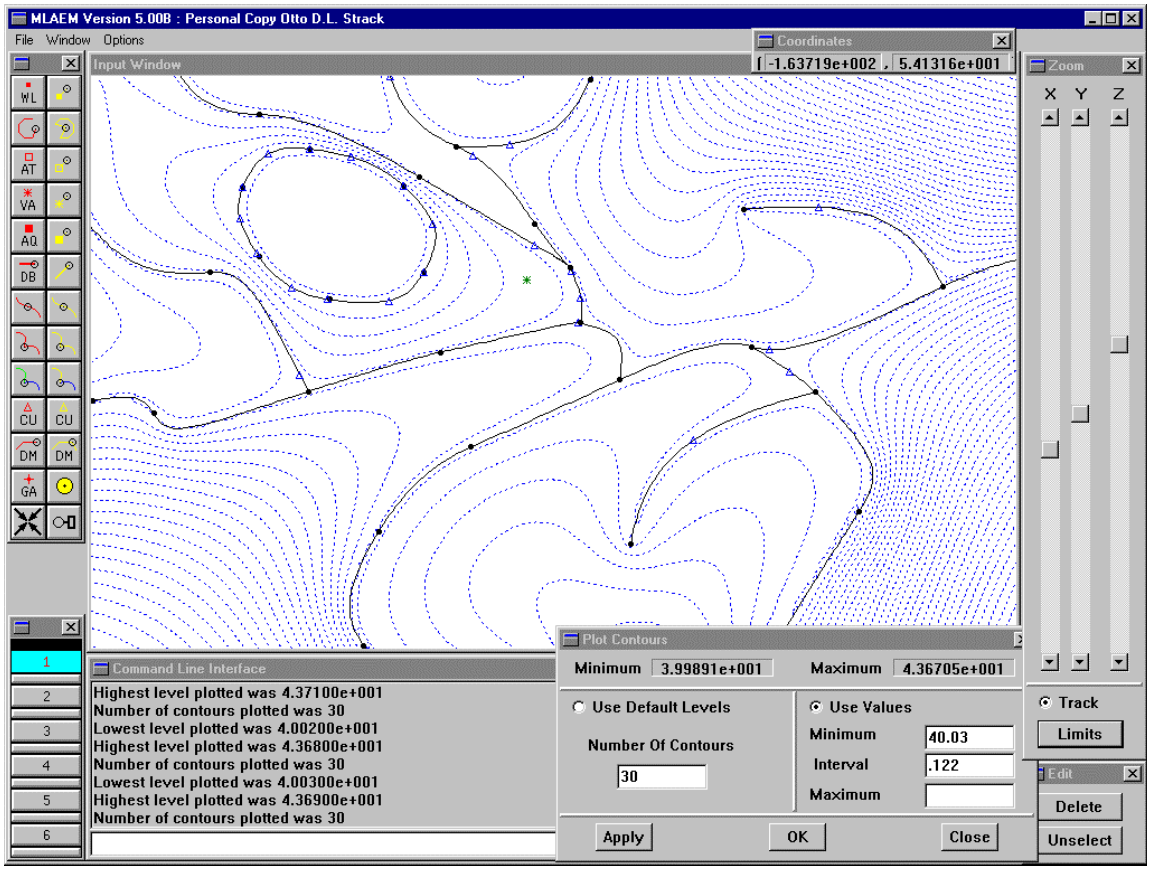
A TUTORIAL TO
MLAEM AND SLAEM

by
O. Erik Strack
and
Frank D. Strack



MLAEM Version 5.00B : Personal Copy Otto D.L. Strack

Coordinates: -1.63719e+002 , 5.41316e+001

Command Line Interface

```
Highest level plotted was 4.37100e+001
Number of contours plotted was 30
Lowest level plotted was 4.00200e+001
Highest level plotted was 4.36800e+001
Number of contours plotted was 30
Lowest level plotted was 4.00300e+001
Highest level plotted was 4.36900e+001
Number of contours plotted was 30
```

Plot Contours

Minimum 3.99891e+001 Maximum 4.36705e+001

○ Use Default Levels
● Use Values

Number Of Contours: 30

Minimum: 40.03
Interval: .122
Maximum:

A TUTORIAL TO MLAEM AND SLAEM

Table of Contents

Part One

Part Two

INTRODUCTION

This is a manual for the computer programs MLAEM (**M**ulti **L**ayer **A**nalytic **E**lement **M**odel)and SLAEM (**S**ingle **L**ayer **A**nalytic **E**lement **M**odel). First, a word of caution. The program should not be used without a familiarity with the textbook *Groundwater Mechanics*, by O.D.L. Strack, published by Prentice Hall, 1989. The book contains the theory of the Analytic Element Method (AEM). This manual, and the program, assume familiarity with this text and the theory. An extensive description of how the AEM is used to model groundwater flow in a large, multi-layered aquifer system can be found in *A Groundwater Model of The Netherlands*, by W.J. de Lange, RIZA, note NR. 90.066, Lelystad, The Netherlands, 1991. This manual contains several parts of, and references to, this text.

The manual is divided into two parts. The first part is a summary of the groundwater mechanics principles on which the program is based, and a tutorial to the structure and use of the program. This part of the manual shows what is possible with MLAEM and SLAEM.

The second part is a more detailed explanation of the different modules of the program. Here each analytic element and its uses are described in detail. Comments and some explanation is given for the other modules. Part Two is intended as a reference manual and the reader who is familiar with the program may wish to skip Part One and use Part Two as such.

The manual refers to both SLAEM and MLAEM by name. In most cases the names are interchangeable; SLAEM and MLAEM are identical, except that SLAEM can only handle a one aquifer system and does not contain "leaky" elements for modeling leakage of groundwater between layers.

SECTION 1 - INTRODUCTION TO THE ANALYTIC ELEMENT METHOD

1.1 SUMMARY OF THE METHOD

The analytic element method, on which the program MLAEM is based, is a technique for modeling groundwater flow in two and three dimensions. It is particularly suitable for modeling flow in large domains, and was originally developed for two-dimensional modeling of regional groundwater flow. To this original development has been added the possibility to model multi-aquifer systems in which resistance to flow in the vertical direction cannot be neglected. MLAEM is a program to model a multi-aquifer system where two-dimensional, steady-state flow using the Dupuit-Forchheimer assumption occurs in each aquifer. The aquifers are assumed separated by initially impermeable boundaries - they are connected using leakage areas implemented by the user. SLAEM is the single-layer version of MLAEM. This manual can be used for both MLAEM and SLAEM, with the exception of the section Modeling a Multi-Aquifer System, which applies only to MLAEM.

The method is based on the superposition of analytic functions, each representing a particular geohydrological feature in an infinite aquifer, and each satisfying either the Poisson or Laplace equations. These analytic functions are referred to as analytic elements; analytic elements have been developed to model uniform flow, rainfall infiltration, rivers, creeks, lakes or polders, wells, cracks, slurry walls, and inhomogeneities in the aquifer properties.

Each function is selected such that it is suitable to simulate the effect of the aquifer feature in question. To solve the system of superimposed functions, or elements, the boundary conditions are specified at selected points (called control points) of the boundaries of the aquifer features. This gives one equation per control point. The total number of control points is equal to the total number of parameters in the solution. This system of equations is solved for the unknown parameters.

Once all parameters are determined, an approximate analytic solution of the problem is known. Although approximate, the solution is truly analytic. Potentiometric heads can be computed at any point in each aquifer without need for interpolation, and velocities can be obtained by analytical differentiation.

1.2 BASIC EQUATIONS - DARCY'S LAW, MASS BALANCE

The principles presented in this section can be found in detail in *Groundwater Mechanics*. The summary is given of modeling groundwater flow in single aquifer systems. MLAEM is just several of these systems connected with leakage functions.

Groundwater flow in shallow aquifers is mainly horizontal. Therefore the Dupuit-Forchheimer assumption can be used, which is interpreted such that vertical resistance to flow is neglected. Aquifers are therefore two- dimensional and isotropic, with stationary flow occurring. The basic equations are as follows:

The Mass Balance Equation:

$$\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} = 0 \qquad (1.1)$$

where Q_x, Q_y are the components of the discharge vector in x and y

directions. Its magnitude equals the amount of flow through a section of unit width occurring over the saturated thickness of the aquifer.

Darcy's Law

$$Q_x = -kh\frac{\partial \phi}{\partial x} \quad Q_y = -kh\frac{\partial \phi}{\partial y} \qquad (1.2)$$

where

k = hydraulic conductivity
h = saturated thickness
ϕ = hydraulic head

1.3 CONFINED AND UNCONFINED FLOW

The thickness h over which the groundwater flow occurs is constant in the case of confined flow (Figure 1, left side):

$$h = H \tag{1.3}$$

In the case of unconfined flow the depth h is variable and equal to he head ϕ (Figure 1, right side

$$h = \phi \tag{1.4}$$

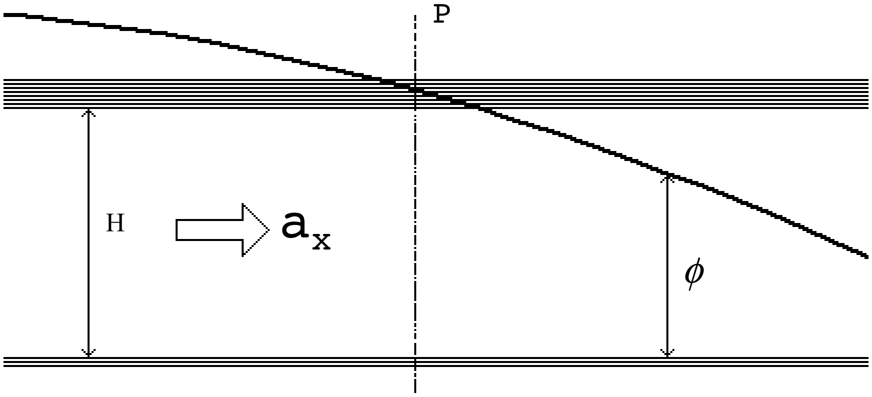


Figure 1 *Confined and Unconfined Flow*

Darcy's law (1.2), can also be written as

$$Q_x = -\frac{\partial \Phi}{\partial x} \qquad Q_y = -\frac{\partial \Phi}{\partial y} \tag{1.5}$$

The discharge potential Φ can be expressed in terms of k, H, and ϕ in the case of confined flow by substitution of (1.2) and (1.3) in (1.5):

$$\Phi_{\text{confined}} = kH\phi + c_c \tag{1.6}$$

and in terms of k and ϕ in the case of unconfined flow by substituting (1.2) and (1.4) in (1.5):

$$\Phi_{\text{unconfined}} = \tfrac{1}{2}k\phi^2 + c_u \tag{1.7}$$

The constants C_c and C_u can be used to satisfy the condition of the continuity in ϕ at the boundary between confined and unconfined flow (see Figure 1, point P). This leads to $(c_u = 0)$:

$$\Phi_{\text{c}} = kH\phi - \tfrac{1}{2}kH^2 \qquad (\phi \geq H) \tag{1.8}$$

$$\Phi_u = -\tfrac{1}{2}k\phi^2 \qquad (\phi \langle H) \tag{1.9}$$

Thus the potential Φ describes groundwater flow in both confined and unconfined conditions, and is a continuous function over the boundary between the two.

1.4 GOVERNING EQUATIONS - LAPLACE AND POISON EQUATIONS

By writing Darcy's Law (1.2) in terms of the potential Φ, we can combine it with the Mass Balance Equation for groundwater flow without infiltration: Laplace's Equation for two-dimensional flow in Φ:

$$\frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} = 0 \tag{1.10}$$

To obtain the governing equation with infiltration into the aquifer, it is noted that the mass balance equation changes to

$$.\frac{\partial Q_x}{\partial x} +. \frac{\partial Q_y}{\partial y} = N \tag{1.11}$$

where N is the amount of infiltration. This leads to Poisson's Equation, the governing equation for flow with infiltration:

$$\frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} = -N \tag{1.12}$$

Every analytic element, representing a particular feature of the aquifer, is a solution to either Laplace's or Poisson's equation. By the Principle of Superposition, if Φ_1 and Φ_2 are solutions of Laplace's equation then so is an addition (multiplied by constants) of the solutions:

$$c_1 \Phi_1 + c_2 \Phi_2 = \Phi \tag{1.13}$$

Addition of solutions of Laplace's equation to the Poisson equation renders a function that satisfies the latter. It follows that the different analytic elements can be combined into an analytic solution of Poisson's equation.

1.5 THE POTENTIAL AND THE STREAM FUNCTION

The potential is written as a sum of harmonic functions (functions that fulfill Laplace's equation) and one or more functions that fulfill Poisson's equation. Each harmonic potential Φ can be represented as the real part of a complex potential $\Omega = \Omega(z)$ that is an analytic function of the complex variable $z = x + iy$ defined in the physical plane. Ω may be written as:

$$\Omega = \Phi + i\Psi \tag{1.14}$$

where Ψ is the stream function. The stream function is constant along streamlines and streamlines are perpendicular to equipotentials, Figure 2. The functions Φ and Ψ satisfy the Cauchy-Riemann conditions:

$$\frac{\partial \Psi}{\partial x} = -\frac{\partial \Phi}{\partial y} = Q_y \qquad \frac{\partial \Psi}{\partial y} = -\frac{\partial \Phi}{\partial x} = Q_x \tag{1.15}$$

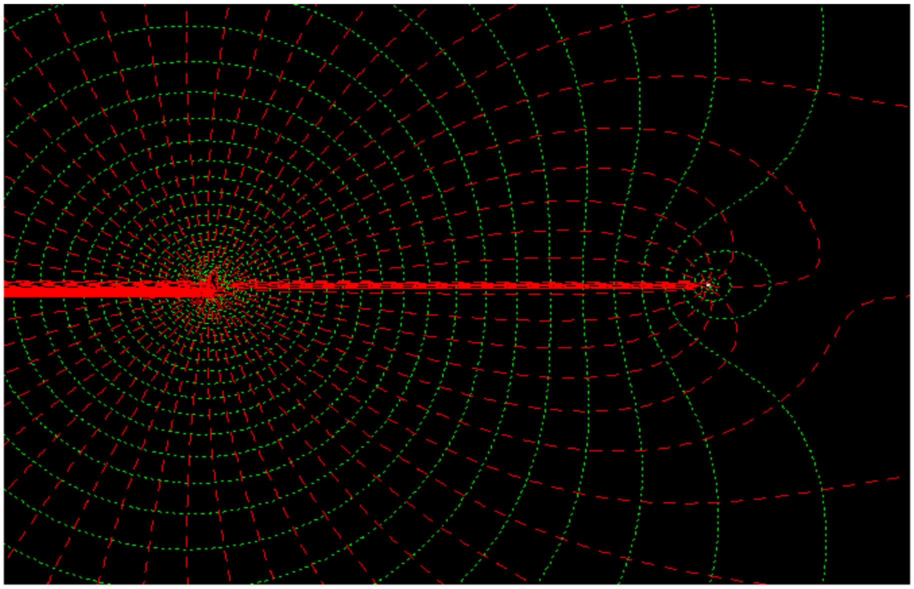


***Figure 2**. Equipotentials and streamlines. The heavy ragged line is a branch cut; appearing as a result of the many-valuedness of the stream function caused by features, in this case wells, that add or remove water from the aquifer.*

An analytic element is either a complex potentials Ω or a particular closed - form solution of the Poisson equation.

In conclusion, a solution to a problem in groundwater flow as described here consists of a combination of analytic elements. Superposition of the complex potential Ω of all analytic elements leads to the complete solution in terms of z . Then at any point in the aquifer the values of the potential can be computed.

In a multi aquifer system, local leakage areas connect the aquifers.

SECTION 2 - A SUMMARY OF MLAEM'S INTERFACE

As was stated in the introduction, MLAEM is based on the analytic element method. Using the analytic element method, separate analytic elements representing aquifer features are combined into a system, or model. The model is then solved, and results are produced. MLAEM is organized into a windows interface that best accommodates this purpose.

There are three pull-down menus in MLAEM and seven main windows that control the operation of the program. The operation consists primarily of the user entering aquifer data and analytic elements into the **Input Window**, occasionally entering the command line interface to perform some selected tasks. After the desired elements have been added, the problem is solved and the user toggles to the **Results Mode** and views the solution in the **Results Window.** To familiarize the user with the layout of the interface, we will briefly discuss each of the features and its purpose.

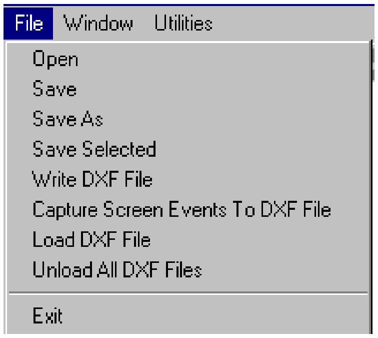
Window Configuration

The size and positions of all the windows in the program can be changed and then saved by using the **Save Window Configuration** option under the **Window** pull-down menu. If some of the windows that should be visible are not, then please see **Appendix A**.

2.1 THE PULL-DOWN MENUS

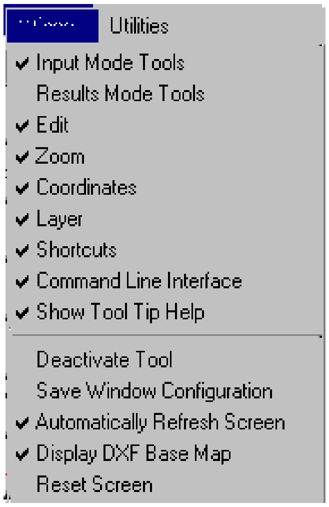
The three pull-down menus in MLAEM are called **File**, **Window** and **Utilities**. They are described below.

The **File** menu consists of options that allow the user to save or read in data pertaining to MLAEM:



- **Open** opens an ASCII element file.
- **Save** saves all the data in MLAEM to the currently loaded ASCII file in a format that the program can then read in at a later time. The currently loaded file is shown at the top of the graphics window. Be careful not to overwrite a file which you don't want to change.
- **Save As** saves all the data in MLAEM to a file selected by the user.
- **Save Selected** saves only the selected (highlighted) elements to an ACSII file.
- **Write DXF File** writes out everything currently in the model, including DXF base maps and contour plots (if they are currently being displayed) to an ASCII DXF (AutoCAD) file. Note: this utility behaves differently in the Input and Results Modes - in the Input Mode the entire contents of the model are written with at the scale of the current window, and in the Results Mode only the features currently visible in the display are written to a DXF file.
- **Capture Screen Events To DXF File** captures traces (both sectional and in plan) when this option is checked, in addition to performing the same actions as the option "Write DXF File". Remove the check mark to stop the capture of screen events.
- **Load DXF File** reads in one or more DXF (AutoCAD) formatted files. Each DXF map is displayed in a different color.
- **Unload All DXF Files** unloads all DXF files from the program.
- **Exit** stops the program.

The **Window** menu's purpose is to allow the user to control the program's window attributes. Each option in the **Window** menu is activated when a there is a check immediately to the left of the selection, and deactivated when there is not:

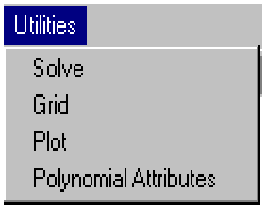


- **Input Mode Tools** is the mode in which the user enters the aquifer data and is characterized by a dark gray screen. The check mark indicates that the tools buttons are visible - remove the check mark to hide the tools.
- **Results Mode Tools** is the mode in which the user views the results and can generate water particle traces. This mode is characterized by a black screen. The check mark indicates that the tools buttons are visible - remove the check mark to hide the tools.
- **Edit** hides or shows the edit window in the lower right corner of the screen.
- **Zoom** hides or shows the **Zoom** window on the right side of the screen.
- **Coordinates** hides or shows the **Coordinates** window at the upper right of the screen.
- **Layer** hides or shows the layer box in the lower left-hand corner of the screen.
- **Shortcuts** hides or shows the shortcut window.
- **Command Line Interface** hides or shows the command line interface at the center bottom of the screen.
- **Show Tool Tip Help** hides or shows the “tool tip” or bubble help which is visible when the user hovers the mouse over a button or scroll bar in the interface. Note: The tool tips can cause undesired screen redraws to occur - disable the tool-tip help in order to prevent this.

The commands in the bottom section of the menu are for adjusting the way the program draws on the screen.

- **Deactivate Tool** deactivates the current tool and returns all the tool buttons back to their up position.
- **Save Window Configuration** saves the current size and location of the windows in the program, so the user can customize the appearance of the interface.
- **Automatically Refresh Screen** continually redraws the screen while the zoom controls are being used. This assures that only one plot is drawn on the screen at once. When deactivated, different types of solutions may be superimposed on each other (i.e., **figure 2** uses this feature to draw both equipotentials and streamlines on the same graph.)
- **Display DXF Base Map** displays the DXF formatted (AutoCAD formatted) maps in the **Input Window** (if any have been loaded**).**
- **Reset Screen** removes contours and trace particles from the screen, leaving the analytic elements and any DXF base map that has been loaded.

The **Utilities** menu:

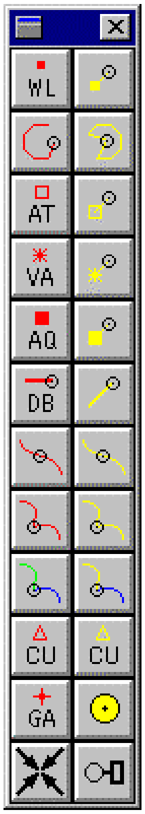


- **Solve** shows or hides a dialog used to solve the system of equations.
- **Grid** shows or hides a dialog used to create a grid necessary for plotting.
- **Plot** shows or hides a dialog used to generate a plot of the solution that has been produced.
- **Polynomial Attributes** allows the user to set the default orders and degrees of overspecification for the doublet and curvilinear elements.

2.2 THE INPUT TOOLKIT WINDOW

Most analytic elements are added to the model using the input toolkit, which can be found in the upper left of the program interface, just below the pull-down menus. The tools are placed in a 12x2 array of buttons. Here the notation (i,j) is used to refer to the button on the ith row of the jth column. For example, **tool (1,1)** refers to the button in the upper left corner of the toolkit, and **tool (12,2)** refers to the button in the lower right corner of the toolkit.

For the tools in rows one through ten, the column one button represents the *add* tool and column two represents the *move*, or *edit* tool of that element. Moving an element will not affect its associated data but only its location or its shape within the aquifer. The elements associated with each button can also be selected by pressing the right mouse button while using the tool. A tool can be selected by clicking on it once; the button will be pressed down. Once a button has been pressed down, it can be turned back to its up position by either selecting **Disable Tool** from the **Window** pull-down menu, or by pressing down any other button.



Well adding tool- tool position (1,1)

This tool is used to add wells to the aquifer in the **Input Window**.

Well moving tool - tool position (1,2)

This tool is used to move wells that have already been added to the aquifer.

Add polygon tool - tool position (2,1)

This tool is used to create polygons. Polygons are used to designate an area in the aquifer where a feature characterized by a boundary polygon will be added, such as a pond, lake, or inhomogeneity.

Move polygon tool - tool position (2,2)

This tool is used to adjust existing polygons. Clicking once inside a polygon will select it. Click multiple times inside nested polygons to "scroll" through the polygons containing the mouse.

Add aquitard data point tool - tool position (3,1)

This tool is used to add aquitard points. Aquitard data points are used to create an interpolation for the properties of variable strength area elements (VARELS). A single polygon must first be selected before adding any points, but points may be added outside the selected polygon. NOTE: To select a polygon, move the cursor within the desired polygon, and click once with the right mouse button.

Aquitard data point moving tool - tool position (3,2)

This tool is used to move aquitard points that have already been added to the aquifer.

Add variable strength area element (VAREL) tool - tool position (4,1)

This tool is used to add VAREL control points. VARELS are used in areas where water is infiltrated into or out of the aquifer, such as a lake or rainfall area. Again, a single polygon must be selected before adding any points, and they must be added within it.

VAREL moving tool - tool position (4,2)

This tool is used to move VAREL data points that have already been added to the aquifer.

Add poly aquifer tool - tool position (5,1)

This tool is used to add information about an inhomogeneity (poly aquifer) in the aquifer. Again, a single polygon must be selected before adding the point. However, the location of the point in the polygon has no bearing on the solution of the problem.

Poly aquifer moving tool - tool position (5,2)

This tool is used to move poly aquifer data points that have already been added to the aquifer.

DOUBLET tool - tool position (6,1)

A DOUBLET is a mathematical function that is used to create a continuity of head between the inhomogeneity and the rest of the aquifer. In most cases, it is useful to add the DOUBLET to the inhomogeneity, however, there are some cases when this is not necessary, and an example will be given in the section on DOUBLETs.

DOUBLET moving tool - tool position (6,2)

This is for adapting the DOUBLET to account for adjustments made to the polygon in which the inhomogeneity data point has been placed.

Curvilinear Strings:

A curvilinear element (CUREL) operates on much the same principle as the VAREL. But where the VAREL is an area representing a large analytic element such as a lake or rainfall area, the CUREL represents a narrow element in the aquifer such as a river or an impermeable wall. It is important to remember that the curve tools only represent the geometry of the element, the CUREL data points are used for adding parameter values to the geometry. Each Curvilinear string is composed of one or more connected curve segments containing two "nodes" each.

Add smooth string tool - tool position (7,1)

This tool adds smooth strings (that is, strings that have a continuous slope at the nodes).

Move smooth string tool - tool position (7,2)

This tool is used to adjust smooth strings - if it is used on a string that is not smooth, it will attempt to make the string smooth.

Add discontinuous string tool - tool position (8,1)

This tool is used to add strings that have sharp bends at the nodes (that is, strings that are discontinuous at the nodes). When creating smooth strings, it is usually it is best to use this tool first and to "smooth out" the string with the **move smooth string tool** after the string has been completed.

Move discontinuous string tool - tool position (8,2)

This tool is used to adjust discontinuous strings.

Add separate string tool - tool position (9,1)

This tool is used to add separate curvilinear strings. This is useful when two strings are connected, but do not have the same properties.

Move separate string tool - tool position (9,2)

This tool is used to adjust separate curvilinear strings, or to separate a single curvilinear string into two strings.

Add curve data point tool - tool position (10,1)

This tool is used to specify the parameter values of a CUREL. Unlike the VARELs and the DOUBLETs, there is no need to select an element before adding the data point; it will add itself to the nearest curve.

Move curve data point tool - tool position (10,2)

This tool is used to move CUREL data points that have already been added.

Global aquifer tool - tool position (11,1)

This tool is used to add global aquifer data to a layer. NOTE: When writing elements to an ASCII file, the aquifer data are automatically saved.

Selection (highlighting) tool - tool position (11,2)

This tool is used to highlight (select) elements. Its primary use is to highlight elements that can then be deleted using the **Edit** window. Only one element is selected or de-selected per click of the mouse. If multiple elements are "stacked" on top of one another, you will have to use the element tools in conjunction with the right mouse button in order to select each element individually. Pressing down the right mouse button while using any tool highlights the elements associated with that tool

.**Center tool** - tool position (12,1)

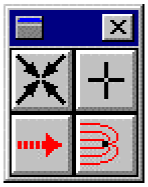
This tool is used to move the area of interest to the center of the screen. The location at which this tool is clicked on in the gray screen becomes the new center of the screen.

Edit tool - tool position (12,2)

This tool is used to edit, view, or adjust the properties of any element in the model. View the element by clicking on it with this tool. If any elements are highlighted, then only those elements will be displayed.

2.2 THE RESULTS TOOL WINDOW

The cursor tool window is found by choosing **Results Mode Tools** from the **Window** pull-down menu. This window contains the tools used to view the results of a solution.



Center tool - tool position (1,1)

This tool is used to move the area of interest to the center of the screen. The location at which this tool is clicked on in the black screen becomes the new center of the screen.

Crosshair tool - tool position (1,2)

This tool is used to get coordinates for the TRACE module. It is used by clicking at the desired location on the screen and then typing a command (in the TRACE module) that will use the coordinates clicked upon.

Tracing tool - tool position (2,1)

This tool is used to generate water particle traces. The path of a water particle will be generated in red from the point clicked upon.

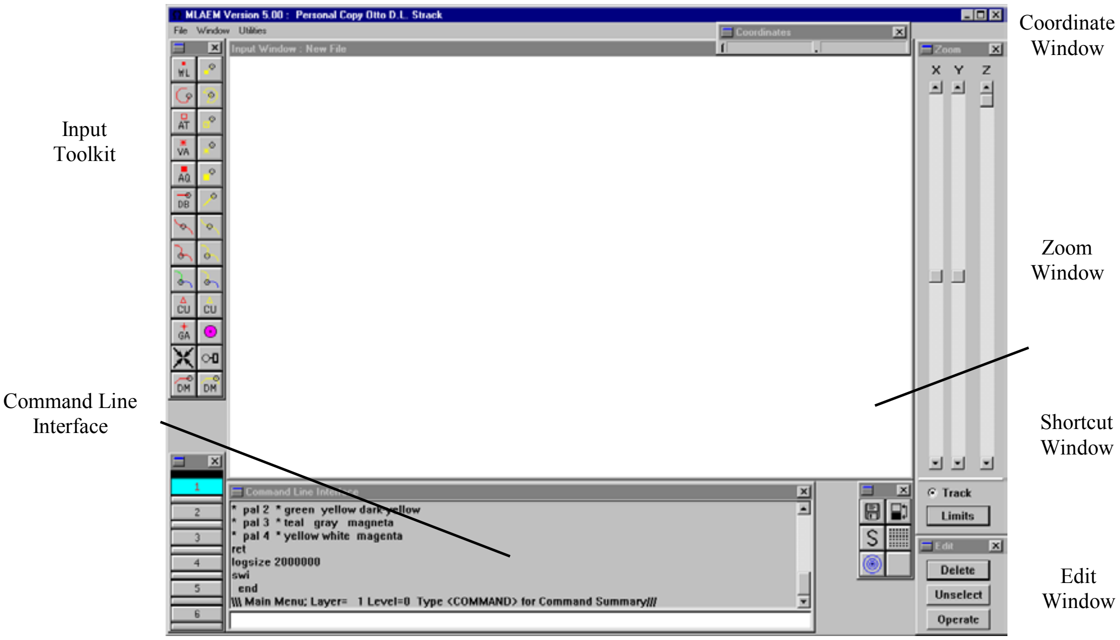
Well pathline generation tool - tool position (2,2)

This tool is used to generate ten pathlines that end up at the well clicked upon.

We will go through an example in *Section 3* where we use all the tools, and where the specifics of how to use the tools will be discussed.

2.3 LAYER, COMMAND INTERFACE, ZOOM, EDIT, SHORTCUT AND COORDINATES WINDOWS

We will now discuss the remaining six windows of MLAEM's interface: the **Layer** window, the **Command Line Interface**, the **Zoom** window, the **Edit** window, the **Shortcut** window and the **Coordinates window**.



```
* pal 2  * green  yellow dark-yellow
* pal 3  * teal   gray   magneta
* pal 4  * yellow white  magenta
ret
logsize 2000000
swi
 end
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
```

***Figure 3a**. The MLAEM interface for the user's reference*

The Layer Window

The **Layer** widow is found at the lower left of the screen. It is used to select which layer or boundary the analytic elements will be added to.

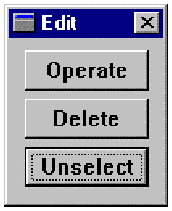
Each of the buttons in the window represent a layer or a boundary. Specifically, the numbered buttons represent each layer of the aquifer, and the thin, unnumbered buttons represent either the top or bottom boundary. To change the layer or boundary into which the element is to be entered, move the mouse into the **Layer** window, and click on the desired button.

The Command Line Interface

The **Command Line Interface** is found in the lower center of the screen. This window can be used to enter data pertaining to the MLAEM, but it is mostly used to enter several routines that are not implemented into the windows interface. Operation of the Command Line Interface is based on modules (subgroups in the program which are dedicated to particular tasks) which the user enters and where commands are given. For a complete discussion of the command line interface, see Part Two, section one, **The Modular Structure of MLAEM**.

The Edit Window

The **Edit Window** contains two or three buttons that allow the user to modify the elements in the **Input Window**: *Operate*, *Delete*, and *Unselect*. *Operate* allows the user to either multiply all the highlighted given strength wells by a factor, or to add a constant to the strength of all the wells highlighted in the model. *Delete* allows the user to delete all the highlighted elements in the model, and *Unselect* returns all the highlighted objects to normal.



The Zoom Window

The zoom controls are found at the far right of the screen. They are used to adjust the area of the modeling domain that is displayed on the screen. The **X** scroll bar allows the user to shift the area of the modeling domain to the left or right, the **Y** scroll bar allows the user to shift the area of the modeling domain up or down, and the **Z** scroll bar allows the user to zoom in and out of an area of interest, effectively magnifying or de-magnifying the area of interest.

The setting of the **Tracking** button determines whether the screen will be redrawn while using the **X**, **Y**, and **Z** scrollbars. When **Tracking** is activated (black dot is shown immediately to the left of the option) the screen will be drawn continually as the scrollbars are being used. When **Tracking** is not activated, the screen will only be redrawn when the button is released. This option is very useful when a complicated plot or DXF base map is being used, because it may take too long to redraw.

The **Limits** button is used to set the maximum amount of zooming allowed, and to set the center of the modeling domain. To change the global center point of the model, adjust the coordinates of the point accordingly.

The **Radius** button controls the maximum radius of the area that may be viewed; as **Radius** is made larger, more of the domain can be viewed by zooming. The button **Take Current Zoom Level** keeps the viewing domain exactly as is, but returns the zoom buttons back to their default position: **Z** is all the way to the top, and **X** and **Y** are centered (Figure 3c.)

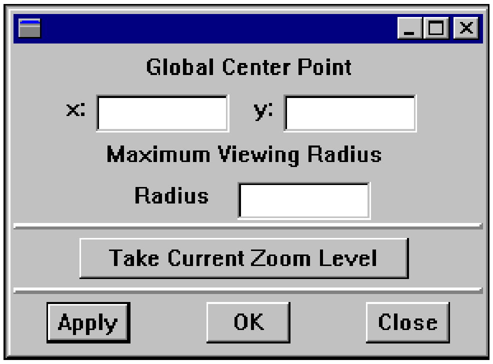


Figure 3b. *The limits window.*

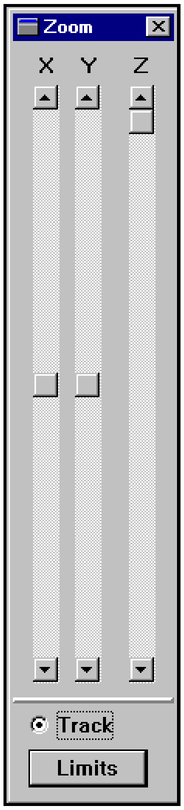
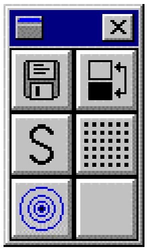


Figure 3c. *The zoom window with the buttons in their default position.*

The Shortcut Window



The shortcut window contains the following shortcut buttons:

Save - button position (1,1)

This is a shortcut to the **Save** option under the **File** pull-down menu. Be careful not to overwrite a file which you do not want to change - no warning will be issued.

Toggle Tool Mode - button position (1,2)

This button toggles between the **input mode tools** and the **results mode tools**.

Solve - button position (2,1)

This shortcut is the same as clicking on **Apply** in the solve dialog box (which is accessed by choosing **solve** under the utilities pull-down menu).

Grid - button position (2,2)

This shortcut is the same as clicking on **Apply** in the grid dialog box (which is accessed by choosing **grid** under the utilities pull-down menu).

Plot - button position (3,1)

This shortcut is the same as clicking on **Apply** in the plot dialog box (which is accessed by choosing **plot** under the utilities pull-down menu).

Reset Screen - button position (3,2)

This is a shortcut to the **Reset Screen** option under the **Window** pull-down menu.

The Coordinate Window

Whenever a tool is being used, the **Coordinates** window shows the coordinates of the cursor within the **Input Window.** This is so that the user can accurately place the analytic elements into the model**.** Scientific notation is used to indicate the value of the coordinates.

This concludes the summary of the MLAEM interface, and in the next section we will learn to use the interface while solving a simple problem.

SECTION 3 - TUTORIAL TO SOLVING A PROBLEM

This is an introduction to using the program to solve a problem. Solving a problem using MLAEM is called modeling because the process involves building a working model of the aquifer system. For the rest of the sections in Part One the manual will explain how to use MLAEM to model a single aquifer, as if it were a Single Layer Analytic Element Model (as the program SLAEM is). In MODELING MULTI AQUIFER SYSTEMS it will be explained how to set up a multi-layered system, how to link the layers, and how to view the results. The general steps to the modeling process for a single aquifer are as follows:

3.1 STEPS IN THE METHOD

1) *Entering aquifer data*: A model aquifer is made by first entering the values that predominantly occur in the aquifer, and then by later adjusting local differences in the base height, hydraulic conductivity, thickness and porosity. These values should hold for most of the model. For example, if most of the aquifer has a base of zero with areas of different height then a base of zero should be entered, and areas inhomogeneous with this should be changed later using Poly-Aquifers and DOUBLETs. For unconfined aquifers the highest expected value for the head should fall below the top of the aquifer (the aquifer must be thick enough).
2) *Adding the reference point*: In order to solve the system of equations, the program needs to know the head at one point, called the reference point.
3) *Entering analytic elements*: The model aquifer is filled with analytic elements, which represent aquifer features such as rivers, lakes, wells, rainfall, and inhomogeneities. These elements can be added in any order, and most combinations are possible. (Bad combinations will be explained in the sections on the analytic element modules)
4) *Solving the problem*: The system of equations constructed using the aquifer data, the reference point, and the analytic elements is solved.
5) *Retrieval of the results:* The results obtained by the solution can be viewed. There are several ways to do this, and they will be explained in the next section, INTRODUCTION TO THE RESULTS MODE.

3.2 A SIMPLE EXAMPLE - POND.DAT

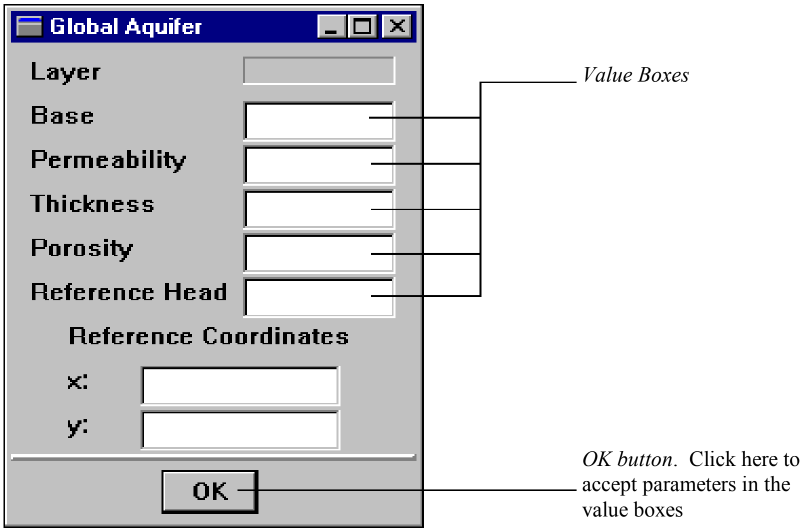
Entering Aquifer Data.

We will now use the above steps to solve a simple problem. The problem consists of a VAREL (representing a the effect of a pond on the aquifer) and a well in a uniform flow field in an unconfined aquifer. At some points in this manual, the VAREL will be referred to as the pond.

First we must make sure that all the elements are entered in the first layer and at the top of the aquifer. If somehow the default layer (layer 1) has been altered, it can be changed by clicking once on the button in the layer window labeled **1**. Make sure that the narrow boundary button above the layer 1 button is black, indicating that the current boundary is at the top. If it is not, then click on it to make it active.

After the layer number has been set, the first step in the modeling process is to set the aquifer parameters. The program works in any set of consistent units; if the hydraulic conductivity is in cm/s, then infiltration from rainfall must be measured in cm/s also.

To enter the aquifer data, click on the global aquifer tool (labeled “GA” in the input toolkit). Immediately, a dialog will pop up.



Value Boxes

OK button. Click here to accept parameters in the value boxes

note: to change the values of the value box, click inside the box with the left mouse button and enter the desired number.

Figure 4. *The Aquifer window*.

In our example, we have unconfined flow. In order to do this we will set the thickness to 100. Do not make the thickness too large, or the base too high or low, because the program uses these parameters and there will be a loss of accuracy. The rest of the parameters we will leave at their defaults (permeability = 1, porosity = 0.3).

Adding the Reference Point

In order to solve the system of equations created by the superposition of the elements, a reference head must be given at a certain point in the aquifer. The reference head together with the uniform flow function (explained below) controls the far-field. The better the flow is defined by analytic elements, the less effect the reference point will have. For a small model, the reference head will have be placed where it best simulates the far-field and gives the best results. In a large model the point must be placed far away, because it should not have any bearing on the solution. The reference head is entered in the **Reference Head** value box and for our example we will use a reference head of 10. The two boxes **x** and **y** refer to the coordinates of the reference point. In this case we will place it at the coordinates (100, 0), so change the **x** box to 100, and leave the **y** box to 0.

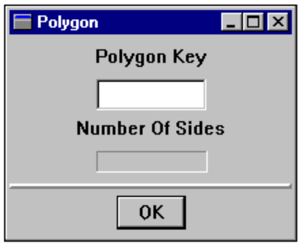
Entering the Analytic Elements

We will have a polygon centered roughly about the point (-45,0), having a diameter of about 30 units, containing a VAREL with an extraction rate of -0.01. [Extraction is defined as positive. Since infiltration is the opposite of extraction (and the pond infiltrates water into the aquifer) the extraction rate must be negative.] We will give the uniform flow field a discharge per unit length of 0.1. Note: the uniform flow function is used because of a lack of definition in the flow pattern. It simulates the effects of distant elements such as wells, rivers (CURELs), and lakes (VARELs). Uniform flow should therefore be considered as part of the far-field solution.

First, we will enter the polygon. Click with the left mouse button on the **polygon** tool, and move the cursor into the **Input Window**. As the tool enters the window, notice that the cursor becomes a circle with the word "poly" attached to it. All the tools have a similar cursor to show which tool is being used in the Input Window.

We will have a four-sided polygon in our model with the four corners located at: (-60,17), (-70, -15), (-30,-17), and (-27, 14). Move the cursor to the coordinates of the first corner (node) of the polygon and click once to attach the node to the aquifer. Now move to the second set of coordinates and click again. Repeat this for all four corners. To complete the polygon, move the cursor back to the first node and click once. If the node is within the circle of the polygon tool, the side being joined will "jump" to the node, closing the polygon.

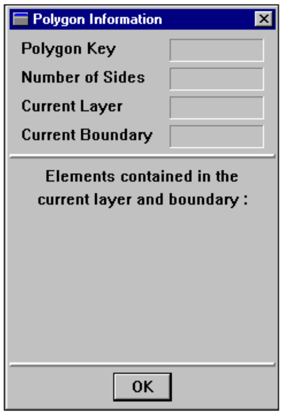
Once the polygon is completed, the **Polygon** window will pop up. The box inside the window labeled **Polygon Key** gives the name dedicated to that particular polygon in the program's memory, and it serves to organize and label the polygons for the user, so they may better be kept track of. Move the cursor into the **Polygon Key** box and type the name. Note that the first character of the polygon's label cannot not be a number.



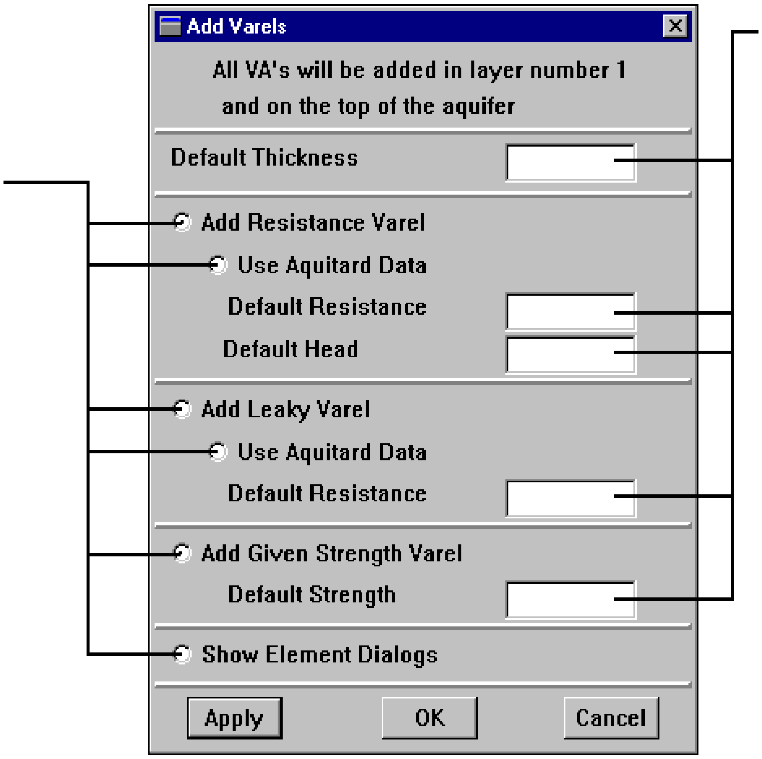
If an error was made entering the polygon, it can be modified using the **move polygon** tool. Click on the tool and move the edit circle on top of the polygon node that is to be moved, then click the left mouse button down, move the mouse to the desired location, and release the button. Nodes can be added to a polygon by clicking the left mouse button down between two nodes, moving the mouse to the desired location of the new node, and releasing the mouse button. Nodes can be deleted by clicking the left mouse button down on a node, moving the node on top of another node, and releasing the button.

If more than one polygon is connected to the node clicked upon, then all polygons connected will be adjusted. However, polygons can be disconnected from other polygons by high-lighting a polygon or polygons and moving the desired node. Only the node connected to the highlighted polygon(s) will be moved. A polygon can be selected while using any other tool by clicking with the right mouse button inside the desired polygon.

Information about the polygon may be obtained by clicking on the **edit** tool (10,2) and then clicking anywhere on the geometry of the polygon. The **Polygon Information** window (which will pop up immediately afterwards) displays all the specifics of the current polygon. **Polygon Key** is the label that is associated with that particular polygon. **Number of Sides** displays the total number of sides that the polygon has. **Current Layer** refers to the layer for which the contained elements will be displayed. **Current Boundary** refers to the boundary for which the contained elements will be displayed. **Elements contained in the current layer and boundary** lists all the elements that are found in that particular polygon, in the layer and boundary specified.



Now that the geometry of our pond has been completed, we must add the VAREL to represent the extraction rate of the pond. Click on the **VAREL** tool and notice that the **Add Varels** window pops up immediately. This window is used to specify the nature of the VAREL that will be added. It is useful to note that this window is typical of the analytic elements; it is shown in Figure 5 and its operation will be discussed. After doing so, it is assumed that the user knows how the windows generally work.



All VA's will be added in layer number 1
and on the top of the aquifer

Default Thickness

○ Add Resistance Varel

○ Use Aquitard Data

Default Resistance

Default Head

○ Add Leaky Varel

○ Use Aquitard Data

Default Resistance

○ Add Given Strength Varel

Default Strength

○ Show Element Dialogs

Options:
In order to choose an option, click on the white circle. The selected option is shown by a black dot in the circle.

Value Boxes show the default value of the element to be added. These can be changed by clicking in the box or hi-lighting the value with the mouse and entering the new value.

Figure 5. *The Add Varels window.*

In most instances, there are several options to choose from in the dialog, and each of these options is usually accompanied by a value box. In this case there are three main options and three secondary options. The three main choices are the option to add a resistance VAREL, a leaky VAREL, or a given strength VAREL. In the resistance VAREL and the leaky VAREL there is also an option to use aquitard (ATARD) data. The parameters used for the element may be either specified in the associated value box or may be interpolated from aquitard data points. Aquitard data points are points in the aquifer where the resistance is known; the program interpolates between those data and obtains results for the VAREL. In that case, the values in the value box are irrelevant and will be ignored. If the ATARD data are not used, then the data entered in the value box will be used directly to obtain the results. The last option, **Show Element Dialogs** determines whether or not to show a similar element dialog every time a VAREL is added to the model. Having the dialog pop up every time is useful when every element is different, but if the VARELs will be the same, or if ATARD data is being used, the user may not want each VAREL window to pop up.

To select an option, click once on the white circle immediately to the option's left. An option is activated when a black dot is shown inside the white circle.

Adding the Variable Strength Area Element

We will be adding a Given Strength VAREL with a default strength of -0.01. Move the mouse into the **Add Varels** window and click on the option **Add Given Strength Varel**. Now move the cursor into the **Default strength** value box and change the resistance to -0.01. The default thickness will be left at its default. Now click **OK** to accept the values and the **Add Varels** window will disappear.

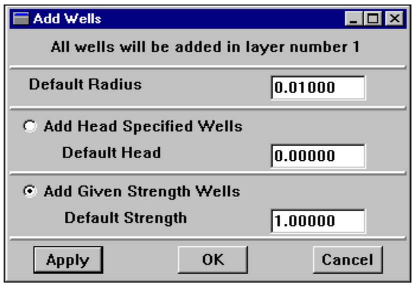
A VAREL must always be added within a selected polygon. To select a polygon, click inside the polygon with the right mouse button. It is selected when it is highlighted in yellow. Move the cursor near the point (-45,0) and click once. A new dialog will pop up: the element dialog. Accept the values in the value boxes or change the values and then click **OK.** The VAREL representing the pond has been added to our model and we are now ready to add a well.

Note that polygon sides cannot be moved off their associated VAREL points. If the user tries to do this, the node being adjusted will stop moving as soon as a VAREL point is about to be removed from the polygon. This also holds for VAREL points in layers other than the current layer.

Adding the Well

To add the well, first click on the **Well** tool. As before, a window will pop up. For our example, we will choose to add a given strength well, so click on the appropriate option and click **OK**.

Now move the cursor to the point (60,0) and click once. Again, an element dialog will pop up. Here we will enter the specific characteristics of the well. We will give the well a strength of 9.5 and a radius of 0.25. Change the values accordingly and click **OK** and the well has been added.



All wells will be added in layer number 1

Default Radius 0.01000

○ Add Head Specified Wells

Default Head 0.00000

● Add Given Strength Wells

Default Strength 1.00000

Adding the Uniform Flow Field

Finally, we must add the uniform flow field to our model. This is done in the **Command Line Interface**. The **Command Line Interface** window is composed of two parts: a part used for displaying output from the program (the upper part of the window) and a part used for typing commands (the long rectangular box, or edit line, at the bottom of the window). Before typing any commands in the **Command Line Interface**, the mouse must be moved down to and clicked in the single edit line at the bottom of the window. As soon as the mouse is moved out of the edit line, the **Command Line Interface** will stop accepting commands from the keyboard.

As far as commands are concerned, words in brackets <> signify that the word is a command. However, only the word inside the brackets need to be typed. For example, if the command is <UNIFLOW>, the user only needs to type UNIFLOW. Also, there are several commands that are accompanied by parameters. In those cases, parentheses enclosing a set of parameters indicate that they are required, and square brackets indicate optional parameters. Thus, the command <UNIFLOW>(Q0)[ANG] consists of the command UNIFLOW, the required parameter Q (discharge) and an optional parameter ANG (angle).

The user is reminded that for a complete discussion of the **Command Line Interface**, they must refer to the Part Two, section one, **The Modular Structure of MLAEM.** Any discussion here made of the interface will be specific to the exercise and will only show the user how to perform that particular task.

Move the mouse into the edit line and click to activate the interface. In order to enter the parameters for a function in the command line, the program must be in the proper module. The uniform flow function is found in the GIVEN module of the program. To enter the GIVEN module, type *GIVEN.* (In the following excerpts, lower case letters indicate user commands and upper case letters indicate program responses; it is recommended to follow this convention whenever using the interface.)

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
given
\\\  Module=GIVEN; Layer=   1   Level=1    Routine=INPUT            ///
<LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET>
<RETURN>
```

Now, to enter uniform flow, or “uniflow”, of 0.1 type *UNI .1*.

```
uni .1
\\\  Module=GIVEN; Layer=   1   Level=1    Routine=INPUT            ///
<LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET>
<RETURN>
```

This completes the information that will be entered in this example.

Solving the Problem

A solution can be obtained by applying the boundary conditions at the control points. (In this case only one, the reference point, and only one equation). Choose the **Solve** option under the **Utilities** pull-down menu in order to display the **solve window**. Accept the defaults and click on Ok to solve the problem. Note that you can also use the **shortcut window** in order to accomplish this - simply click on the button marked **S**.

The program now has a solution in memory. In the next section, **INTRODUCTION TO THE RESULT - VIEWING MODE**, this solution will be used.

Saving the File

In order to save the model you have just created, choose **Save As** from the pull-down menu (or click on the “diskette” button in the **shortcut window**). The model will be saved in ASCII format. Note that the solution is not saved.

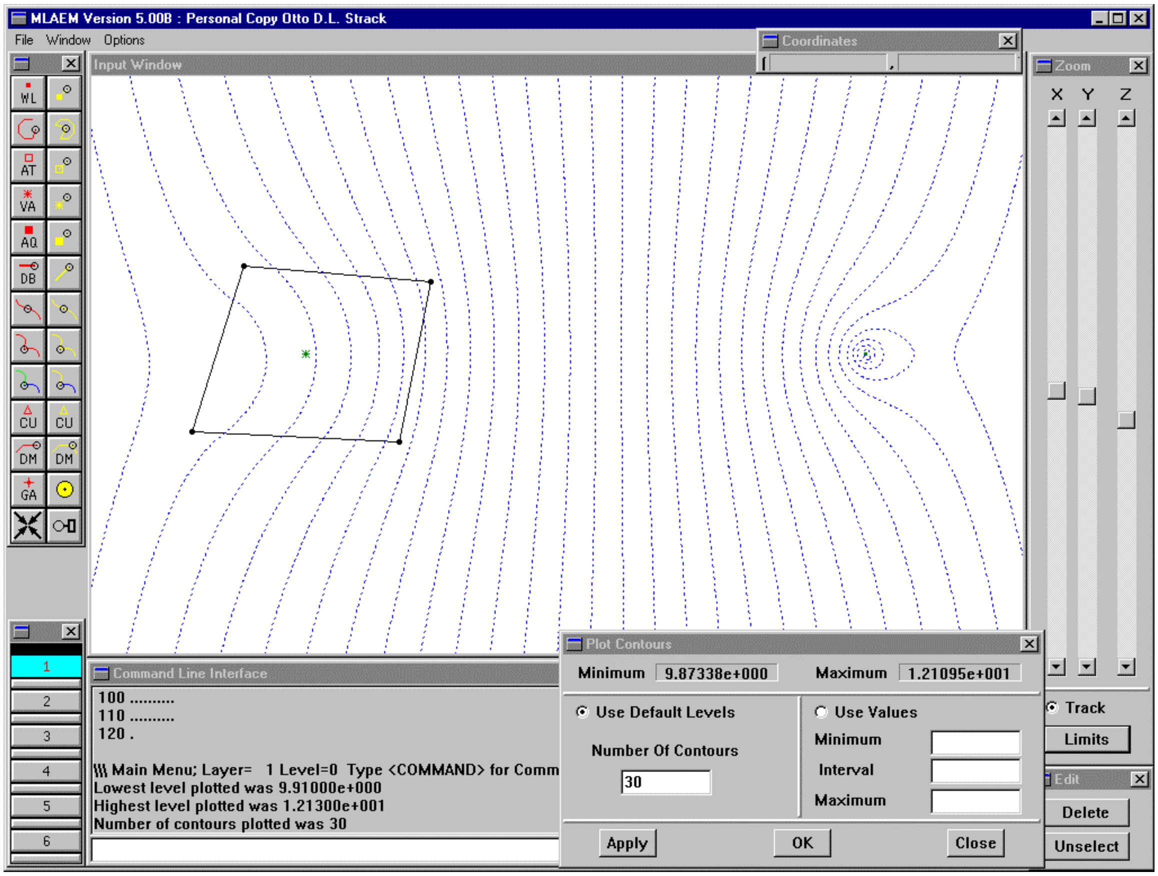
Saving the Solution

The solution can be saved, and then read back in later. The following lines show how to save the solution.

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
save
<SOLUTION><GRID><BOTH><RETURN>
solution
<FILENAME><R>{to abort}
pond.sol
SOLUTION FILE HAS BEEN WRITTEN
PROBLEM:  UNNAMED
```

A word of caution: the program uses different types of binary files to save solutions, grids, and combinations of solutions and grids. The program will produce an error message if one is read in the place of another and data in the program may be corrupted. It is therefore recommended to use different endings for the different types of files. One possibility is to use for solutions .sol, for grids use .grd, for combined solution and grids use .sag (**s**olution **a**nd **g**rid). For conventions used in the groundwater model of the Netherlands (NAGROM) see the text by W.J. de Lange (its reference is in the introduction).

Figure 6 shows the pond and the well entered in the interface. If your model looks drastically different, you may want to go through this section again.



MLAEM Version 5.00B : Personal Copy Otto D.L. Strack

```
100 ..........
110 ..........
120 .

\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Comm
Lowest level plotted was 9.91000e+000
Highest level plotted was 1.21300e+001
Number of contours plotted was 30
```

Plot Contours

Minimum 9.87338e+000   Maximum 1.21095e+001

● Use Default Levels   ○ Use Values

Number Of Contours: 30

Minimum:

Interval:

Maximum:

Figure 6. *The pond and well*. (The pond and well have been enlarged using the **Zoom Window**.)

SECTION 4 - INTRODUCTION TO THE RESULTS MODE

In the preceding section, **TUTORIAL TO SOLVING A PROBLEM**, we entered a sample problem with a pond, well, and uniform flow field. This section explains how to view the solution obtained. If you have not exited the program since the last section, then the solution will still be in memory. If you did, then you can either enter the data again and solve or read it in from a saved ASCII file and solve. You can also read the solution file (pond.sol) via the **Command Line Interface**:

4.1 READING THE SOLUTION

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
read
<S>OLUTION,<G>RID,<B>OTH,<D>IFGRID,<R>ETURN
s
PLEASE ENTER FILENAME; <R> TO ABORT
pond.sol
SOLUTION FILE HAS BEEN READ
PROBLEM:  UNNAMED
```

Now the solution is in the program's memory. We will view the solution graphically, by way of piezometric contours.

4.2 VIEWING GRAPHICAL DATA - GRID AND PLOT

Graphical data for a particular layer can be obtained by means of piezometric contours (contours of constant head), or by means of contours of the potential or the stream function. Plots are produced by the use of a subroutine that contours, using interpolation, a set of values of the function for the current layer. The set of values used corresponds to a rectangular grid.

Making the Grid

In order to display the grid window, choose **Grid** from the **Utilities** pull-down menu. The grid window displays the number of grid points and the type of function to be calculated. Type in 50 for the number of grid points and click on **Ok**.

Now that the grid has been calculated a plot can be made using the plot pull-down menu. We will switch to the Results Mode, even though plots can also be made in the Input Mode. This is done by choosing **Results Mode Tools** from the **Window** pull-down menu or by using the shortcut window. The screen will change to its characteristic black and only the well will be drawn on the screen. Any time the model contains polygon geometry, VARELs, aquifer points, ATARD points, or CUREL data points, they will not appear in the Results Mode screen, until their drawing properties are changed - they can be enabled by returning to the **Command Line Interface** and typing the name of the element desired (in this case, *VAREL.*) You will then enter the module of that tool.

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
varel
\\\  Module=VAREL ELEMENT; Layer=   1        Level=1   Routine:INPUT       ///
<LAYER>(NR)<TOP><BOTTOM><LKUP>(<ON>/<OFF>)[<LRES>/<RRES>]<PUTAT><HELP>
<GIVEN>[<POLY>/<CONSTANT>[<POLY>]]<LEAK>[POLY]<RESISTANCE>[<POLY]<MRES>[POLY]
<APTERMS>(N)<OVERFOLD>(N)<FAR>(FACTOR)<FTERMS>(N)NODETOL>[TOL]
<POLYGON>[/KEY]<THICKNESS>(H)<POROSITY>(n)<HELP><RESET><RETURN>
<DRAW>[<GIVEN>/<RESIS>/<MRES>/<LEAK>/<ON>/<OFF>][<TOP>/<BOT>/<ALL>][<ON>/<OFF>]
```

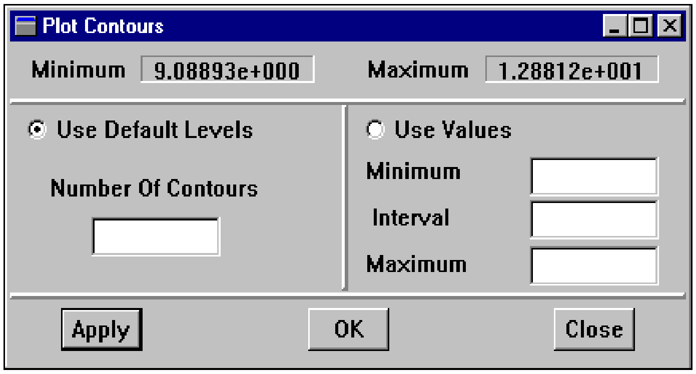
The command <DRAW>[<GIVEN>/<RESIS>/<MRES>/<LEAK>/<ON>/<OFF>] (among other things) controls whether or not that particular element is being drawn on the Results Mode screen. Type DRAW ON to display the VAREL geometry in the Results Mode.

```
draw on
\\\ Module=VAREL ELEMENT; Layer=  1       Level=1   Routine:INPUT      ///
<LAYER>(NR)<TOP><BOTTOM><LKUP>(<ON>/<OFF>)[<LRES>/<RRES>]<PUTAT><HELP>
<GIVEN>[<POLY>/<CONSTANT>[<POLY>]]<LEAK>[POLY]<RESISTANCE>[<POLY]<MRES>[POLY]
<APTERMS>(N)<OVERFOLD>(N)<FAR>(FACTOR)<FTERMS>(N)NODETOL>[TOL]
<POLYGON>[/KEY]<THICKNESS>(H)<POROSITY>(n)<HELP><RESET><RETURN>
<DRAW>[<GIVEN>/<RESIS>/<MRES>/<LEAK>/<ON>/<OFF>][<TOP>/<BOT>/<ALL>][<ON>/<OFF>]
```

Now the pond and the well both appear on the screen and we are ready to generate a plot of our model.

Generating the Plot

Select **Plot** from the **Window** pull-down menu. This will activate the **Plot** window which is used to generate any plot in MLAEM. The **Plot** window consists of three parts: Contour **Maximum and Minimum**, **Use Default Levels** and **Use Values**. The maximum and minimum boxes are simply the maximum and minimum values of the contours. The **Use Default Levels** option uses the minimum and the maximum values to generate a plot. The only control that the user has is over determining how many contours will be generated. However, as is the tendency with generalizations, it is often satisfactory, but allows little specialized viewing. In the cases where a general plot is not sufficient, for example when only a certain interval is of interest, the user may choose the **Use Values** option. It consists of three value boxes: the maximum and minimum values to be plotted and the interval at which the contours will be drawn, starting from the minimum. It is useful to note that it is acceptable to make the minimum and the maximum contours the same and that only one contour will be drawn.



Minimum 9.08893e+000 Maximum 1.28812e+001

● Use Default Levels ○ Use Values

For our example, we will choose the default option, with 50 contours plotted. When the values have been entered, click **OK** or **Apply** and the plot will be generated. (The difference between **OK** and **Apply** is that when **OK** is selected, the window will be deactivated, and when **Apply** is chosen, the window will remain activated. **Close** deactivates the window without generating a plot.)

Once a plot has been generated, it can be changed by creating a new plot (if **Automatically Refresh Screen** in activated, the old plot will be erased and the new will be drawn) or it can be erased by choosing **Reset Screen**.

Differences in Solutions

An important feature in the command line interface closely related to grids is the DIFGRID command from the READ menu. This command can be used to find the differences in head between two plots, which is useful when comparing the solutions of two different models. To use DIFGRID, you must first have saved a grid with the command <SAVE> . The number of grid points and the window used to calculate the grid points must be the same as the current grid and window.

For an example, we will first save our grid to “pondns.grd”, remembering that we calculated 50 grid points. Keeping the window the same, we add a second well and compute another grid with 50 points. Now we can subtract “pondns.grd” from the current grid in order to see the difference that the well makes in the solution. Performing a DIFGRID is done as follows:

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
read
<S>OLUTION,<G>RID,<B>OTH,<D>IFGRID,<R>ETURN
d
PLEASE ENTER FILENAME; <R> TO ABORT
pondns.grd
GRIDFILE HAS BEEN SUBTRACTED FROM CURRENT GRID
```

4.3 OBTAINING RESULTS FROM PLOTS USING THE RESULTS MODE

The crosshair tool (the tool in the upper right-hand corner of the Cursor Mode toolkit) can be used to obtain results during the presentation of plots. Heads, discharges, potentials, lower boundaries, coordinates, and a plot of the discharge normal to any line (NDISCHARGE) can be found by positioning the cursor at the desired point, clicking and entering one of the commands.

In this exercise, we will generate surface information about the aquifer. Click on the crosshair tool and move it to the desired point and click to place a red cross on the plot (the information generated next will be specific to that point.) In order to obtain information about the point, we must be in the command line. Notice that once the cursor is in the command line, the program automatically enters the module RESULTS (The module’s name is TRACE, but may be accessed by either the command *TRACE* or *RESULTS*.) There are several features in the RELULTS Module that are very helpful. Following is the command line that appears when the module is entered.

```
\\\  Module=TRACE; Layer=   1   Level=1    Routine=INPUT            ///
<LAYER><HEAD><POTENTIAL><DISCHARGE><LEAK><ERLEAK><MHEAD><ERHEAD>
<RRES><LRES><BASE><SURF><TRACE>[ELEV]<WGENERATE>(# LINES)[ELEV]SECTION
<NDIS>[ACC,MAX NR POINTS]<ENDDIS>[ACC, MAX NR POINTS]
<MARKER><COMMAND><LKUP>(<ON>/<OFF>)<COOR><RETURN><MENU>
```

The commands LAYER, BASE, HEAD, POTENTIAL, DISCHARGE, LEAK, and TRACE generate information about the specific point at which the cross is located. All the commands work in a similar fashion. As an example we will obtain the level of the phreatic surface by typing the command SURF:

```
surf
UNCONFINED: X,Y, PHREATIC SURFACE -4.398977E+01  1.054800E+00  1.182836E+01
```

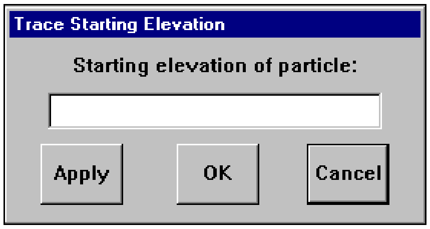
In this bit of data, unconfined refers to the type of flow, in this case unconfined. The next three terms label the three values thereafter, respectively; -4.398977E+01 refers to the **x** coordinate of the point, 1.054800E+00 refers to the y coordinate, and 1.182836E+01 refers to the elevation of the point on the phreatic surface.

After a command has been entered in the command line, the command can be repeated every time the right mouse button is pressed. The program will continue to do this until another command is typed or another tool is selected from the Tools window.

4.4 USING PLOTS TO TRACE PARTICLES

Particle Traces

Particles can be traced using the **tracing tool**. Click on the tracing tool and the elevation dialog will pop up. The value entered in this box is the elevation at which the particle trace will start. In our example, we will make the starting elevation 10 (the head at our reference point). Now move the cursor to a point and click. A particle pathline will be generated. This can be repeated as needed.


Trace Starting Elevation

Starting elevation of particle:

Figure 7 shows a series of particle traces done as directed in this exercise.

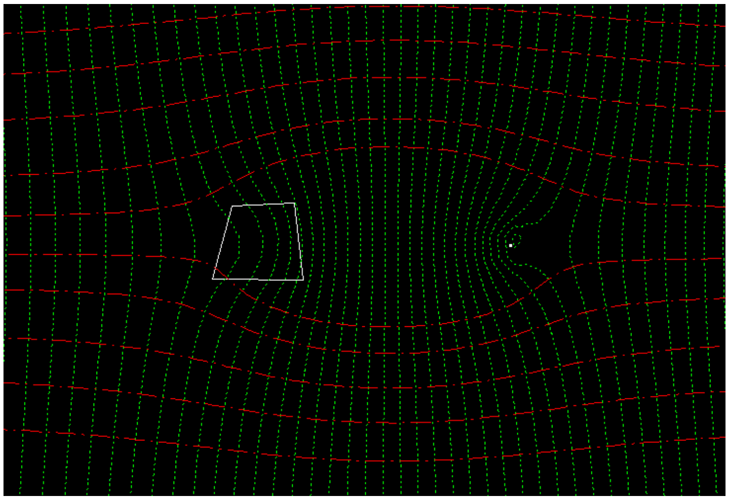


***Figure** 7. Several particle traces*

Well generation traces

The well generation tool (WGEN) generates ten pathlines that all end up at the well that has been clicked upon. Click upon the tool and then click on the well. The following plot will be generated.

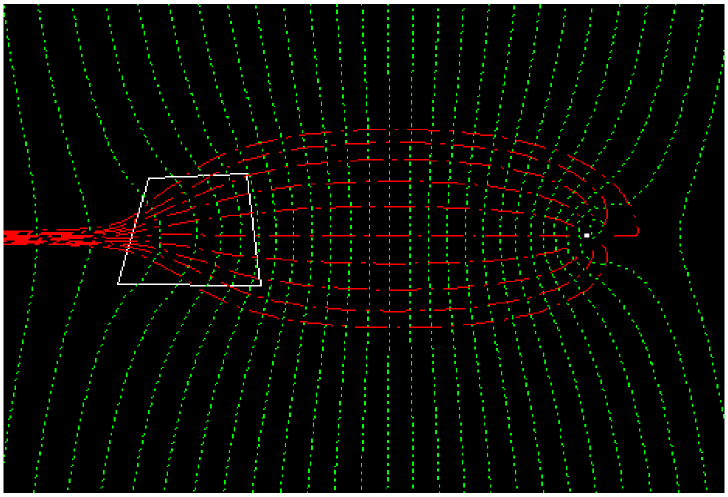


***Figure 8**. The Wgen plot*

Section Traces

A sectional view (side view) of the path of a particle can be obtained by entering the command <SECTION> after having generated either a particle trace or a WGEN. The path of the particle will then be re-colored with a solid line by the program. Press the enter key to proceed to the SECTION module The MIN and MAX values indicate the minimum and maximum elevation of the particle during the trace:

```
section
MIN VAL AND MAX VAL :    .000000E+00   .128867E+02
\\\  Module=TRACE             Level=2    Routine=SECTION INPUT     ///
<LOWBOUN>[<ON>/<OFF>]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]<PLOT>
<HELP><VIEWPORT>(FACTOR)[X1,Y1]<RETURN>
```

The command <VFAC> is used to specify the vertical factor by which the height of the section will be multiplied. You can figure out what a good value for VFAC should be by examining the relationship between the thickness of the aquifer and the length of the trace. In larger models, a much larger VFAC is needed than in smaller models. A good thickness to length ratio to look for is about one, two or three parts in ten. Since the maximum elevation of the particle was about 13 (from above) and the length of the section was about 400 (a little less then the length of the window) our ratio is about 1 in 30, so we will enter a VFAC of 3. To see the sectional graph, type PLOT or move the mouse out of the command line window (be careful not to move the mouse out before typing the desired commands as they will no longer be available. Note that all the tools disappear while in section trace. In order to get the tools back, type "return" to exit the section module.

```
vfac 3
\\\  Module=TRACE             Level=2    Routine=SECTION INPUT     ///
<LOWBOUN>[<ON>/<OFF>]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]<PLOT>
<HELP><VIEWPORT>(FACTOR)[X1,Y1]<RETURN>
plot
```

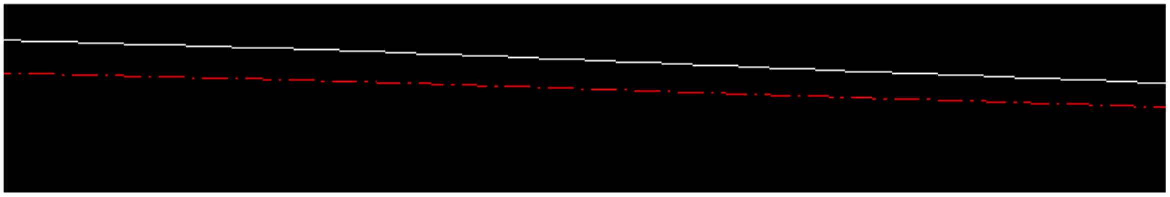


Figure 9. A sectional graph

Start particle traces in this mode by moving the cursor into the window and then using the trace module as before. Type return to exit this module and re-enable the other MLAEM tools.

4.5 VIEWING SECTIONAL PLOTS

A plot of an arbitrary section (from side view) of the aquifer can also be obtained. This is done with the SECTION module in the **Command Line Interface** (no particle traces can be made in this module):

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
section
\\\  Module=SECTION            Level=1    Routine=INPUT            ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
```

You must specify the sectional slice to look at. Since the section looks like a line from above, you must specify its endpoints. Usually a section is taken through the middle of the window, and over most of the length, focusing on the most interesting parts. Since the area to the left of the pond and to the right of the well are of little interest, we will view the area from (-70,0) to (70,0).

```
section -70 0 70 0
\\\  Module=SECTION           Level=1    Routine=INPUT           ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
```

You can choose to see the base and phreatic surface of the aquifer:

```
base on
\\\  Module=SECTION           Level=1    Routine=INPUT           ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
surface on
\\\  Module=SECTION           Level=1    Routine=INPUT           ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
```

Just as you must make a GRID before making a contour plot, so you must compute the points on the surface:

```
compute
MIN VAL AND MAX VAL :   0.000000E+00  0.113000E+02
YOU MAY ENTER WINDOW AND/OR VFAC PRIOR TO PLOTTING
\\\  Module=SECTION           Level=1    Routine=INPUT           ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
```

In this example we will use a VFAC of 5 (Figure 10).

```
vfac 5
\\\  Module=SECTION           Level=1    Routine=INPUT           ///
<SECTION>[RX1,RY1,RX2,RY2]<WINDOW>[RX1,RY1,RX2,RY2]<VFAC>[FAC][<W>]
<LAYER>(NR.START,NR.END)<COMPUTE>(NR. OF POINTS)<HEAD><POTENTIAL><PSI>
<LOWBOUN>(<ON>/<OFF>)<BASE>(<ON>/<OFF>)<TOP>(<ON>/<OFF>)<SURFACE>(<ON>/<OFF>)
<PLOT><VIEWPORT>(FACTOR)[X1,Y1]<HELP><PAGE><RETURN>
plot
```

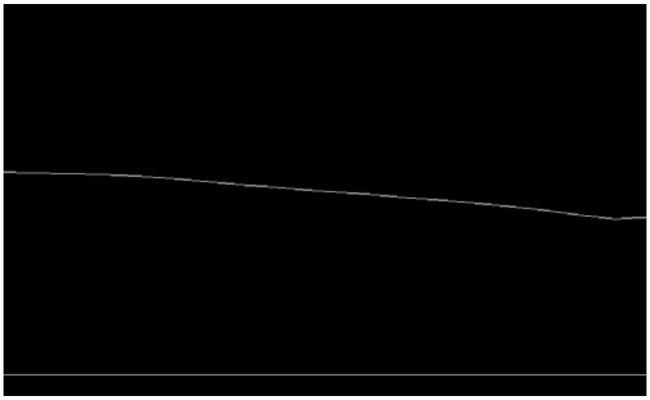


Figure 10. *The section plot.*

SECTION 5 - TUTORIAL TO THE SERVICE MODULES (SWITCH, PSET, MAP)

The service modules are used to control features in the program. The SWITCH module controls input and output. The PSET module controls screen functions, such as the line types on plots. The MAP module allows you to superimpose hand-made figures onto plots.

5.1 THE SWITCH MODULE

The most important function of the SWITCH module is to set the prefix for input files. An input file is a list of program commands written in ASCII, which can be made using the **Save** or **Save Selected** option from the **File** pull-down menu or which can be made by entering commands into a simple text editor (for example *Notepad*). The dialog which is shown when opening (or reading in) a file defaults to the directory set by the INPUT PREFIX. You can check what the INPUT PREFIX is by simply typing *prefix* in the SWITCH module:

```
switch
\\\  ROUTINE SWITCH                                    ///
<INPUT/OUTPUT/MESSAGES/ERROR>(FN)[LU]
<INPUT/OUTPUT/MESSAGES/ERROR ECHO ON/OFF/APPEND>(FN)[LU]
<LOG ON/OFF>[(FN)[LU]]<LU><HELP><RETURN><END>
<CALL>(FILENAME)<BACK>
<PREFIX>[<INPUT>/<OUTPUT>/<READ>/<SAVE>/<HELP>][PREFIX]
prefix
INPUT  PREFIX:  DAT
OUTPUT PREFIX:  DAT
READ   PREFIX:  BIN
SAVE   PREFIX:  BIN
HELP   PREFIX: .
```

If the prefix is incorrect it can be changed by using the <PREFIX> command followed by the directory path (including the drive). For example, to change the input prefix, type:

```
prefix input c:\mlaemw\files
```

Please note that the command lines in the SWITCH module are often difficult to decipher. There is a paragraph in this section called EXPLANTION OF A COMPLEX COMMAND LINE which explains the most difficult one.

To read PONDNS.DAT into MLAEM choose **Open** from the **File** pull-down menu.

The program will proceed to read the file. If the user is reading in a file that was written in a text editor, then it is important that the data is entered correctly. If everything in the file is correct, then it will be read without error messages. If there is a mistake in the file, for example if the command <GIVEN> is not entered before the coordinates and discharge of the well, then an error message will be printed, and the coordinates will not be read in. The program will not stop, but will continue to read the file (sometimes this causes a major problem because the program ends up in the wrong module with the wrong commands -- this problem can be avoided by adding extra RETURN statements). Here is an example of an ASCII file ready to be read in.

```
return
layer 1
aquifer
  base 0
  thick 100
  return
given
  pond
  0 0 10 -.01
  uniflo .1
  return
reference
  100 0 10
  return   * note return is necessary to exit the REFERENCE module.
window -40 -50 80 50
well
   60 0 9.424778 .25
+++ILLEGAL COMMAND IN WELL+++
  return
solve
  10 .

grid 20
  10 ..........
  20 ..........
  30 .
switch
  end
```

The lines above will be shown in the command line interface after reading in a file.

In the above output file, the <RETURN> command at the beginning is necessary because the file is read in SWITCH: MLAEM must move out of SWITCH and into another module. The <SWITCH> command at the end of the file is necessary because the SWITCH module must be accessed in order to end the input sequence with the command <END>. SWITCH then returns control to the keyboard.

It is also possible to read input files with other input files. For example, you may have two input files. One has the window, aquifer data, the reference point, and given elements; it's called aqint.dat. The other one has wells; it's called wellint.dat. One input file can then read in the two input files using the CALL statement. This is called a CALLFILE and may look like this:

```
return
switch
   call aqint.dat
   call wellint.dat
   end
```

Each of the files that are called must contain the command <BACK> to send switch back to the callfile. For example, wellint.dat might look like this:

```
ret
well
  giv
  0 0 .01
ret
swi
  back
```

Call files may call other files, creating **nested files**. Up to five levels of nesting are allowed (see figure 11). Call files are especially convenient when working with large models and data files (see conventions used in NAGROM). Be careful not to click on **Save** after opening a call file - the call file will be overwritten.

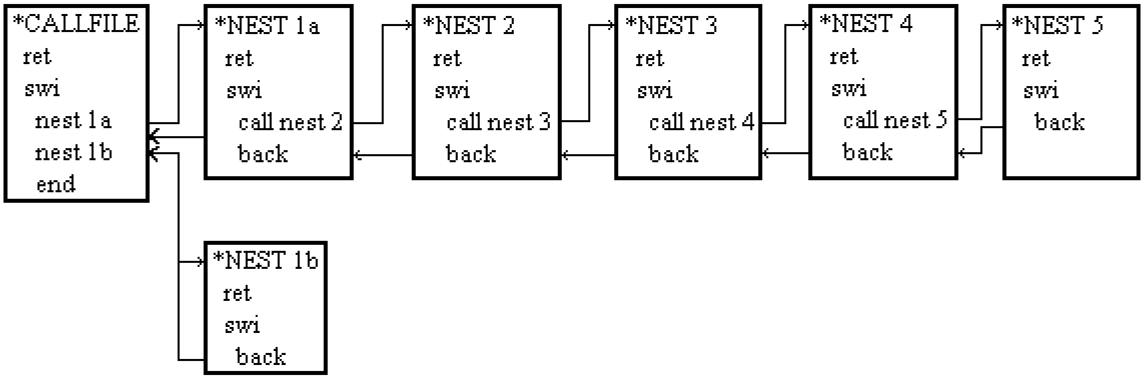


Figure 11. *Nesting files*

5.2 EXPLANATION OF A COMPLEX COMMAND LINE

Input files can be made, once comfortable with the program, directly in a text editor. Before then they can be made using the <INPUT ECHO ON>(Filename) command:

<INPUT/OUTPUT/MESSAGES/ERROR ECHO ON/OFF/APPEND>(FN)[LU]

This is one of the most difficult command lines to understand in the program (the meanings of the words ERROR, ECHO, APPEND become clear in the example). The slashes between the first four commands indicate that a choice of one of the four must be made. The space between ERROR and ECHO indicates a new required list of parameters. Since ECHO is followed immediately by a space, it must be chosen (if echo had been immediately followed by a slash and other command words, another choice could have been made). The space between ECHO and ON indicates yet another list of parameters. These three are again separated by slashes, so a choice must be made. Finally the parameter (FN) (a filename or device which will be read or written to) is required, and an optional parameter [LU] (logical unit) can be chosen. This makes all of the following combinations possible:

input echo on Filename * echoes input typed by the user onto an file. output echo on Filename * echoes output produced by MLAEM onto a file. message echo on Filename * echoes messages (menus) onto a file. Error echo on Filename * echoes error messages onto a file.

The above combinations with the choice of <OFF> (instead of <ON>):

input echo off Filename * turns input echo to file off output echo off Filename * turns output echo off message echo off Filename * turns message echo off error echo off Filename: * turns error message echo off

The above combinations with the choice of <APPEND>:

input echo append Filename * appends (adds) input echo to a file
output echo append Filename * appends output echo to a file
message echo append Filename * appends message echo to a file
error echo append Filename * appends error message echo to a file

The command line above the one we just discussed looks like this:

<INPUT/OUTPUT/MESSAGES/ERROR>[FN][LU]

It also has several possibilities:

input Filename * read input from a file.
output Filename * sends the output to a file.
Message Filename * sends program messages to a file.
Error Filename * sends error messages to a file.

A log (an exact copy) of all the commands typed and all the program's responses (messages and error messages) can also be made using the command <LOG ON/OFF>[(FN)[W]]. If the optional FN is not chosen, as in the following example, the file will be written to LOG.DAT:

```
\\\  ROUTINE SWITCH                                    /// <INPUT/OUTPUT/MESSAGES/ERROR>(FN)[LU]
<INPUT/OUTPUT/MESSAGES/ERROR            ECHO            ON/OFF/APPEND>(FN)[LU]            <LOG
ON/OFF>[(FN)[LU]]<LU><HELP><RETURN><END>
<CALL>(FILENAME)<BACK>
<PREFIX>[<INPUT>/<OUTPUT>/<READ>/<SAVE>/<HELP>][PREFIX]
log on
```

5.3 THE PSET MODULE

The PSET (PLOT SET) module controls screen functions, such as the line types on plots. The following is the PSET command line:

```
pset
\\\  ROUTINE SET PLOT MODE                         ///
<COLOR><BW><SINGLE><TEXT>(<ON>/<OFF>)[N]<DUAL><PRINTER><PLOTTER><SCREEN>
<PALETTE>(CODE)<PMHI>(CODE,TYPE)<PMTYPE>(CODE,TYPE)<HELP><PAGE>(<ON>/<OFF>)
<LONG>(<ON>/<OFF>)<META><RETURN><LDTYPE>(CODE,TYPE)<LCTYPE>(CODE,TYPE)
<MANCOLOR>(<ON>/<OFF>)
```

The following are some of the things that can be done using the PSET module. The line types and color types can be changed using the <LDTYPE> <LCTYPE> commands. The color palette can be changed with the <PALETTE> command.

Since the purpose of this tutorial is to give an introduction the program, no details or examples of these commands are given. The user is referred to the online help files. Type HELP in any module in order to access the help for that module.

5.4 THE INITAEM.DAT FILE

Parameters in the PSET and SWITCH modules may be set in the INITAEM.DAT file. INITAEM.DAT is read automatically by the program when it is loaded. This allows input prefixes and screen conditions to be set before using the program. A typical INITAEM.DAT is the following:

```
ret
swi
  prefix input dat
  prefix output dat
  prefix read bin
  prefix save bin
  prefix help .
  ret
pset
* comments head   layout trace
*  pal 1  * blue   white  read
*  pal 2  * green  yellow dark-yellow
*  pal 3  * teal   gray   magneta
*  pal 4  * yellow white  magenta
  ret
logsize 2000000    * set the maximum text log size to a high number for computers with a large amount of memory
swi
  end
```

The modules SWITCH, PSET, and PLOT are the three modules that you will find reappearing in other modules. They are so useful that it is convenient to have them at hand in modules such as TRACE. SWITCH reappears in TRACE and MAP. PSET reappears in TRACE. PLOT reappears in CURSOR, TRACE, and SECTION.

5.5 THE MAP MODULE

The module MAP allows you to include complementary figures in plots. The figures are composed of curves (broken up in straight segments) and/or points.

To make a curve, enter the coordinates of points, and MAP will connect them. A new curve can be started by entering <CURVE> again. The following sequence makes a line and a point:

```
map
\\\  Module=MAP          Level=1   Routine=INPUT          ///
<CURVE><POINT><WINDOW>(X1,Y1,X2,Y2)<PLOT>(ON/OFF)<HELP><SWITCH><MAXIMUM>
<MAPWINDOW><READ><SAVE><RESET><RETURN>
curve
\\\  Module=MAP          Level=1   Routine=INPUT CURVE    ///
(X,Y)
-10 -10
\\\  Module=MAP          Level=1   Routine=INPUT CURVE    ///
(X,Y)
10 10
\\\  Module=MAP          Level=1   Routine=INPUT CURVE    ///
(X,Y)
point
\\\  Module=MAP          Level=1   Routine=INPUT POINTS   ///
(X,Y)
-1 1
\\\  Module=MAP          Level=1   Routine=INPUT POINTS   ///
(X,Y)
comman
```

The map can be added to layouts and plots by typing the PLOT ON command:

```
\\\  Module=MAP          Level=1   Routine=INPUT          ///
<CURVE><POINT><WINDOW>(X1,Y1,X2,Y2)<PLOT>(ON/OFF)<HELP><SWITCH><MAXIMUM>
<MAPWINDOW><READ><SAVE><RESET><RETURN>
plot on
\\\  Module=MAP          Level=1   Routine=INPUT          ///
<CURVE><POINT><WINDOW>(X1,Y1,X2,Y2)<PLOT>(ON/OFF)<HELP><SWITCH><MAXIMUM>
<MAPWINDOW><READ><SAVE><RESET><RETURN>
ret
```

Maps can be saved by entering the <SAVE> command, and can be read in again using the <READ> command. For most efficient storage in the binary map files it is recommended to group curves together and points together.

INTRODUCTION TO PART TWO

The following sections are a more detailed description of the different parts of MLAEM. First the analytic element tools are explained in detail. Following these modules is a section on how to model an aquifer with multiple layers. Following this are sections containing comments for the Result-Viewing modules, the Service modules, and the Stand-Alone commands.

The permeabilities, resistances and discharges chosen in the following sections were chosen to show the greatest effect possible of each element on the model and were not chosen to be realistic. For example, the permeability .01 m/day is a very low permeability in a normal aquifer.

Models are made by the superposition of the analytic elements. However, not all elements can be superimposed. Control points may of course never coincide, but some elements may not be modeled in certain ways with other elements.

The modules **CHECK,** and **TRACE** can be used extensively to view the results of the model developed. In general, it is important to check the reference head control point with the CHECK module or **edit** tool (by moving the cursor into the aquifer and clicking) to assure the accuracy of the solution. However, only a quick word on what to check and which commands to use will be given in the following sections. These modules can best be understood by using them and referring to the on-line help files.

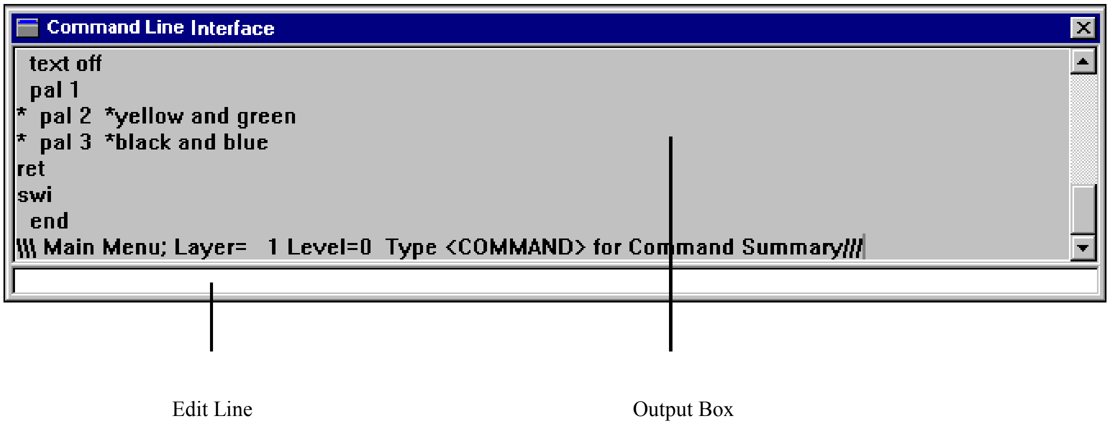
SECTION 6 - TUTORIAL TO THE COMMAND LINE INTERFACE

MLAEM was originally based solely on a command line interface, very similar to the one currently in MLAEM. All the tasks in the program were executed from the command line, and, as now, it was so structured to best accommodate the analytic element method.

The structure is modular, with a module for each element, and modules for each type of task to build and use the model. Each aquifer, or layer, is intended to be entered separately. The layers are then connected using leakage elements. The layer number (one for the top layer, two for the next one down, etc.) is set before entering the elements pertaining to that layer. When elements are entered, each one is added to the layer that was specified. The layer number is then changed to enter elements in other layers. Separate modules (menus), are used to enter each different analytic element and to view results.

The Window

The window is composed of two parts: a part used for displaying output from the program (the upper part of the window) and a part used for typing commands (the long rectangular box, or **edit line** at the bottom of the window). Before typing any commands in the command line, the mouse must be moved down to and clicked in the single edit line at the bottom of the window. As soon as the mouse is moved out of the edit line, the **Command Line Interface** will stop accepting commands from the keyboard.



```
 text off
 pal 1
*  pal 2  *yellow and green
*  pal 3  *black and blue
ret
swi
 end
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
```

If more of the output from the program needs to be seen (i.e. for viewing a help file), the window may be enlarged by moving the cursor to the top border of the window (the cursor will turn into a double-arrow), clicking down and dragging the window up until it is of appropriate size. Scrolling is also possible by moving the scroll bar at the far right of the window.

6.1 THE MODULAR STRUCTURE

The program has four different types of modules: those that are used to enter the analytic elements (analytic element modules), those that support the viewing of results result-viewing modules, those that control special features of the program (service modules), and those that perform only a single task and are made up of one command (stand-alone commands). The first three types of modules are introduced in the following sections. The stand-alone commands, however, are introduced as they are needed in each of the sections. Most of the modules (menus) can be accessed by the main module (main command menu):

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
```

Type COMMAND to see a list of all the commands in the main module:

<WELL><POLYGON><ATARD><VAREL><AQUIFER><DOUBLET><STRING><CUREL>
<REFERENCE><GIVEN><AREL><LINESINK><DROOT><CHECK><TRACE>
<RESULTS><SECTION><GRID>[NR.GRID PTS]<SWITCH>[FILENAME]<PSET>
<MAP><READ><SAVE><HELP><LAYER>(NR)<LKUP>(<ON>/<OFF>)
<WINDOW>[X1,Y1,X2,Y2]<SOLVE><ITER>[ACCURACY,MAX.NR.ITER]
<ZSOLVE>[ACCURACY,MAX.NR.ITER]<COINCIDE>[DIST.][<CHECK>]
<DELETELOG><LOGSIZE>(NR. CHARACTERS)<FONT>
<LAYOUT><PAGE><TITLE>[PROJ. NAME]<RESET><STOP>

The structure of MLAEM is thus a division of the program into smaller menus called modules. A complete tree of the structure of these modules is shown in Figure 12.

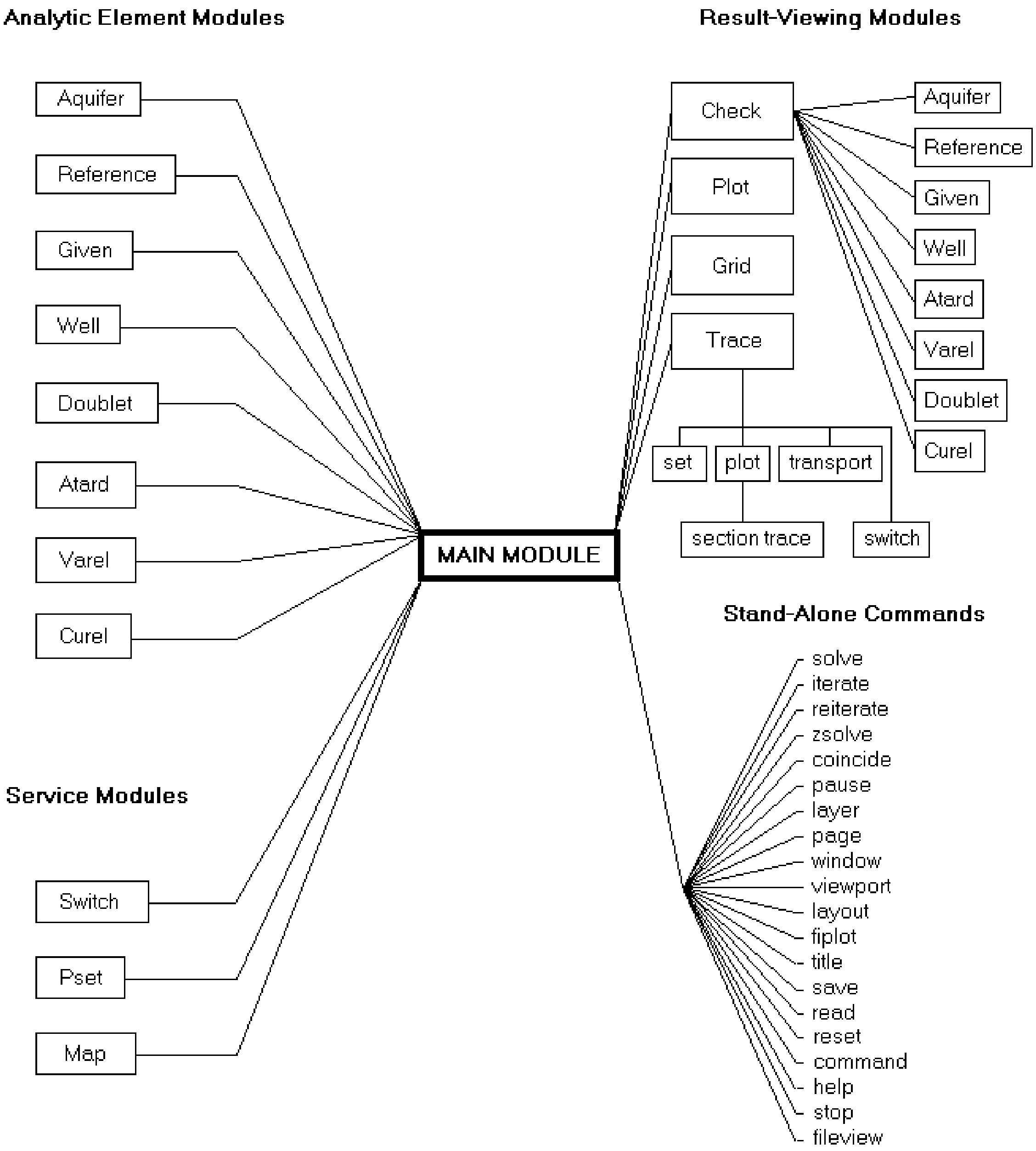


***Figure 12**. The Modular Structure of MLAEM.*

6.2 USING A MODULE

Most of the choices in the main command list will bring you one “menu" level lower in the program, into a module. For example, if you want to enter a given analytic element, type <GIVEN> (the brackets < > signify that the word enclosed is a command word. Command words will always be surrounded by these brackets in the text and in the program. However, you should only type the word inside: in this case GIVEN.

```
given
\\\  Module=GIVEN; Layer=   1   Level=1    Routine=INPUT            ///
<LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET>
<RETURN>
```

The words in the top line of program output mean the following: Module indicates the current module. Layer indicates which layer the element that follows will be entered in (in this case layer one [the top aquifer]). Level indicates how many times you must type <RETURN> before you return to the main command module, and Routine indicates which process you are in (input, type of input, check). The command <RESET> resets all parameters in the module to their default values (including those in other layers). (In the main module <RESET> resets every input parameter in MLAEM). The command <HELP> will display the help file for the current module. To enter a pond into the model(without using VARELs) use the command <POND>:

```
pond
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
```

The set of parenthesis surrounding XC,YC,RAD and EXTRACTION RATE, or a set of parentheses enclosing parameters after a command word indicate a required parameter. Box brackets [ ] indicate an optional parameter, and in a case such as [[LABEL]] the first set indicates that [LABEL] is an optional parameter, and the second set indicates that a label must be surrounded by brackets (to set it apart from a command.). Labeling allows you to recognize certain analytic element types when checking the output. To enter the pond with its center (XC,YC) at the origin, a radius of 10 and an extraction rate of -.01 enter:

```
0 0 10 -.01
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
```

Some command lines in the program are very difficult to understand. For an explanation of one of these the user is referred to the section about the service modules, where a complex command line in the SWITCH module is explained.

6.3 FREE FORMAT DATA ENTRY

There are also other ways to enter the data above, and commands in general. Command words such as <GIVEN> can be abbreviated, as long as the command remains unique: <GIVEN> can be abbreviated to <GIV>. MLAEM also supports Free Format data entry (a program written by P.A. Cundall). The program scans all data entered on the input line, and separates it in command words and data to be interpreted as integer or real numbers. The following symbols are defined as separators in the input line:

blanks, slashes (both forward / and backward \), and parentheses

These separators may be used in any combination to separate the data. You also do not need to differentiate between integers and real numbers (The number 200 will be interpreted as an integer if an integer is required, and as 200.0 if a real number is required).
An asterisk [*] may be used to add a comment (the program does not read beyond a asterisk), anywhere in the input line. The entry above can thus be entered in many different ways. A few options as an example:

```
CORRECT:
0 0 10 -.01     *option #1
(0 0) 10 -.01  *option #1
* option #3: make entire line a comment
INCORRECT:
0 0
10 -.01    *option #4            -- cannot split entry line
0 0 * option #5  10 -.01      -- 10 will not be read because of asterisk
```

If mistakes are made in data entry an error message will occur. The program will, however, continue to read the next entry. It will not wait for a response to the error message, so you may ignore the message (this is so that data files can be read in their entirety).

6.4 ENTERING DATA IN A MODULE

Returning to our example, after this command line is displayed (in the GIVEN module, routine GIVEN POND):

```
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
```

You have the option of entering the parameters (XC,YC,RAD,EXTRACTION RATE). If you choose this option (as we did), the program will again come up with a prompt for a pond:

```
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
```

Because the program remains in the same routine and does not return to the main menu, you do not need to enter <GIVEN> and then <POND> again if you wish to enter more than one pond. You can just repeat the parameters (XC,YC,RAD,EXTRACTION RATE) over again for each pond. Every module in the program works the same way. Once you are in a routine, you will remain there unless you do one of four possible things:

First, any command of the main module corresponding to the routine that you are in (in this case GIVEN POND) will bring you to its corresponding routine. In the case of the example, these commands are <LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET> and <RETURN>.

```
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
0 0 10 -.01
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
rain .005 0 0 5 2 45
\\\  Module=GIVEN; Layer=   1   Level=1    Routine=INPUT            ///
<LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET>
<RETURN>
```

In the case that <COMMAND> is one of the options in the main module of the routine you are in, you may use that to return to the main module of that routine.

In order to move one level higher in the program, use the <RETURN> command:

```
\\\  Module=GIVEN              Level=1    Routine=GIVEN POND        ///
(XC,YC,RAD,EXTRACTION RATE)
return
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
```

To return to the main program module either type <RETURN> however many times are need to return to level 0, or move the mouse from the input window, move it back and click once.

The command <STOP> will stop the program.

Help files of all the modules may be accessed via the command line by entering the module that requires help, and typing *HELP*. The file will be read into the **Command Line Interface** and may be viewed in the output section of the window.

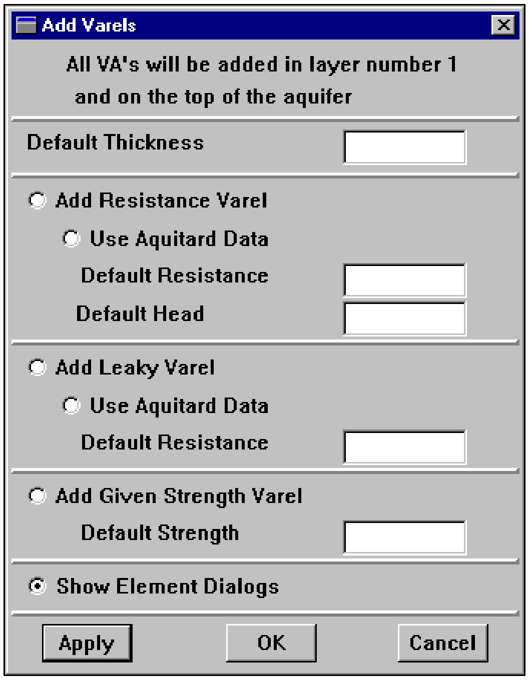
SECTION 7 - VARIABLE STRENGTH AREA ELEMENTS

7.1 NORMAL DATA ENTRY AND USE

Variable **S**trength **A**rea **E**lements (VARELs) can be used to model areal infiltration, such as from rainfall or irrigation. They can also be used for simulating wide rivers or lakes that are not in direct contact with the aquifer, so that infiltration occurs through the bottom. They have been labeled as variable strength because several points of different strengths may be added into one polygon; the program then interpolates between the given points, calculating values for the entire VAREL, which will be of varying strength.

VARELs are bounded by a polygon of any shape. First add the polygon and then enter in the VAREL points accordingly. A single polygon must be highlighted before any VAREL points may be added.

There are three types of area elements, GIVEN, RESISTANCE, and LEAKY (leaky elements are used for modeling flow between layers and will be explained in **MODELING A MULTI AQUIFER SYSTEM)**.



Add Varels

All VA's will be added in layer number 1
and on the top of the aquifer

Default Thickness

○ Add Resistance Varel

○ Use Aquitard Data

Default Resistance

Default Head

○ Add Leaky Varel

○ Use Aquitard Data

Default Resistance

○ Add Given Strength Varel

Default Strength

● Show Element Dialogs

Given Strength VARELs have a given extraction rate, and are used to model a given leakage into or out of the aquifer, (when modeled on top of the aquifer), or model leakage into or out of another aquifer below the one being modeled (when modeled on the bottom of the aquifer.) Whether the element is entered into the top or the bottom is determined by clicking on the boundary in the Layer Box before entering the element.

IMPORTANT: in both SLAEM and MLAEM, VARELs on the bottom of the aquifer have negative strength for extraction, and positive strength for recharge - the opposite of elements on the top, and the opposite of other elements like CURELS and wells. This is due to a sign convention explained in **MODELING A MULTI AQUIFER SYSTEM.**

Resistance VARELs can be used to model a lake or river bottom by specifying the head of the lake (**Default Head** and the resistance (**Default Resistance**) of the bottom. Remember that when the VARELs are added to the model a dialog will pop up (unless the **Show Elements Dialog** option has been deactivated) and the default values may be modified for that particular point at that time.

There is also the option to use Aquitard Data. Aquitard Data will be explained in the next section.

The optional parameter **Thickness** sets the thickness of the lake or river bottom (this thickness will override the default thickness in the aquifer). A VAREL is shown in Figure 13:

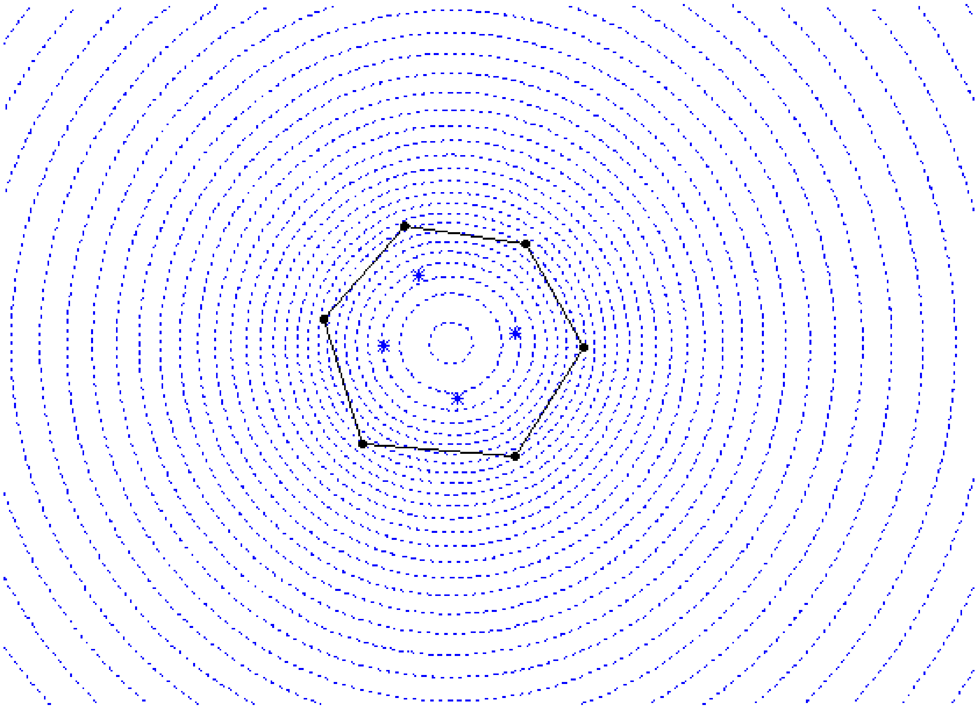


Figure 13*. A VAREL in an aquifer with four data points*.

The leakage of the **Resistance** element is computed as the difference between the given head above a lake bottom and the piezometric head in the area below, divided by the resistance of the bottom.

It is usually adequate to assume that the river bottom coincides with the upper boundary of the saturated zone. The piezometric head in the aquifer may not fall below the elevation of the bottom of the lake because this will cause an incorrect computation of the leakage. In this case the **Resistance** element must be changed to a **Given** element.

If the aquifer below the lake or river bottom is unconfined then the boundary condition becomes non-linear and the program must solve the problem iteratively. More than one step (iteration) will be needed to solve the problem.

Set the maximum number of accuracy and the maximum number iterations in the **solve window**. After clicking on **Apply** or **Ok**, the following message is displayed in the **command line interface** (after choosing an accuracy of .005 and a maximum of 3 iterations):

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
iter .005 3
ITERATION    1
SOLVING    7 EQUATIONS
  10 .......
REQUIRED ACCURACY,(LEAKAGE-COMP.LEAKAGE)/AVG.LEAKAGE  0.50000E-02  0.33691E+00
ITERATION    2
SOLVING    7 EQUATIONS
  10 .......
REQUIRED ACCURACY,(LEAKAGE-COMP.LEAKAGE)/AVG.LEAKAGE  0.50000E-02  0.23014E-01
ITERATION    3
SOLVING    7 EQUATIONS
  10 .......
REQUIRED ACCURACY,(LEAKAGE-COMP.LEAKAGE)/AVG.LEAKAGE  0.50000E-02  0.18303E-03
```

After the specified number of iterations, the program returns the given accuracy and the difference between the leakage given by the user (given head - piezometric head)/resistance and the computed leakage, divided by the average of all the leakage. Of course, the smaller the difference, the better the result. If the error in leakage does not improve with added iterations, the problem may be the result of an incorrect value in one or several of the elements.

The accuracy of the result can be further observed by checking all the control points in the CHECK module. If the accuracy of the result is not as good as desired, then the command <ITER> can be issued again.

Boundaries of VARELs are chosen where the leakage jumps, i.e., across river banks, or across boundaries where the resistance jumps; boundaries should also be chosen where the VARELs cross inhomogeneities, since the leakage sharply changes when the aquifer properties jump. All VARELs may be superimposed, so that the desired jumps may be generated simply by adding a new VAREL on top of the old one with its boundary where the jumps in leakage are desired. In order to have the program compute the proper jump, place several pairs of control points opposite one another on either side on the boundary. Associate one control point of each pair to one VAREL, and the other to the other one. VARELs may have sides in common, but it is not necessary. Spaces between elements do not cause problems.

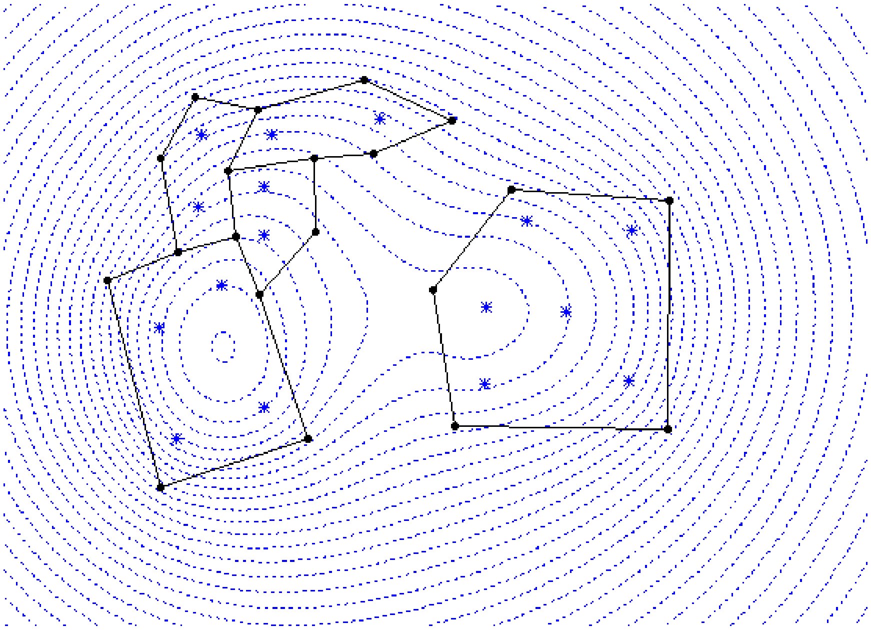


***Figure 14**. A Variable **S**trength **A**rea **E**lement mesh*

7.2 CONSTANT STRENGTH AREA-SINK (AREL) COMPATABLITY

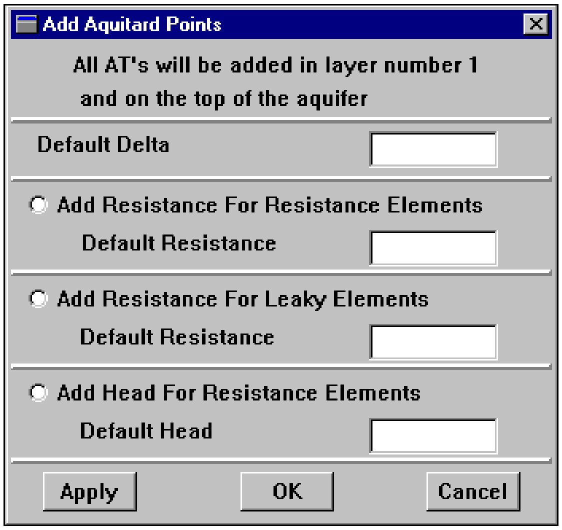
The format of the four-sided constant strength area-elements (ARELs) used in previous versions of MLAEM is supported in the Graphical User Interface. Old data sets containing these elements can still be read into the program. When they are read in, the elements are assigned a four-sided polygon with the key "AREL" and are given a single VAREL data point. If the user changes the key of the polygon to something other than "AREL" then the program will consider the element as a VAREL.

In order to create an AREL, create a polygon with the key "AREL" and add a VAREL data point. Please note that these special polygons can only contain one data point and that a new polygon with the key "AREL" will have to be created for every AREL element. Aquitard data cannot be used with AREL elements. When the data point is added, it will move itself automatically to the center of the polygon. The data points cannot be moved from this location other than by adjustment of the polygon. This is in order to maintain compatibility with previous versions of the programs.

IMPORTANT NOTES: Unlike the VAREL elements, ARELs cannot be superimposed on each other or on VAREL elements. If this is done, the solution will be incorrect, so please use caution when adding area elements. Also note that the AREL elements are calculated with single precision accuracy (as opposed to the VAREL elements, which use double precision) and are more sensitive to the numerical issues discussed in the appendix.

SECTION 8 - AQUITARD DATA POINTS

Aquitard points can be used to enter resistance and given heads as continuous functions inside polygons. They are points where data about the aquifer are known. For example, if the resistance of the bottom of a lake is known in several places, aquitard control points may be placed at those points. Those points are then used by the program to create an interpolation for the properties of the lake bottom. Thus, the resistance of the entire bottom of the lake will be represented as a functions and values can be computed at any point.



Add Aquitard Points

All AT's will be added in layer number 1
and on the top of the aquifer

Default Delta

○ Add Resistance For Resistance Elements

Default Resistance

○ Add Resistance For Leaky Elements

Default Resistance

○ Add Head For Resistance Elements

Default Head

The same principle applies to both the resistance of resistance elements and leaky elements. However, since resistance elements refer to such features as lake and ponds, the head of the feature must be known and entered into the polygon. If only one head point is added, it's location within the polygon has no effect on the solution as the head will be constant throughout. If more than one point is added, then the program will interpolate the surface.

Aquitard points are added into the model using the **ATARD** tool (3,1). A single polygon must first be selected before adding any points. The points may be added outside the selected polygon if so desired. Experience dictates that the best interpolations are achieved with a delta value near or equal to zero.

The **ATARD** moving tool is used to move aquitard points that have already been added to the aquifer. Click on the tool and move the cursor into the **Input Window**. Click on the data point to be moved and, without releasing the button, move the data point to the new location and release the button.

As with all the elements, ATARD points may be added into MLAEM using the command line. In order to enter the aquitard module, type *ATARD*. The name ATARD was given to the "aquitard" module because the program scans for the first four characters, which are the same in the case of AQUIfer and AQUItard.

This is the aquitard command line:

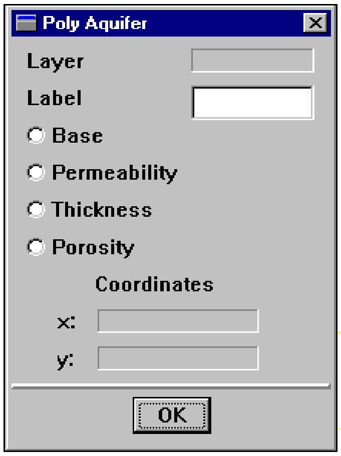
```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///

atard
\\\  Module=AQUITARD ; Layer=   1  Level=    0      Routine:INPUT      ///
<RRESISTIVITY><LRESISTIVITY><FHEAD><POLYGON>[/KEY]<DELTA>(D)<HELP><RETURN>
<LAYER>(NR)<TOP><BOTTOM><LKUP>(<ON>/<OFF>)<RESET>
<DRAW>[<RRES>/<LRES>/<MRES>/<FHEAD>/<ON>/<OFF>][<TOP>/<BOT>/<ALL>][<ON>/<OFF>]
```

To access the aquitard help file, enter the module ATARD and type *HELP*.

SECTION 9 - INHOMOGENEITIES (POLY AQUIFER POINTS)

Inhomogeneities are areas where the aquifer properties differ from the rest of the aquifer. The inhomogeneities may pertain to permeability, thickness, base height, or porosity. They are all bounded by a polygon located within the aquifer. Most inhomogeneities require the use of the **doublet tool** (a mathematical function whose use will be explained in the next section) to create accurate results. However, inhomogeneities in porosity do not require DOUBLETs on their boundaries.

To enter an inhomogeneity into the model, a polygon must first be created. Once that has been done, click on the **poly aquifer** tool and then click within the highlighted polygon. The location of the point within the polygon has no effect on the solution as the inhomogeneity is applied throughout the polygon. The **Poly Aquifer** window will then pop up. Here the type of inhomogeneity to be entered is specified.



There are four options as far as inhomogeneities are concerned: thickness, base elevation, porosity, and permeability. Click on the desired option and a value box will appear. Enter the appropriate value and accept the parameters by clicking **OK.**

When making a mesh of inhomogeneities, the different domains should share common boundaries. (This is not a requirement in the program; inhomogeneities may be intended to be far apart. However, you can determine from a plot whether elements that are not linked should be; peaks will be visible in the contours).

Inhomogeneities are never allowed to cross. Such a dualism in the aquifer parameters will cause unpredictable results. They are, however, allowed to be nested.

The model may now be solved. However, the boundary between the inhomogeneity and the rest of the aquifer will have a “jump”, where the aquifer properties change abruptly. To remedy this problem, the user should add “doublet” elements with the **doublet tool**.

We will create an inhomogeneity in the next section, after having discussed DOUBLETs, and we will compare an inhomogeneity without DOUBLETs to one with DOUBLETs.

SECTION 10 - DOUBLET ELEMENTS

10.1 DATA ENTRY AND USE

Doublets are used to model the effect of the boundaries of inhomogeneities entered with the POLY AQUIFER tool. As was stated in the previous section, inhomogeneities may pertain to permeability, thickness, base height, or porosity. They are all bounded by a polygon located within the aquifer. Also recall that inhomogeneities in porosity do not require DOUBLETs on their boundaries.

Once an inhomogeneity has been added to the model it may be solved. However, the boundary between the inhomogeneity and the rest of the aquifer will have a "jump", where the aquifer properties change abruptly. The DOUBLET tool is used to remedy this problem.

The DOUBLET is a mathematical function that uses overspecification to "smooth" out the boundary between the inhomogeneity and the rest of the aquifer. Click on the **DOUBLET** tool and move to the polygon. The DOUBLET is added to the perimeter of the inhomogeneity. The DOUBLET tool has a tolerance similar to the polygon tool in that it will attach itself to the nearest side of the polygon. Once the elements have been added to the entire perimeter of the inhomogeneity, its overspecification can be checked by clicking on the **edit tool** and clicking on each of the sides of the DOUBLET. The **Polynomial Order** window for that side of the function will then pop up. Here the degree of accuracy of the DOUBLET may be specified. The higher the order and overspecification of the polynomial, the smoother the boundary will be between the inhomogeneity and the rest of the aquifer. The default order and degree of overspecification which is associated with the doublet element can be specified in the polynomial attributes window and is saved along with the window configuration. The concept of overspecification for Analytic Elements was recently introduced by Jankovic and Barnes.

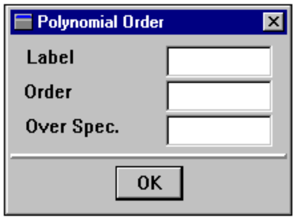


Figure 15a is a plot of the inhomogeneity without DOUBLETs and figure 15b is a plot of an inhomogeneity with DOUBLETs. (Both have the same aquifer properties and grid.)

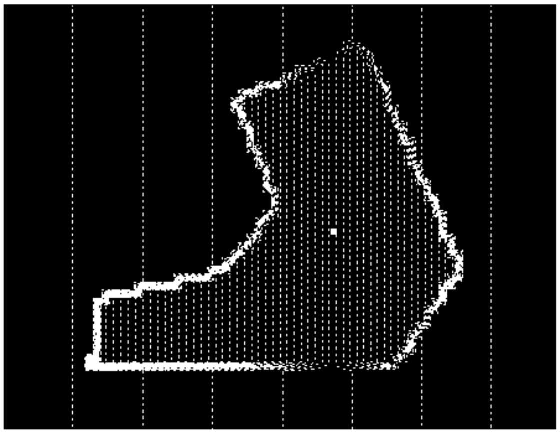


***Figure15a**. An inhomogeneity without DOUBLETs*

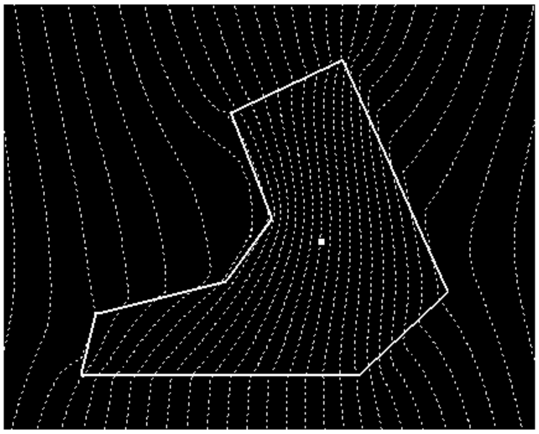


***Figure 15b.** DOUBLETs applied to the inhomogeneity.*

Control points can be checked in the CHECK - DOUBLET module or by clicking on them with the edit tool. Note that when overspecification is used, the values listed as head in and head out will not be exactly identical because the values are optimized in a least squares sense.

We will now create an inhomogeneity with a DOUBLET in an uniform flow field. First change the aquifer parameters for the global aquifer. Set the thickness to 100 and set the reference head to be 10 at (100,0). Then move into the command-line and enter a uniform flow field of 0.01 (recall that this is done in the <GIVEN> module.) Now create a polygon of any shape, and highlight it. Click on the **add inhomogeneity** tool and move it into the polygon. Click it anywhere (remember that the location of the point has no effect on the solution) and the **Poly Aquifer** window will pop up. Choose the appropriate option (in this case we will have a low permeability of 0.01) and accept the parameters.

Now select the **DOUBLET** tool and click on the first node of the polygon and move the cursor to the next node. A solid black line will be drawn. This line represents the DOUBLET element. Click again to anchor it. Repeat as needed to complete the DOUBLET.

Now solve and plot.

To obtain additional help on DOUBLETs or the **Poly Aquifer** Tool, enter the DOUBLET module in the command line (by typing *DOUBLET*) and type *HELP*.

10.2 SOLUTION PROBLEMS

Complex models including doublets will sometimes give the error message “**singular system of equations**” or “**ill-conditioned matrix**” when they are solved. Often times these problems can be resolved by increasing the order of the doublet polynomials (this can be done by clicking on a doublet with the edit tool). The orders of doublets near wells and other aquifer features should be set higher than the orders of doublets not near any aquifer features.

Other times obtaining a solution requires a reorganization of the doublet elements. The best solutions are obtained by adding doublets head-to-tail in a counter-clockwise direction around the inhomogeneity polygons.

SECTION 11 - CURVILINEAR ELEMENTS

11.1 CURVILINEAR GEOMETRY

Curvilinear elements (CURELs) can be used to represent aquifer features that are narrow and whose width is negligible, like a leaky wall or a stream.

CURELs are similar to VARELS in that their geometry must be added to the model before any data pertaining to them may be entered. Please note that the CUREL geometry is the most difficult feature to add into the model properly, and some practice will be required in order to do this correctly.

CURELs can be entered alone, in strings, or in closed strings connected at both ends.

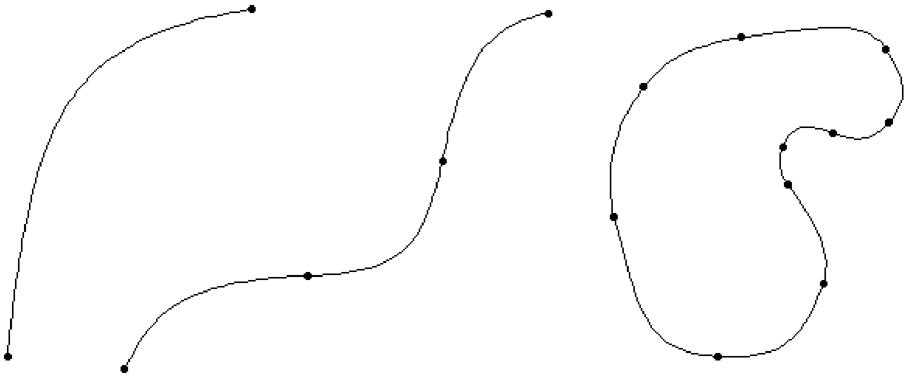


Figure 16. Three types of CURELs: a single segment, a string, and a closed string.

MLAEM contains three tools to create curvilinear strings - the **smooth curvilinear tool**, the **discontinuous curvilinear tool**, and the **add separate string tool**.

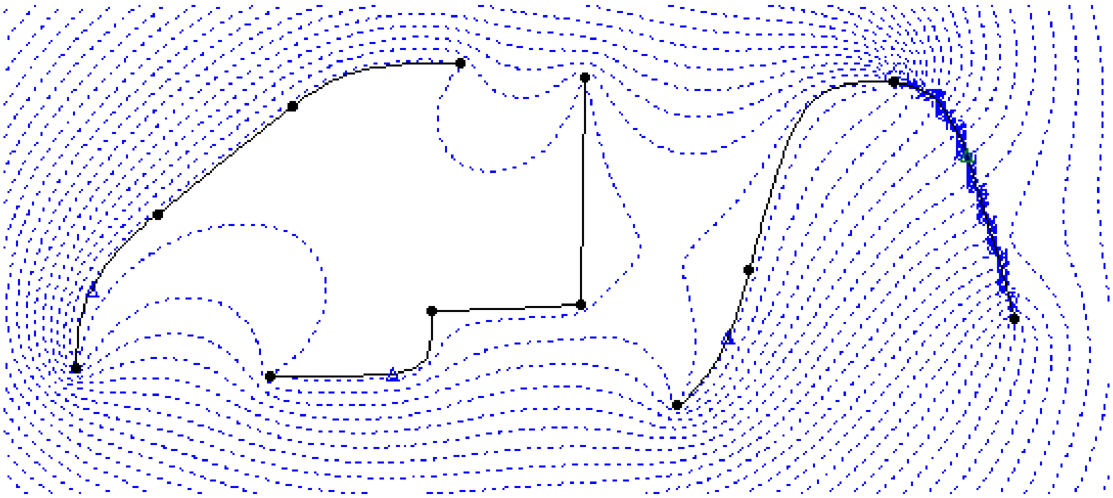


Figure 17. Three string types: smooth, discontinuous and separate.

Creating and Editing Smooth Curvilinear Strings

To create a smooth CUREL click on the **add smooth string tool** and move the cursor into the **Input Window**. Click once with the mouse button to attach the first node of the CUREL to the aquifer. Notice that as you move the mouse around, the line remains straight, pivoting about the first node. Click again at a location away from the first node to start the curve. (Do not worry about the nature of the curve at first, because it can be edited later using the **move smooth string tool**). The curve will orient itself about the spot where you just clicked. Now move the mouse around to get a feel for how it moves. Move the endpoint to the desired location and click again to anchor it. A dialog prompting you for a string "key" will pop up. This is the name, or key, that the program uses to identify a string of curvilinear segments.

The first segment of the CUREL has been added. To modify the curve, click on the **move smooth string tool** and move the cursor to the segment that needs to be modified. Click on either one of the nodes or on the "cross" in the middle of the segment and it will become red as long as the button is held down. Holding down the button, move the cursor around to get an idea of how the segment moves. By grabbing the "cross" at the center, the segment can also be made straight - it will "snap" to a straight line once the curve is straight within the tolerance "STOL" which can be set in the STRING module in the command line interface. The default value for STOL is 0.0001.

To add segments to the string, use the **add smooth string tool** and start a new segment on one of the nodes of the first segment and finish the second segment as desired. The two segments will belong to the same string. Two strings can be joined (provided that they are of the same type, as discussed later) by either inserting a segment between two strings or moving the end node of one string onto the end node of another string. When strings are merged, a dialog will pop up asking for a new key for the merged string. Strings can be closed by creating a series of joined segments and ending the last one on the node of the first segment.

You can view the key of a string and see which segments it includes by clicking on one of the nodes of the string with the **edit tool**.

Note that it is often more convenient to make smooth strings by first using the **add discontinuous string tool** and then editing the discontinuous string with the move **smooth string tool** in order to make it smooth. This is more convenient because it is easier to achieve a desired shape with the discontinuous tools.

Creating and Editing Discontinuous Curvilinear Strings

Discontinuous Curvilinear Strings are added and edited in much the same way as smooth curvilinear strings, except that you must use the add discontinuous and move discontinuous string tools. The difference is that the discontinuous tools allow the strings to have sharp bends at the nodes.

Segments can be added to strings and strings can be merged and closed in the same fashion as with the smooth string tools.

Creating and Editing Separate Curvilinear Strings

The **add separate string tool** creates a separate string at the start of each new segment. This is useful when connecting two strings with different types. This can be seen in the third section of Figure 17, which has a head specified string connected to an impermeable wall string.

Separating Strings

The move separate string tool is used to separate a string into two parts and to move already separate strings. High-light the segment which you wish to move off a string and grab the node. Once the node is moved a separate string will be created. Strings can be connected again by using the **move smooth string tool** and the **move discontinuous string tool**.

Changing Existing Strings from Smooth to Discontinuous and Back Again

Smooth curvilinear strings can be made discontinuous by adjusting the string with the move **discontinuous string tool**.

Discontinuous strings can be made smooth by adjusting the string with the **move smooth string tool**.

Curvilinear String Adjustment Notes

Curvilinear segments cannot bend beyond ninety degrees. This can cause a string to "lock up" to movement in a particular direction because one of the segments in the string is about to bend more than ninety degrees.

Because the movement of the curvilinear strings is very complex, it is best to "zoom in" to the area that you are working on. If you encounter locking up of the string or unpredictable behavior, zooming in will usually solve the problem.

11.2 CREATING CURVILINEAR ELEMENTS BY ADDING DATA POINTS

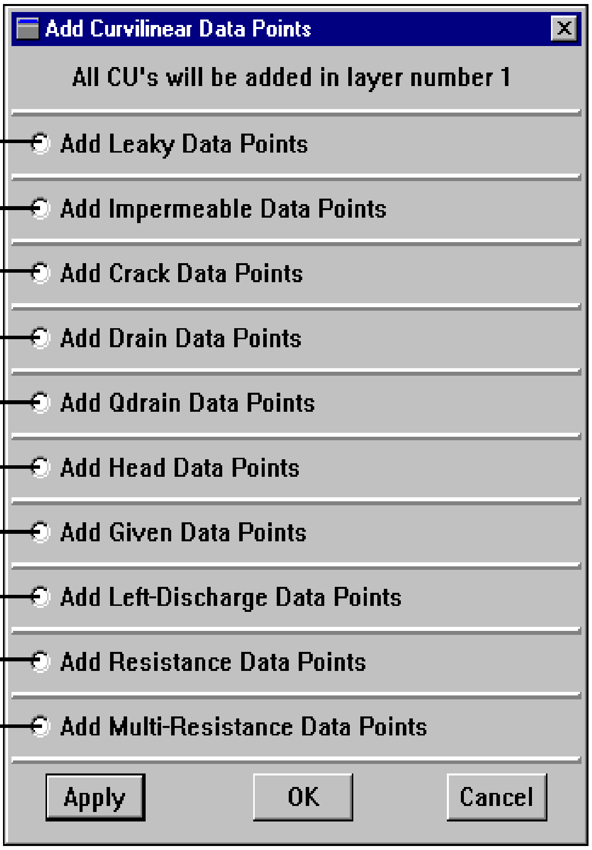
To change a string geometry to an actual curvilinear element aquifer feature, add data points to the string. These curvilinear data points work in much the same way as the Aquitard (ATARD) data points do for VAREL elements, except that in the case of VAREL elements the user adds the control points, whereas in the case of curvilinear elements the control points are added by the program. Once a single data point has been added to a string, the program automatically adds control points along the entire string. The number of control points that are added is determined by the order and degree of over-specification of the polynomials of the string segments. The number of control points added to a curvilinear segment by the program can range from about 4 to over 80.

The values of the parameters (for example value of the head or the resistance) are interpolated at each control point from the data points added. If only one data point is added to a string, all the parameters will have the same value at each control point. If more than one data point is added, the values will be interpolated linearly between points and will be constant and equal to the first or last data point at each end of the string.

Strings can only contain one type of data point per layer, and strings containing different data point types cannot be merged. Two different string types can be connected by using the **add separate string tool**.

To add a data point to a curvilinear geometry, click on the **curve data point** tool. The **Add Curvilinear Data Points** window will pop up. The type of element that will be added is determined here. Select the desired element type, and click OK (Figure 18 shows this window). Now move to the segment to which the data point will be added and click. The program will place the point on the segment.   Now the parameter window will pop up (Figure 19). Enter the required information and accept the values. In order to connect river tributaries to rivers, CU data points "snap" to each other and to the end-nodes of strings. Be careful not to add duplicate data points at the same location on the same string (check this with the edit tool).

Solution results may be improved by increasing the order of the curvilinear polynomial and the degree of overspecification, as was done with DOUBLETs. Click on the **edit** tool and click on a segment. The same overspecification window will pop up as did with the DOUBLETs. Adjust the parameters as needed, again, the higher the values, the more accurate the solution.



All CU's will be added in layer number 1

Used to create a leaky wall

Used to create an impermeable wall.

Used to add a crack in the aquifer

Used to add a drain in the aquifer

Used to add a drain with constant head and that is discharge-specified

Used to create an area where the head is known

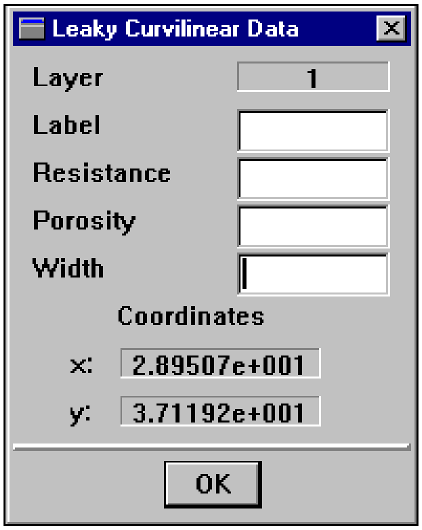
Used to add points of given strength

Used to add left-normal discharge data

Used to add constant resistance data points

Used to add data with varying resistance

Figure 18*. The* ***Add Curvilinear Data Points*** *Window*



Layer	1
Label	
Resistance	
Porosity	
Width	

Coordinates

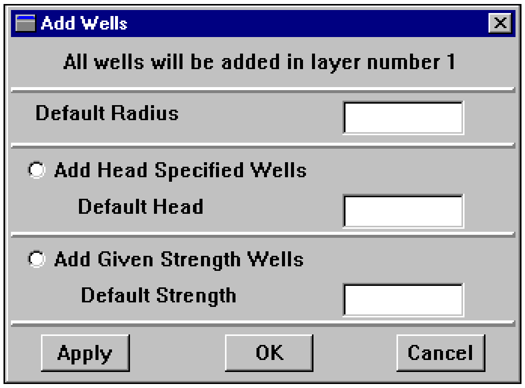
x: 2.89507e+001

y: 3.71192e+001

Figure 19*. The* ***Curvilinear Data*** *Window (specifically, the Leaky window)*

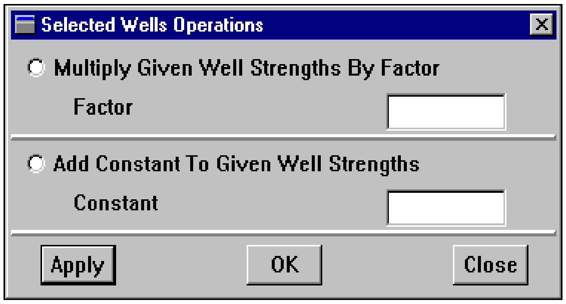
SECTION 12 - WELL ELEMENTS

The **Well Adding Tool** is found in tool position (1,1). This tool is used to add wells to the aquifer in the **Input Window**. When the tool is clicked upon, the Add Wells window appears. The user may choose to add either head-specified wells or given strength wells. The default well is type **Head**, which is a head-specified well. **Head** refers to the head held in the well pipe. The radius for head specified wells must be given because the discharge of the well depends on its radius (the larger the radius, the larger the discharge). The default value for **Radius** is 0.01. In some cases this default can cause problems (in larger models), and you may want to adjust it. The program computes the strength (discharge) of the well such that the head matches the specified head. The control point will never quite be perfectly matched. This type of solution takes longer to solve than one where the strength of the element is given. The accuracy of the solution be can found by typing <CONTROL> in the WELL-CHECK module in the **Command Line Interface.**


All wells will be added in layer number 1

The second kind of well is type **Given Strength**, which is a discharge-specified well. **Strength** refers to the discharge, and **Radius** refers to the well radius. Well discharges are usually expressed in units that are not consistent with units of hydraulic conductivity. In the USA, for example, well discharges are often expressed in terms of gallons per minute, while the hydraulic conductivity is often expressed in terms of feet per day.

You can use the **Operate** button in the **Edit Window** to enter a factor by which all well discharges will be multiplied in order to provide the necessary conversion between units.


Selected Wells Operations
○ Multiply Given Well Strengths By Factor
Factor
○ Add Constant To Given Well Strengths
Constant
Apply
OK
Close

In order to do this, first high-light the wells you wish to operate on. Then click on the **Operate** button in the Edit Window. The **Selected Wells Operations** window will pop up. Choose the option **Multiply Given Well Strength By Factor** and enter the desired factor. All the selected wells will be converted automatically by the program. (A similar procedure may be used to add a constant to all the selected **Given Strength** wells.)

Notice that the **Head** and **Given** wells are both marked in the aquifer by different color squares. In the case of every element, the different types of elements are indicated by different colors, allowing the different types of the same element to be recognized by sight.

Wells may be moved with the **Well Moving** tool. Click on the tool and then on the well you wish to move. Without releasing the mouse button, move the cursor to the desired location and release the mouse button. Although the well has been moved to the new location, the move will have no effect until the system is solved again.

A well's head or strength may be changed using the **Edit** tool. Click on the tool and then on the well to be edited. The element dialog from before will pop up. Modify the desired values and accept the changes by clicking **OK**.

When working with a model, be sure never to have any dry areas, especially not in the area of interest (unless a dry area is intended). In the plot in Figure 19, the strengths of two wells are specified too high, so that the area around the well is dry (aquifers can be dried up in many ways by modeling incorrectly. A dry area can be seen by a jagged, or rough line marking the beginning of the dry area, and is visible around the right well in Figure 19).

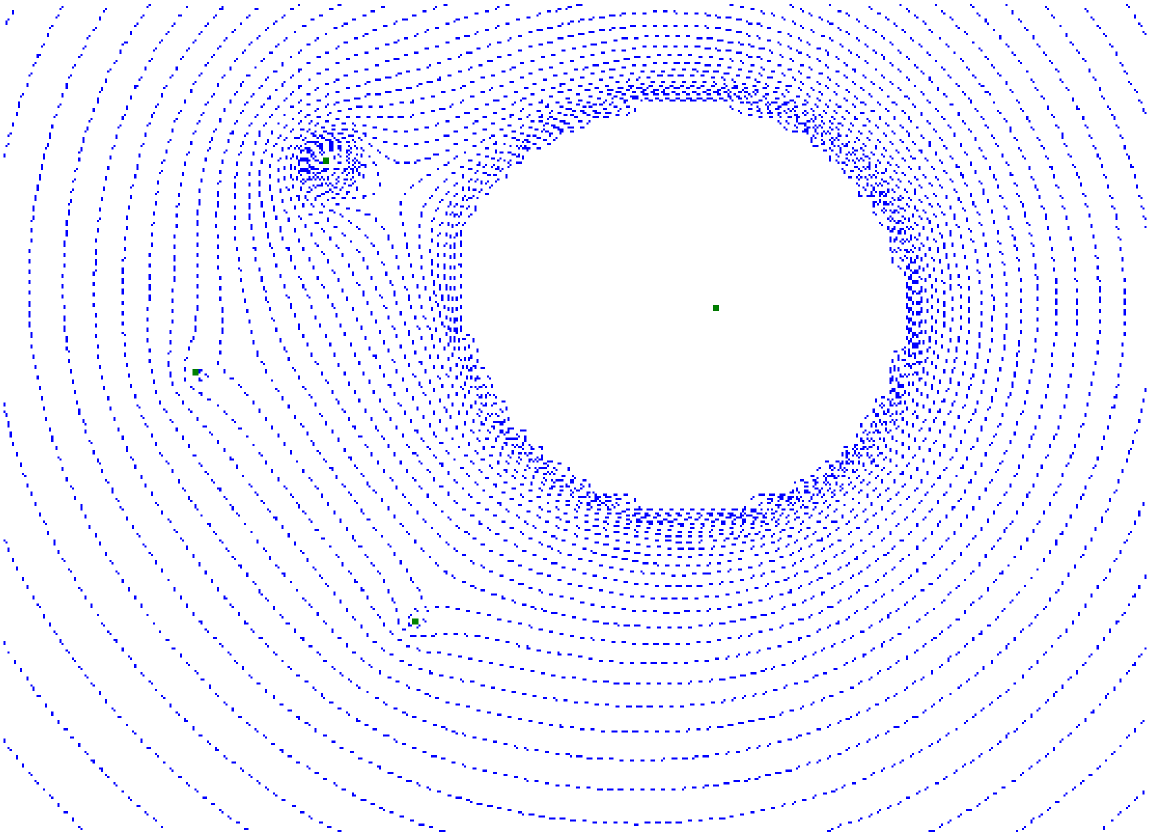


***Fig.20**. Two dried-up wells. Avoid having dry areas in a model*

SECTION 13 - THE SELECTION, CENTER, EDIT AND RESULTS TOOLS

The selection (highlighting), center, edit, and results tools are used to operate on or change the properties of analytic elements, not to create them. Here an overview and guide to using each tool will be given.

13.1 THE SELECTION TOOL

This tool is used to highlight (select) elements. Its primary use is to highlight elements that can then be deleted or modified using the **Edit** window. To use this tool, click on the tool, move the cursor into the **Input Window**, and click on the desired elements. The color of selected elements is yellow, and any element within the radius of the tool's circle will be selected. When deleting, all the selected elements will be removed from the model, and when modifying well parameters, only the parameters of the selected wells will be modified. The selection tool can also be invoked by pressing down the right mouse button while using any other tool.

13.2 THE CENTER TOOL

This tool is used to move the area of interest to the center of the screen. The location at which this tool is clicked on in the window then becomes the new center of the screen. This is particularly useful when zooming. Click with the tool on the area of interest, and then when the **Z** scrollbar is used, the point clicked upon will remain in the center of the screen.

13.3 THE EDIT TOOL

This tool is used to edit or view the parameters of any element in the model. View the element by clicking on it with this tool. If any elements are highlighted, then only those elements will be displayed. The elements window will pop up and its parameters may be either modified or viewed. To see the aquifer properties, click anywhere in the model where no element is located, and the **Aquifer** window will pop up.

The **Edit** tool is also used to set overspecification to CURELs and DOUBLETs. To set the overspecification, click on the CUREL string or on the DOUBLET, and the overspecification window will pop up, and the parameters may be modified.

13.4 THE RESULTS TOOLS

Crosshair tool
This tool is used to get coordinates for the TRACE module. It is used by clicking at the desired location on the screen and then typing a command (in the TRACE module) that will use the coordinates clicked upon.

Tracing tool
This tool is used to generate water particle traces directly. The path of a water particle will be generated in red from the point clicked upon, and the elevation specified.

Well pathline generation tool
This tool is used to generate ten water particle pathlines that end up at the well clicked upon. The elevation used for these traces is the elevation used for the last trace previous to using the *well pathline generation tool*.

The **center** tool works exactly the same as the **center** tool in the **Input Tools** window.

SECTION 14 - THE GIVEN MODULE

The elements in the GIVEN module are different from most of the other elements in two ways: they are simpler functions than those discussed previously and they may only be accessed via the **Command Line Interface**. Given parameters are known beforehand, and the elements simulate rainfall, ponds, and uniform flow. These elements are usually less prevalent in a larger model. The main GIVEN command line follows:

```
\\\ Main Menu; Layer=   1 Level=0  Type <COMMAND> for Command Summary///
given
\\\  Module=GIVEN; Layer=   1   Level=1    Routine=INPUT           ///
<LAYER><UNIFLOW>(Q0)[ANG]<RAIN>(N,XC,YC,A,B)[ANG]<HELP><POND><RESET>
<RETURN>
```

Uniform Flow

The UNIFLOW function is part of the far-field solution and is used to simulate the effects of distant elements. It superimposes a cross-flow on the other elements. Uniform flow can best be checked using the CHECK-GIVEN module. Here we have UNI .01, in the *GIVEN* module, where .01 is the discharge *Q* per unit width (the angle has a default of 0 degrees), see figure 20.

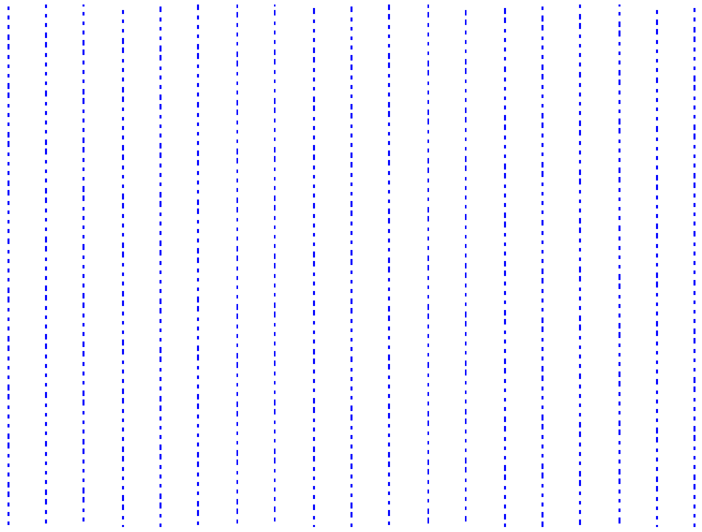


Figure21*. The uniform flow field superimposed on an aquifer with constant head.*

The Rainfall Function

The RAIN function is a particular solution to the Poisson equation. It is a simple quadratic function that produces a family of concentric elliptical equipotentials. The rainfall function is the only function for which infiltration is positive (because the function asks for the infiltration rate). The center, ratio of principal axes (only the ratio is important), and orientation of these ellipses can be specified.

It is advisable to orient the elliptical equipotentials such that they fit as well as possible the boundary conditions. This improves the accuracy of the solution, and reduces the number of degrees of freedom required of the other elements to obtain a good match of the boundary conditions. It is possible to set the minor principal axis of the ellipses to zero, so that the rainfall function reduces to that for rectilinear flow.

Since the potential function due to the rainfall decreases quadratically with the distance measured from the center of the ellipses, negative values of the potential will always occur some distance outside the area modeled. In some cases this is not desirable (especially if only few elements are specified away from the area modeled). In such cases the rainfall may either be modeled by the use of one or more large area elements with given infiltration, or by the use of the POND function discussed below.

Rain can be checked using the CHECK - GIVEN module.

In Figure 11 we have RAIN .005 0 0 5 2 45, in the GIVEN module, where .005 is the infiltration rate, (0,0) the coordinates of the center of the ellipses, 5 and 2 the major and minor axis, and 45 the angle in degrees.

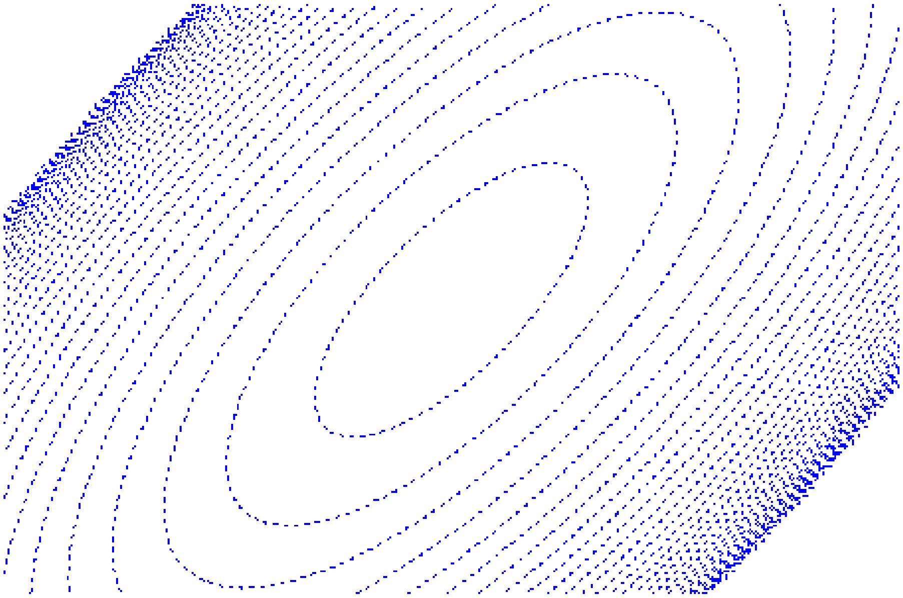


Figure 21. The rainfall function superimposed on an aquifer with constant head. Note that the aquifer is dry on the two sides of the function (explained above).

The Pond Function

The POND function may be used to specify circular ponds with a given rate of extraction (or infiltration, in which case the entry is negative). These ponds should be envisioned as located above the phreatic surface, with leakage occurring through their base, not unlike the VAREL function. They may also be used to simulate infiltration into the aquifer by any other means. Figure20 is a plot of a pond.

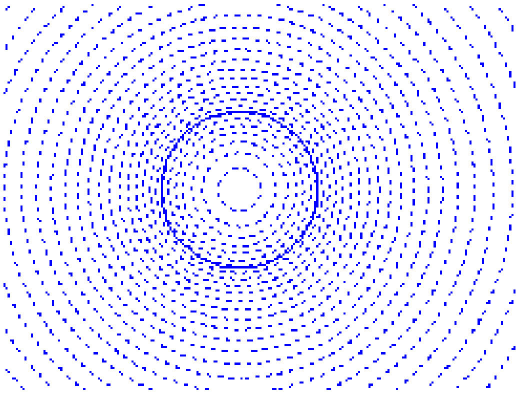


Figure 22*. A pond superimposed on an aquifer of constant head.*

SECTION 15 - MODELING MULTI-AQUIFER SYSTEMS

This is an introduction to how to use MLAEM to model a multi aquifer system. The first step in modeling a multi aquifer system is to set up the layers (aquifers).

15.1 SETTING UP THE AQUIFERS

The global base and thickness of each layer and the thickness of each separating bed can be set using the global aquifer tool. For each layer the base and thickness are specified, and from these parameters the thickness of each separating bed is computed. For example, if the top layer (layer 1) has base 0, and layer 2 has base -40 and thickness 30, the separating bed between layer 1 and layer 2 is calculated to have a thickness of 10. This computation is carried out for all layers. If ever the thickness values for the aquifers and the base values are so specified that a negative value for a separating bed occurs, the program will prompt for a change until a correct set of parameters is entered. When entering global parameters via the command line interface, precede the commands by the word global from within the Aquifer menu. (The **global aquifer** tool controls the global aquifer, and the **poly aquifer** tool only controls the aquifer within its polygon.)

Once the parameters of all layers have been specified, the base of the system (level of the lowermost layer) can be changed using the <SYSBASE> command. This not only changes the base of the lowest layer, but also computes the shifted bases of all the layers above, keeping the thickness of each aquifer and separating bed the same.

The aquifer parameters can be specified along with the other input files for each layer. The sequence for entering a model can be done as follows. First set the layer to number 1 using the **layer tool**, then enter all data for layer 1, including the aquifer data. Then set the layer to number 2 using the **layer tool**, and enter all data for layer 1. This process is continued for however many layers are modeled.

15.2 CONNECTING LAYERS USING LEAKAGE ELEMENTS

Layers are connected using LEAKY VARELs, in the VAREL module or with the VAREL tool. These elements are entered in the same way as GIVEN and RESISTANCE elements. The way a leakage element works is somewhat similar to a RESISTANCE element simulating leakage through a river bottom on the top of an aquifer except that what would be the specified head in the RESISTANCE element is the computed head in the aquifer above the element, and what would be the river bottom is the separating bed.

The leakage of the LEAKY element at each control point is computed as the difference between the piezometric head in the layer above and the piezometric head in the layer below, divided by the resistance of the element at the control point. This computation is carried out with the assumption that the layer below is confined. If the layer below is not confined, then there will be an error in the computed leakage.

The sign convention for flow between layers is that flow down is negative and flow up is positive. This holds for all area elements. This creates a problem when the discharge of a GIVEN element on the bottom of an aquifer is specified. Even though in these cases there is no flow between layers, water is still leaving or coming from the bottom of the aquifer, supposedly to or from the aquifer below. This accounts for the reason that water being extracted from the bottom of an aquifer is negative (flow downward) and recharge into the bottom is positive (flow upward) for given elements (Note: this is the opposite of other elements like wells and CURELs.

In CHECK the sign convention for leakage between layers can be verified. Specify the range and enter <DISCHARGE> in CHECK -VAREL.

To help decide how make the mesh of leakage elements it is possible to use DIFGRID in the **read** module. The difference between the heads in the lower layer and the upper layer can be calculated, and if the resistance of the separating bed is the same across the aquifer, then the most leakage will occur where the contours created by DIFGRID are highest. There should be more elements in areas where the contours are close together and fewer elements where the contours are far apart.

In the RESULTS mode, a particle pathline between layers can be shown.

SECTION 16 - THE RESULT MODULES

16.1 CHECK MODULE

The module CHECK, found in the **Command Line Interface,** allows the user to check that the data entered are indeed stored as intended, and to check that the solution obtained fulfills the boundary conditions with sufficient accuracy. It enables the user to obtain a variety of constants and parameters calculated by the program and to obtain the values of heads, potentials, and discharges at specific points. The module CHECK obtains its information through communication with the various analytic element modules: Most of the commands in CHECK will get you to the next level (level 2) and into the specified analytic element routine from where you may retrieve data specific to the module.

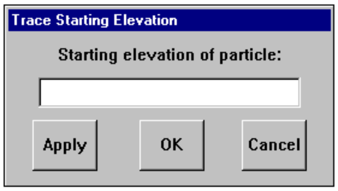
It is good practice to prepare a list of control points along with piezometric data in advance, and enter them into a file which can then be read by the program and processed. You may also have the program print the value at any point of the complex potential OMEGA with the potential as its real part and the stream function PSI as its imaginary part. Values of the two components of the discharge vector may be obtained in a similar way. All derivatives that need be determined to compute the discharge vector are programmed as separate functions obtained by analytic differentiation. Should you have reasons to doubt the accuracy of these derivatives, you may have the program compute the components of the discharge vector through numerical differentiation. It will then print the numerically obtained values alongside those determined analytically. Entering the command *CONTROL* from level 1 will cause all control points to be printed, each control point containing expected and computed values for comparison. It is essential to enter this command after each run and examine the results critically. Should unexplainable differences occur, please contact either your agent or the author of the program.

16.2 TRACE (RESULTS) MODULE

The module **Trace** deals with path line tracing and contaminant transport. The path line tracing routine makes use of a Runge-Kutta iterative procedure for determining particle paths in three dimensions. Path line elevations are determined using the approximation that the resistance to flow in the vertical direction is zero. Contaminant transport is at present implemented as retardation and decay. Path lines may start at any elevation of the aquifer, and will terminate upon reaching elements. Exceptions are leaky walls, and under some conditions VARELs. Path lines are always continued through leaky walls; they will either terminate at VARELs or path underneath these elements.

Pathline traces may be entered two ways: using the **tracing** tool or using the **crosshair** tool.

*Using the **tracing** tool*: Click on the tracing tool and the **Trace Starting Elevation** window will pop up. Here the user enters the starting elevation of the particles to be traced. After a value has been entered, click **OK** and the window will disappear. Click on the location where the particle trace is to be started and a particle pathline will be generated. To create another pathline, follow the procedure above and the new one will be added to the plot.



*Using the **crosshai**r tool*: Click on the crosshair tool, move the cursor into the **Results Window**, and click at the location in the model where data are to be obtained. Data will be generated about the point at which the cross is placed. Move the cursor into the command line and notice that the program has automatically entered the **Trace** module. Now the type of information desired will be entered. In this case we will generate a particle trace; type the command *TRACE* and a pathline will be generated.

The following commands allow the their respective information to be obtained: <BASE>, <SURF>, <HEAD>, <POTENTIAL>, <DISCHARGE>, and <LEAK> (For more information on the specific meaning of the commands, see the help file). Type the command and the specified information will be produced about the point at which the cross hair is located. To obtain that information again about another point in the aquifer, move the cursor to the point where the data is to be obtained and click with the right mouse button. The same data will be automatically generated every time the right mouse button is clicked. For a more detailed explanation of one of the commands and how the information is to be interpreted, see section **4.3 Obtaining Results From Plots Using the Results Mode**.

Plots of path lines may be saved by entering the command *SAVE* prior to generating the path line. They may be retrieved by entering the command *READ*.

Sectional plots of path lines may be generated as follows: First produce on the screen a pattern of path lines, either directly or by reading in a file. Then move the cross-hair to any point of any one of the path lines, and enter the command *SECTION.* The program will then search for a path line that contains the specified starting point. It does this by using a tolerance which has a default value, but can be reset. Once the section has been drawn on the screen, the command line for the section module is displayed. In this command line the default window may be reset, and a scale factor for the vertical coordinate may be specified, if so desired. Entering the command *PLOT* or moving the mouse out of the **command line interface** will result in the section to be displayed that contains the desired path line. At this stage the cross-hair again appears, allowing you to enter starting points of further path lines contained in the same section.

Sectional plots of path lines may be saved for later retrieval by entering the command *SSAVE* prior to determining the path lines in section. They may be retrieved by the command *SREAD*. Path lines may also be initiated manually. This is done by entering the command *CURSOR OFF* prior to *PLOT* or *LAYOU*T. Attributes that control the path line tracking may be set by entering the command *SET*. This will lead you into level 2 of the module TRACE. Numerous commands may be entered from *SET*: The most important ones are the maximum and minimum values of the step used in the Runge-Kutta procedure. The maximum value is that normally used; under certain conditions (for example the approaching of a well) the step size is reduced to the minimum value. The step size is gradually increased again when the condition necessitating the decrease no longer prevails. The *SET* routine also allows for the setting of parameters controlling time and concentration level markers. Specifying a time increment or concentration decrement (only one is allowed at a time) will cause markers to appear on the path lines at points where the time or concentration matches the initial value plus or minus an integer times the specified increment. You can instruct the machine to print path line data at some specified length interval on the path line.

Contaminant transport is currently supported in terms of retardation and decay. The retardation and decay parameters may be entered after typing the command *TRANSPORT*, which will lead you to level 2. Note that the time markers placed by the program correspond to the travel times of the solutes, rather than of the water particles; specification of a retardation factor (which should be greater than 1) will cause the time markers to be closer together, since: the solutes move slower than the water.

SECTION 17 - THE SERVICE MODULES

17.1 SWITCH

Because the command <SWITCH> is used often from a variety of places in the program, it is available from numerous menus, besides the main menu. This module makes it possible to re-direct input and output. The ability to re-direct input makes it possible to prepare one or more input file(s) in advance, containing commands to be read by the program. You may switch from one file to another, but never back to the same file: only one level of switching is implemented. You may, for example, read certain data in from a file, switch back to the keyboard, enter some data by hand, and then switch back to one or more files.

Alpha-numerical output from the program is divided into three groups:

- MESSAGES
- ERROR MESSAGES
- OUTPUT

The messages from the program are, among others, command lines and menus. Error messages are generated when data on an input line are missing, illegal commands are given, or some error condition is detected by the program. Output is defined as the numerical data produced by the program and obtained, for example, through CHECK. You may redirect any of these three types of output to a file or device. If you re-direct to NUL, the output will be suppressed. In addition to redirecting the above three kinds of output, you may echo them to any file or device. For example, if you set the output echo to PRN, the output will appear both on your screen, and on the printer. It is generally preferable to echo the output to a file, because than the data may both be archived, and be printed at any time after terminating or pausing the program.

The above features make it possible to view the data read in from a file without having the program messages interfering. To do this, switch the input to a file, suppress the messages ordinarily displayed before each command by sending them to NUL, echo the input to the screen by setting input echo to CON, and leave the error messages at the default. As a result, the program will display each command as it reads it from the file without irrelevant messages to interfere, while displaying the error messages whenever a bad command line is read.

It is possible to activate logging either on the default log file (LOG.DAT), or to a file with a name of your choice. Logging includes all input and output during a session. Using a log for all important runs is highly recommended; in this way it is easier to track down the cause of errors.

Upon entering the command LU a list will be displayed of all file names and their associated logical units. This list is useful in particular if you wish to send more than one type of output to a single file. The switch module is written in FORTRAN, which does not allow two different logical units to be attached to a single file, which makes it necessary to deviate from the default logical unit assignments for the above case. The program will tell you which assignment to make, and it is explained in the help file how to do this.

17.2 PSET

The module PSET controls the flow of graphics generated by the program

The program uses different line types (e.g., solid, dashed), different line colors. Each type is reserved for a particular class of curves. These classes are defined in the HELP file; they are identified by code numbers. Each of these code numbers is associated with a certain line-type, color, marker type, and marker size. For example, the default for colors is that types 1,2,3, and 4 are associated with codes 1,2,3 and 4 (the actual color for each type depends on your graphics adapter, as well as the palette that is currently active). You can re-set this association in any way you like. For example, if you associate code 2 with line type 1, then the path lines will appear as solid lines rather than as dashed lines because code 2 corresponds to path lines, and line type 1 is solid.

SECTION 18 - THE STAND-ALONE COMMANDS

There are miscellaneous commands that may be entered from the main menu and sometimes from other menus as well. They are referred to here as stand-alone commands because they do not cause a new menu to be displayed. These commands are listed below:

<READ> will display the following command line:

```
 <S>OLUTION,<G>RID,<B>OTH,<D>IFGRID,<R>ETURN
```

which will allow you to read in a solution, grid or both. It will also allow you to read in a grid and subtract it from the grid currently in memory by use of DIFGRID. The two grids must have the same number of grid points.

<SAVE> will display the following command line:

```
<S>OLUTION,<G>RIDl<B>OTH,<L>AYOUT,<P>=LAYOUT+GRID,<R>ETURN
```

which will allow you to save a solution, grid, both a solution and grid, a layout, and both a layout and a grid.

<HELP> will display the help file that corresponds to the current menu.

<LAYER>(NR) will set the current layer number throughout the program.

<LKUP>(<ON>/<OFF>) sets the default boundary for leaky elements to the top (ON) or to the bottom (OFF) of the aquifer. This command also sets the boundary button in the “layer” window.

<WINDOW>[X1,Y1,X2,Y2] sets the viewing domain of the model to a window with a lower left-hand coordinate equal to (X1,Y1) and an upper right-hand coordinate equal to (X2,Y2).

<SOLVE> will solve the problem. This command uses a default accuracy and number of iterations.

<ITER>[ACCURACY,MAX.NR.ITERATIONS] will solve the problem to the specified ACCURACY and with the maximum number of steps specified by MAX. NR. ITERATIONS

<ZSOLVE>[ACCURACY,MAX.NR.ITERATIONS] will solve the problem to the specified ACCURACY and with the maximum number of steps specified by MAX. NR. ITERATIONS, but will first initialize the solve matrix to zero. This command is useful for difficult solutions that have the tendency to create the error “Singular System of Equations” or “Ill-Conditioned Matrix”. This command is the same as choosing the option “Reset Matrix Before Solving” option in the solve window.

<COINCIDE>[DIST.][<CHECK>] allows you to check whether or not control points coincide. It also allows you to set the minimum tolerance (DIST) between control points. By adding the command CHECK, the program will check for coinciding control points after the command is entered.

<DELETELOG> clears the text log from the Command Line Interface window.

<LOGSIZE>(NR. CHARACTERS) sets the maximum number of characters of the text log to NR. CHARACTERS. The default is 40,000 for PC systems and 2,000,000 for UNIX systems. Computer systems with large amounts of memory can generally operate smoothly with a higher text log limit. If the maximum text log size is too large, the command line interface will lock up after a certain period of program operation.

<FONT> pops up a window allowing the user to select the font used in the **command line interface**.

<LAYOUT> causes a layout of all elements to be displayed.

<PAGE> clears the screen.

<TITLE>[PROJ. NAME] makes it possible to enter the title (less than 17 characters) of the project currently in the program or to display the current title (by pressing ENTER after the command <TITLE>).

<RESET> resets all parameters in the program to their default values.

<STOP> ends the program session.

APPENDIX A
WINDOW CONFIGURATION PROBLEMS

The default size of the main MLAEM window is the size of the screen that the program is being run on.. MLAEM accommodates different screen resolutions by using different versions of the **aem.cfg** file included on the directory where the program is located. If some of the windows are not visible on your screen then you will have to use a different version of the **aem.cfg** file. The configuration filenames are broken into two groups, one for large fonts and one for small fonts (in order to determine your screen resolution and whether your system is using large fonts or small fonts, see the **Settings** tab in the **Display** section of the **Control Panel** in Microsoft Windows 95 or NT).

Small Font Files

Small font configurations are saved in the files **aemS1600.cfg** (for a screen resolution of 1600x1200), **aemS1280** (for 1280x1024), **aemS1152.cfg** (for 1152x1024), **aemS1024.cfg** (for 1024x762), **aemS800.cfg** (for 800x640 or SVGA), and **aemS640.cfg** (for 640x480 or VGA - this resolution is not recommended).

Large Font Files

Large font configurations are saved in the files **aemL1600.cfg** (for a screen resolution of 1600x1200), **aemL1280** (for 1280x1024), **aemL1152.cfg** (for 1152x1024), **aemL1024.cfg** (for 1024x762), **aemL800.cfg** (for 800x640 or SVGA), and **aemL640.cfg** (for 640x480 or VGA - this resolution is not recommended).

Changing the Window Configuration

In order to change the window configuration, copy the desired file above to the name **aem.cfg**.

Other Configuration Issues

In some installations of the Microsoft Windows operating system, the options used are such that the tool-button windows are not completely visible. These windows cannot be resized with the mouse and you will have to edit the "aem.cfg" file manually. The following section is a partial listing of a "aem.cfg" file for a screen resolution of 800x640 along with comments (following the //'s). The tool-button windows are the first four entries:

```
inputToolkit 4 42 70 409 , 0 0              // input mode toolkit window
cursorToolkit 4 42 70 89 , 0 0              // results mode toolkit window
ShortcutWindow 621 64 70 121 , 0 0          // shortcut window
layers 4 451 70 221 , 0 0                   // layers window
mlmnvw 0 0 801 601 , 0 0                    // main program window
…
```

In each of these lines, the coordinates listed have the following format. For example:

inputToolkit x y width height , x-offset y-offset

Here *(x,y)* are the coordinates of the upper left-hand corner of the window (in pixels), *width* is the width of the window (in pixels) and *height* is the height of the window (in pixels). The *x-offset* and *y-offset* values are not used.

In summary, if a window is not completely visible, first try re-sizing the window. If this does not work, then locate the window in the "aem.cfg" file and change the width and height until the window is fully visible. It usually takes several tries of changing the values and restarting the program before they are right.

APPENDIX B
NUMERICAL ACCURACY, UNITS, FILES, PRINTING

NUMERICAL ACCURACY

Numerical inaccuracies result from the loss of significant digits during the mathematical operations performed by the program. Such a loss may be caused simply by the range of values of the variables, or by the nature of the problem that one attempts to solve. The former loss of accuracy may be controlled to some extent by the user; the latter cannot be avoided, but should be understood in order to interpret the results produced by the program appropriately. The range of values of the variables depends on natural conditions, as well as on the choice of the coordinate system and the reference level for the piezometric head.

If the origin of the coordinate system is chosen far away from the center of the area modeled, then the range of values of the coordinates is small with respect to their average magnitude, which will cause loss of accuracy.

A similar problem arises when the reference level for the piezometric head is chosen to be high with respect to its range of values.

In order to minimize unnecessary loss of accuracy, you are encouraged to choose the origin of your coordinate system near the center of the area modeled, and to set the reference level for the piezometric head to a value that is in the order of the range of values that you expect.

Unavoidable numerical inaccuracies may result from natural conditions, such as a large range in boundary values, or aquifer characteristics. An example of such a case is a highly permeable crack intersecting an area of very low permeability. Such loss of accuracy is often unimportant, but the modeler should establish whether this is the case.

The program has a feature built in, referred to in this manual as the CONTROL feature. When activated, CONTROL will produce a list of all the control points on all the elements, and print the computed boundary values, alongside the boundary values entered by the user. This command should be given after each run, and it should be determined whether the accuracy is acceptable or not.

PROGRAM UNITS

The program works in any set of consistent units. That means, for example, that if the hydraulic conductivity is entered in cm/s, then infiltration from rainfall must be measured in cm/s also. Furthermore, in this example the units of time would be seconds. The program always thinks in terms of these 'program units'. In certain instances, the program units are inconvenient (for example, it is inconvenient to be forced to think in terms of seconds, as in the above case). For two cases, the program provides the user with the possibility to convert from 'user units' to the program units.

FILES

The program has the capability to save solutions, grids, combined solutions and grids, path lines in plan, path lines in section, maps in binary form, and files containing plots generated during a session. These binary files are quite different, and extreme care must be taken not to mix them up. Should you ever read the wrong file, the program is likely to lock up the system, requiring a cold re-boot. It is recommended that you adopt a system of naming the files to avoid mix-up.

The developer of the program reserves the following extensions for the various files: SOL for solutions, GRD for grids SAG for combined solutions and grids, PTH for path lines in plan, PTV for path lines in section, MAP for maps, and PLT for plot files. In addition to binary files, there are various alphanumeric files made for input, or created by the program. The author reserves the extension DAT for such files, with the exception of log files, which have the extension LOG. The program will sometimes need as many as 12 files to be open at one time. You should make sure that your system is capable of handling at least this number of files.

PRINTING

There are three ways to print from MLAEM:

DXF Files

The program can write out files in DXF (Auto-CAD compatible) format. From the main pull-down menu choose **Write DXF File** - everything currently displayed on the screen (except particle traces) will be written to a DXF file.

In order to write particle traces out to a DXF File, choose **Capture Screen Events to DXF File** from the main pull-down menu - everything currently displayed on the screen will be written, in addition to any particle traces made while the **Capture Screen Events to DXF File** option is checked. Sectional traces can also be captured in this fashion - in this case, choose **Capture Screen Events to DXF File** after the sectional plot is visible on the screen. Be careful not to leave this option checked longer than necessary. The program will continue to write to the file and it is possible to exceed all available disk space.

After the DXF file has been written, use Auto-CAD or other CAD software to print.

Note: The size of the markers (VAREL control points, curvilinear data points, etc.) and the accuracy with which the curvilinear elements are written to the DXF file is dependent on the zoom level. The more the model is zoomed in, the more accurate the drawing is, and the smaller the markers are. By writing a DXF file from the **input window**, the whole model is written out. By writing a DXF file from the **results window**, only the part of the model that is currently visible on the screen will be written out (note that this is a good way to "clip" large DXF files).

SURFER Formatted Files

The program can write out grids and layouts in SURFER format via the GRID module. See the GRID module online help for further instructions.

Screen Dumps

Operating systems (such as Windows and UNIX) have the ability to make screen dumps and have utility programs (such as Paintbrush or Microsoft Word in Windows) that can be used to edit and print the screen dumps. Under Windows press the PRINT SCRN key to capture the entire screen and press ALT + PRINT SCRN to capture the currently active window. To paste the screen capture in Microsoft Paintbrush, choose "Paste" under the Paintbrush edit pull-down menu or press CTRL + V. To paste the screen capture in Microsoft Word, open a document and press CTRL + V. The resolution of the printout in Microsoft Word is higher than in Microsoft Paintbrush.