have already been fully tested with GAEP. Check the "Tested Configurations" section of this appendix to see if your digitizer has been previously tested.

- Once the digitizer has been configured, run the program TABTEST to test the communications with the digitizer. Set the TABTEST driver, port (COM1 or COM2) and the communications settings (baud rate, etc.), then use the <F2> "CommTest" command (see the "Program TABTEST" section of this appendix for details).

- Once the communications test is successful, use TABTEST's <F3> "DriverTest" command to ensure that the digitizer driver is working and that puck coordinates are being read in inches from the lower left corner of the digitizer (see the "Program TABTEST" section of this appendix for details).

- Once all of the tests are complete, use the <ESC> command to leave TABTEST, and tell TABTEST to write the current settings to TABSETUP.BAT in the \WHAEM installation directory.

- If your digitizer required the execution of a DOS program to set up the digitizer configuration, you will now need to modify the TABSETUP.BAT file in your \WHAEM installation directory to include the command(s) required to configure the digitizer. Place the configuration commands at the beginning of TABSETUP.BAT. The TABSETUP.BAT file is run automatically prior to each execution of GAEP.

The following section of this appendix contains a technical description of the protocols supported by GAEP. Four digitizer protocols, plus the use of a Microsoft (or compatible) mouse or direct keyboard entry are supported.

Formatted ASCII Protocol

This is the favored protocol, when possible. The digitizer transmits the puck coordinates directly as a formatted string, in inches from the lower left corner of the tablet. This protocol is the easiest to debug in most cases. (CalComp digitizers usually refer to this protocol as mode 8.)

To configure the formatted ASCII driver, the TABLET environmental variable should be set as follows:

SET TABLET=ASCII <PORT> <X IN> <Y IN> <# BUTTONS>

where:

- <PORT> Is the serial port used. Only COM1 and COM2 are supported.
- <X IN> Is the size of the digitizer in the X direction, in inches.
- <Y IN> Is the size of the digitizer in the Y direction, in inches.
- <# BUTTONS> Is the number of buttons on the puck.

The configuration used is described in CalComp 2500 Series User's Manual as follows:

- Mode — Run - the tablet transmits continuously
- Commands — Enabled (optional - the driver doesn't use commands)
- Transmit rate — 50 points per second. Can be set as you like; this setting works well with the 2500 and an 80386 or 80486 system.
- Line feed — Disable (required)
- Out of proximity — Enable (optional)
- Margin data — Disable (optional)
- Resolution — 1000 lpi (optional - may be set as desired)
- ASCII Format 8 — Required. This format transmits "XXXX.X YYYY.Y CC T0 <CR>", where XXXX.X is the X position in inches, YYYY.Y is the Y position in inches, CC is a two-character code for the puck button currently pressed, T0 is the digitizer status setting (ignored) and CR is a carriage return.

NOTE:

The number of significant digits transmitted is dependent on the digitizer resolution; check your digitizer manual for details.

•	Baud rate	(optional)
•	Data bits	7 (required)
•	Stop bits	(optional)
•	Parity	(optional)
•	Echo	Disabled
•	Handshake	Enabled
•	Cursor buttons	Set according to your hardware
•	Beeper	Disabled (GAEP beeps when points are entered)

SummaGraphics MM Binary Protocol

To configure the SummaGraphics MM Binary driver, the TABLET environmental variable should be set as follows:

SET TABLET=MMBINARY <PORT> <X IN> <Y IN> <#BUTTONS> <LPI>

where:

•	<PORT>	Is the serial port used. Only COM1 and COM2 are supported.
•	<X IN>	Is the size of the digitizer in the X direction, in inches.
•	<Y IN>	Is the size of the digitizer in the Y direction, in inches.
•	<# BUTTONS>	Is the number of buttons on the puck.
•	<LPI>	Is the number of lines per inch on the digitizer.

The configuration used is described in the SummaGraphics MM1812 Technical Reference as follows (all settings are selected by switches on the tablet, except as noted):

•	Mode	Stream - the tablet transmits continuously. Driver sends the "@" command to select this mode.
•	Transmit rate	110 points per second. This setting works well with the MM1812 and an 80386 or 80486 system.
•	Report format	Binary
•	Resolution	500 lpi
•	Baud rate	9600
•	Data bits	8 (required)
•	Stop bits	1 (required)
•	Parity	Odd

SummaGraphics MM ASCII Protocol

To configure the SummaGraphics MM ASCII driver, the TABLET environmental variable should be set as follows:

SET TABLET=MMASCII <PORT> <X IN> <Y IN> <# BUTTONS> <LPI>

where:

• <PORT>	Is the serial port used. Only COM1 and COM2 are supported.
• <X INCHES>	Is the size of the digitizer in the X direction, in inches.
• <Y INCHES>	Is the size of the digitizer in the Y direction, in inches.
• <# BUTTONS>	Is the number of buttons on the puck.
• <LPI>	Is the number of lines per inch on the digitizer.

The configuration used is described in the SummaGraphics MM1812 Technical Reference as follows (all settings are selected by switches on the tablet, except as noted):

• Mode	Stream - the tablet transmits continuously. Driver sends the "@" command to select this mode.
• Transmit rate and	110 points per second. This setting works well with the MM1812 an 80386 or 80486 system.
• Report format	Binary
• Resolution	500 lpi
• Baud rate	9600
• Data bits	7
• Stop bits	2
• Parity	Odd (by default)

SummaGraphics Bit Pad Plus Protocol

The Bit Pad Plus uses a different protocol than other SummaGraphics digitizers. It is not configurable, so simply setting the TABLET environmental variable is sufficient:

SET TABLET=BITPAD <PORT> <X IN> <Y IN> <# BUTTONS> <LPI>

where:

• <PORT>	Is the serial port used. Only COM1 and COM2 are supported.
• <X IN>	Is the size of the digitizer in the X direction, in inches.
• <Y IN>	Is the size of the digitizer in the Y direction, in inches.
• <# BUTTONS>	Is the number of buttons on the puck

- <LPI> Is the number of lines per inch (200 on the Bit Pad Plus)

The configuration used is described in the SummaGraphics Bit Pad Plus Technical Reference. No tablet settings are required.

- Baud rate 9600
- Data bits 8 (required)
- Stop bits 1 (required)
- Parity Odd (required)

Microsoft Mouse

GAEP supports the use of the mouse for digitizing in two manners. First is the "Mouse Markup" mode, which allows the user to add features to an existing map. This mode is available regardless of the TABLET setting and is selected by commands in GAEP. An alternative use of the mouse is to use the absolute mouse cursor position as a digitizer, so that the position of the mouse on the screen can be scaled as you desire. It is anticipated that this has little use in the context of GAEP and its application is discouraged.

The use of the mouse as a "digitizer" is supported in the digitizer drivers and is documented here only for completeness.

To use the absolute mouse position for digitizing, the TABLET environmental variable should be set as follows:

SET TABLET=MOUSE

- No options are required.

Note:

You will need to set the digitizer origin in GAEP, just as if you had a digitizer (see GAEP manual).

Keyboard Data Entry (For Systems Without Digitizers)

GAEP supports digital map data entry without the use of a digitizer by making the user's keyboard into a "digitizer." The user can use a quadruled sheet (8 squares to the inch vellum works quite well), and trace the features to be digitized onto the sheet, along with geo-referenced origin locations. When GAEP requests data from the "digitizer," the following message appears on-screen :

```
[KEYBOARD DIGITIZER; F1 - BUTTON 1, F2 - BUTTON 2]
```

To "digitize," you simply press the F1 key for "tablet button 1" or the F2 key for "tablet button 2." The digitizer driver will then ask for the coordinates of the point to be entered from your grid sheet. The default is to enter the data in inches, but you may use any grid coordinates you wish.

To use the keyboard data entry method for digitizing, the TABLET environmental variable should be set as follows:

SET TABLET=KEYBOARD <MAXIMUM X> <MAXIMUM Y>

where:

- <x inches> Is the size of the digitizer in the X direction, in inches.
- <y inches> Is the size of the digitizer in the Y direction, in inches.

Note:

If the X and Y are maximum values, the driver defaults to a 20" (X direction) by 24" (Y direction) space, calibrated in inches. You are not restricted to any particular grid coordinate system for data entry; for example, you might have a grid sheet calibrated in grids that was 500 grids on the X axis by 400 on the Y axis. To tell the driver this, you can set the TABLET environmental variable as follows:

SET TABLET=KEYBOARD 500 400

and GAEP will work properly, showing the grid sheet extent while performing data entry.

Note:

You will need to set the digitizer origin in GAEP, just as if you had a digitizer (see GAEP manual).

Program TABTEST

Testing serial communications devices such as digitizers can be time consuming and frustrating due to a lack of standards and because digitizers usually make no directly visible signals. TABTEST is designed to facilitate this process by allowing you to experiment with parameter settings and instantly monitor the effect. The configuration process with TABTEST is subdivided into three steps:

1. Configure the driver for the communications port (COM1 or COM2), baud rate, parity, and number of data and stop bits ("Port" and "Baud, Etc." commands).
2. Establish basic communications with the digitizer ("CommTest" command).

3. Test proper functioning of the digitizer driver ("DriverTest" command).

```
F1-Help      F2-CommTest          F3-DriverTest        F10-Quit
F6-Tablet    F7-Port              F8-Baud,etc.

Tablet:      Formatted ASCII      (+x.xxx,+y.yyy,bb,s)
Port:        COM1
Baud:        9600   Parity: Even  Data bits:7   Stop bits:2




Current setup lines for TABSETUP.BAT:

MODE: COM1:9600,E,7,2
SET TABLET=ASCII COM1 12 12 4

TABTEST v0.2- Graphics Tablet Setup/Test Program
Copyright (c) 1994 V.A. Kelson
```

Figure 32 TABTEST Menu

Commands

Help <F1>

- Displays a help screen

CommTest <F2>

- Tests low-level communications, displaying results one byte at a time.

- For ASCII mode drivers (ASCII and MMASCII) The tablet response as discussed in the driver reference (see above) will be printed on the screen, with continuous update. The user should be able to easily read the puck coordinates and button status. When test is complete, press <ESC> to return to the TABTEST menu.

- For binary mode drivers (MMBINARY and BITPAD) The tablet response will be displayed as five 2-digit hexadecimal numbers, continuously updated. Since it is not easy to read these numbers to ensure they are correct, the user can only look to ensure that they remain constant when the puck is motionless on the tablet and that they change in a regular pattern when the puck is moved. When test is complete, press <ESC> to return to the TABTEST menu.

- During the communications test you may need to experiment with different drivers and the various communications parameters.

Note: Common Blunders

The author has typically made two major blunders at this step in testing, both of which seem trivial. First, make sure that you select the correct communications port (COM1 or COM2), and second, make sure that the digitizer puck is on the active digitizing surface. The driver uses "RUN" mode to read coordinates, and points are only transmitted by most digitizers when the puck is on the digitizer.

DriverTest <F3>

- Tests the tablet driver. Prints puck location in inches from the lower left corner of the tablet.

- The results of this test are the same for all tablet drivers. The current puck coordinates will be printed and continuously updated. The values printed should be the x- and y-coordinates of the puck in inches from the lower left corner of the tablet.

- During the use of the driver test, you may need to experiment with the selected protocol selection and the number of lines per inch (LPI) setting. A common problem is that the test looks fine, but the number of inches is off by some factor. This usually indicates that the LPI setting is wrong.

- Again, be aware of the "Common Blunders" mentioned above.

Tablet <F6>

- Selects the tablet driver. A menu is displayed, showing the digitizer driver choices.

Note:

After the tablet driver choice is made, it is usually necessary to set the port parameters (see below).

Port <F7>

- Chooses the serial port (COM1 or COM2) where the tablet is connected. A menu of choices is printed; choose the appropriate port, depending upon your cabling.

Note:

After the port choice is made, it is usually necessary to set the port parameters (see below).

Baud, etc. <F8>

Sets up the COM port parameters for serial communications with the digitizer. The choices are:

- Baud rate: 300, 600, 1200, 2400, 4800, 9600 bps
- Parity: Odd, Even or None
- Data Bits: 7 or 8
- Stop Bits: 1 or 2

Notes:

- Binary mode tablet drivers (MMBINARY and BITPAD) MUST use 8 data bits. ASCII mode drivers (ASCII and MMASCII) usually use 7 data bits.
- It is usually best to use the default communication parameters for your tablet; check your tablet's reference manual for details.
- Many digitizer models require the execution of a DOS program which sends command to the tablet, initializing the protocol, baud rate, etc., prior to use. It is most critical that the user check the digitizer manual THOROUGHLY and perform any necessary tasks prior to attempting to use TABTEST.

Quit <ESC>

- Exits TABTEST. The user is prompted whether or not to update TABSETUP.BAT. If the user wishes, the updated settings for the TABLET environmental variable and a DOS MODE command will be placed in \WHAEM\TABSETUP.BAT for execution at startup for future GAEP and TABLET runs.

Note:

TABTEST does not retain any of the settings internally after termination. Each TABTEST run starts "from scratch." In most cases, the user will only run TABTEST once when installing W*h*AEM , and the tablet will work thereafter.

Tested Configurations for Various Digitizers

During the development of GAEP, several digitizers have been tested, and their known configurations are shown here. This information is provided as an example of the way one might use the system.

<u>Note:</u>

If you have a digitizer not shown here and you are successful in using the device with one of the supported protocols, it would be much appreciated if you would record the tablet and software configuration information and send it to the author:

Vic Kelson
Indiana University
SPEA Groundwater Modeling Laboratory
PV418
Bloomington IN 47405
Internet: vkelson@ucs.indiana.edu

cc:=> Steve Kraemer
USEPA/RSKERL
P.O. Box 1198
Ada, OK 74820
Internet: kraemer@ad3100.ada.epa.gov

<u>CalComp 2500 (12"x12")</u>
Soft switch settings:

- Bank 1: 0 0 0 0 0 0 0 1
- Bank 2: 1 0 1 1 0 0 0 1
- Bank 3: 0 1 1 0 1 0 0 0
- Bank 4: 0 0 1 0 0 0 0 1
- Bank 5: 0 1 1 1 0 0 1 0

Environmental variable settings:
SET TABLET=ASCII COM1 12 12 4 for COM1: connection
MODE COM1:9600,E,7,2
SET TABLET=ASCII COM2 12 12 4 for COM2: connection
MODE COM2:9600,E,7,2

<u>CalComp 9500 (48"x36")</u>
Soft switch settings:

- Area 1(left to right): 0 1 0 0 0 0 0 0 0 1 1 0 1 0 0 0
- Area 2 (Port A): 0 0 1 0 0 0 1 0 1 0 1 0
- Area 4 (top to bottom): 0 0 1 0 0 0 0 0 0 0 0 1 1 0 0 0

Environmental variable settings:
SET TABLET=ASCII COM1 48 36 16 for COM1: connection
MODE COM1:9600,E,7,1
SET TABLET=ASCII COM2 46 36 16 for COM2: connection
MODE COM2:9600,E,7,1

CalComp DrawingBoard II (48"x36")

Soft switch settings (18 switches per bank, 2 banks):

Bank A: 1 1 0 0 0 1 0 1 1 1 0 0 1 0 0 0 0 0

Bank B: 0 0 1 0 0 1 0 1 0 0 0 0 0 0 0 0 0 0

Environmental variable settings:

SET TABLET=ASCII COM1 48 36 4 for COM1: connection

MODE COM1:9600,E,7,2

SET TABLET=ASCII COM2 48 36 4 for COM2: connection

MODE COM2:9600,E,7,2

It is expected that other DrawingBoard II models will use the same soft switch settings.

Calcomp Estimat (36" x 30")

Soft Switch Settings

Bank A: 1 1 0 0 0 1 0 1 1 1 0 0 1 0 0 0 0 0

Bank B: 0 0 1 0 0 1 0 1 0 0 0 0 0 0 0 0 0 0

Environmental variable settings:

SET TABLET=ASCII COM1 36 30 4 for COM1: connection

MODE COM1:9600,E,7,1

SET TABLET=ASCII COM2 36 30 4 for COM2: connection

MODE COM2:9600,E,7,1

SummaGraphics SummaSketch Professional (18"x12")

DIP Switch Settings (0=off, 1=on):

Bank 1: 1 1 1 0 0 0 0 0

Bank 2: 0 0 0 0 0 0 0 0

Bank 3: 0 0 0 0 0 0 0 0

Environmental variable settings:

SET TABLET=MMBINARY COM1 18 12 4 500 for COM1: connection

MODE COM1:9600,O,8,1

SET TABLET=MMBINARY COM2 18 12 4 500 for COM2: connection

MODE COM2:9600,O,8,1

SummaGraphics Bit Pad Plus (12"x12")

No switch settings required.

Environmental variable settings:

SET TABLET=BITPAD COM1 12 12 4 200 for COM1: connection

MODE COM1:9600,N,8,1 (REQUIRED)

SET TABLET=BITPAD COM2 12 12 4 200 for COM2: connection

MODE COM2:9600,N,8,1 (REQUIRED)

Summagraphics SummaGrid IV (24"x36")

(Model CEM2436). Dip switch settings (0=off, 1=on):

Bank A: 1 0 0 1 1 1 0 0

Bank B: 1 0 0 0 0 1 0 0

Bank C: 0 0 0 0 0 0 1 0
Environmental variable settings:
SET TABLET=MMBINARY COM1 24 36 4 500 for COM1: connection
MODE COM1:9600,O,8,1
SET TABLET=MMBINARY COM2 24 36 4 500 for COM2: connection
MODE COM2:9600,O,8,1

It is expected that these settings will be the same for other SummaGrid models.

Note:

Before using the SummaGrid, the DOS program supplied with the digitizer must be run to set the appropriate tablet protocols. Check the tablet manual prior to testing.

Appendix C Bibliography

CalComp Inc. CalComp 2500 Series User's Manual.

Kelson, V. A, H.M. Haitjema, and S.R. Kraemer, GAEP: A Geographic Preprocessor for Groundwater Flow Modeling, *Hydrological Science and Technology*, 8(1-4):74-83, 1993.

Summagraphics Corporation. Bit Pad Plus User's Guide. 1987.

Summagraphics Corporation. MM1812 Data Tablet (MM Format) Technical Reference. 1986.

Appendix C Acknowledgments

This document describes the operation of the GAEP program with a number of different hardware configuration systems. The GAEP program was developed by Vic Kelson at the SPEA Groundwater Modeling Laboratory, Indiana University. The author acknowledges Phil DiLavore for his work on the initial design of GAEP. Thanks also to Jack Wittman of IU and Dr. Stephen R. Kraemer of the USEPA for assistance and guidance with this work.

CalComp is a trademark of CalComp Inc.

Summagraphics, SummaSketch, MM1812, and Bit Pad Plus are trademarks of Summagraphics Corporation.

Microsoft and MS-DOS are trademarks of Microsoft Corporation 80386 and 80486 are trademarks of Intel, Inc.