

MIGRATE_{v9}

User's Guide

NOTICE

Although this software has been tested and the documentation reviewed and experience would indicate that it is accurate within the limits given by the assumptions of the theory implemented, no warranty as to workability of this software or documentation is made. No warranty or representation either expressed or implied (including warranties of fitness) shall apply to this software or documentation. No responsibility is assumed for any errors, mistakes or misrepresentations that may occur from the use of this software or documentation. The user accepts full responsibility for assessing the validity and applicability of the results obtained with this software and documentation for any specific case.

Copyright © 1995 R.K. Rowe, J.R. Booker and M.J. Fraser

All rights reserved. Printed in Canada.

Not to be reproduced without the written permission of the distributor.

Distributed by:

GAEA Technologies Ltd.
P.O. Box 334
Waterloo, Ontario, Canada

Contents

INTRODUCTION	6
Using this Guide	7
CHAPTER 1	8
Description	8
Features	9
GUI Features	10
Assumptions	10
Integration	11
Advection	11
Diffusion	12
Dispersion	13
OtherMechanisms	13
Radioactive or Biological Decay	14
Two DimensionalContaminantMigration	14
BoundaryConditions	14
Constant ConcentrationTop Boundary	15
Finite MassTop Boundary	15
CHAPTER 2	18
Program Use	18
Menu Line	18
Status Line	18
Desktop	19
Input Lines	19
Multiple Choice Boxes	19
Single ChoiceBoxes	20
List Boxes	20
History List Boxes	20
Cancel	21
Short-cut keys	21
Customizing the Program	21
CHAPTER 3	21
Open Output	22
Save Data	22

Save Data As	22
Save Output As	22
Change Dir	22
DOS Shell	23
File Dialog	23
General Data	27
Constant Concentration at Surface	29
This is the concentration of the contaminant species in the landfill.	
Finite Massat Surface	30
ImpermeableBottom	32
Zero Concentration Bottom	32
AquiferBottom	32
Layer Data	33
Fractures	36
Primary Liner (Subtitle D) Landfill	41
Leakage Rate (Subtitle D) Landfill	42
Primary and Secondary Liner Landfill	44
Source Data	45
Head Data	45
GeomembraneData	46
Liner Data	46
Aquifer Data	47
CHAPTER 5	48
Concentrations	48
CHAPTER 6	50
Concentration versus Time	51
Concentration versus Distance	53
Colour Concentrations versus Depth and Time	54
List Outputto Screen	55
CHAPTER 7	63
Add-Ons	65
CHAPTER 8	66
Environment	66
Mouse Control	68
Colour Control	69
CHAPTER 9	71

HelpContents	71
Help Index	71
Help Usage	71
RevisionHistory	71
About	71
APPENDIX A	83
Example Summary	83

a

INTRODUCTION

There are two programs that are distributed as part of the MIGRATE software, these are MIGRATEv9 and MIGRATE-GUI. The MIGRATEv9 program performs the contaminant migration calculations, and has the DOS filename MIG900.EXE. MIGRATE-GUI provides the graphical user interface to MIGRATE, and has the DOS filename MIGRATE.EXE.

The MIGRATE-GUI program can call the MIGRATEv9 program interactively or set up a batch file that can be used to run the MIGRATEv9 program at a later time. It is recommended that the MIGRATE-GUI program be used for creating, editing, executing, and displaying MIGRATE datasets. To start the program switch to the directory containing the MIGRATE programs, and type MIGRATE.

Using this Guide

The user's guide is arranged into several chapters and appendixes, and is designed to make it fast and easy to learn all of the features of the programs. Below is a brief description of each section of the guide:

Chapter 1. Program Features describes the features of the MIGRATEv9 program. The transport mechanisms, two-dimensional migration equation, and boundary conditions implemented in the programs are also discussed.

Chapter 2. Basics explains how to interact with the MIGRATEv9 program, and conventions used throughout the program.

Chapter 3. File Menu describes how to access input and output files, change directories, shell to DOS, and exit the program.

Chapter 4. Data Entry Menu describes the data required to use the MIGRATEv9 program.

Chapter 5. Execute Menu describes how to execute the MIGRATEv9 program using the data entered previously.

Chapter 6. Output Menu describes how to display and print the output from the MIGRATEv9 program.

Chapter 7. Tools Menu describes how to import POLLUTE files the calculator, and add-ons.

Chapter 8. Preferences Menu describes how to customize the program to the user's preferences.

Chapter 9. Help Menu discusses the various types of help that can be accessed by the user.

Bibliography lists the references used in the manual and other pertinent references.

Appendix A. Examples contains numerous example datasets and detailed descriptions for their use in the program, these examples are also contained on the installation disk.

PROGRAM FEATURES

Description

The contaminant impact from sources such as landfills and spills will be dependent on the interaction between the engineered systems and the hydrogeology. Computer modeling may be used to simulate this interaction and to assess the probable contaminant impact of a contaminant source. In addition, computer modeling may be used to examine the effects of changes in the design and performance of the engineered systems, or the local hydrogeology.

MIGRATEv9 is a computer program that implements a solution to the two-dimensional dispersion-advection equation for a layered system. The analytical solution is developed for the case of two-dimensional contaminant transport for a single solute, and for a layered deposit of finite thickness (Rowe and Booker, 1985, 1986, 1987, 1991; Rowe et al, 1994). A Laplace transform and a Fourier transform are used to find the solution of the advection-dispersion equation. The solution is then inverted numerically using a technique proposed by Talbot (1979).

Unlike finite element and finite difference formulations, MIGRATEv9 does not require the use of a "time-marching" procedure. MIGRATEv9 uses a finite-layer formulation that provides numerically accurate results for a given idealization while requiring relatively little computational and data entry effort. Thus, in its basic mode of operation the concentration of contaminant can be directly determined at any specified time without calculating the concentration at earlier times.

One or more landfills, or other contaminant sources, may be modelled. These contaminant sources may be adjacent or offset from each other. The subsurface geology beneath the landfills, or other contaminant source, is split into a number of layers, and these layers may be further sub-divided into sublayers (see Figures 1.1 & 1.2). If appropriate these layers may have different properties. No gain in accuracy is achieved by dividing a layer into sublayers (except in the time varying properties option), however, concentrations will be determined at the boundaries between these sublayers. The contaminant concentration of a single species is calculated at these depths and at specified lateral distances.

The landfills (or other contaminant sources) may be modeled as a surface boundary condition, or as a physically buried layer within the layered system (see Figures 1.3 &

1.4). If the landfills are modelled as a surface boundary condition then the contaminant mass may be either infinite (i.e., constant concentration) or finite (i.e., concentration decreases with time as mass is removed from the source).

The base stratum beneath the deposit may be either permeable or impermeable. If the base stratum is permeable, the concentration at the interface between the deposit and the permeable stratum will depend on the outflow velocity within the permeable stratum. If the base is impermeable, the flux at the deposit-base interface is assumed to be zero.

If a permeable base stratum exists beneath the deposit, it can be modeled either as a boundary condition or as a physical layer. The first approach assumes a high vertical dispersion coefficient within the base stratum giving a uniform concentration throughout its thickness. The second approach allows consideration of a vertical diffusion/dispersion coefficient in the base layer, and will give a diffusion profile through its thickness. If it is modelled as a boundary condition then the following are calculated: the flux entering this stratum, the mass of contaminant located in the permeable base stratum, and the mass of contaminant that is removed from the permeable base stratum due to advective transport.

Features

The following features are included in **MIGRATEv9**:

- one or more landfills, or other contaminant sources, may be modelled;
- the deposit may be subdivided into individual layers, with
- the contaminant source can be modeled either as a surface
- depletion of the source (i.e., landfill) with time can be modelled, or the source can have a constant concentration with time;
- removal of leachate by means of a leachate collection system
- the bottom boundary of the deposit can be impermeable,
- the effect of geochemical reactions on a non-conservative ion may be considered using linear sorption;
- the mass or flux of contaminant entering and leaving the

New and improved features in this release are:

- some or all of the layers may be fractured, the fractures may
- the properties can be varied with time to simulate such effects
- radioactive or biological decay can be modelled separately for the source, deposit, and base;
- top and bottom of landfills may have different widths, to model the effect of side-slopes.

GUI Features

The MIGRATE-GUI (Graphical User Interface) is used to interactively interface with the **MIGRATEv9** program. This program uses a top menu bar with pull-down sub-menus and pop-up data windows, detailed information on how to interact with the program is found in **Chapter 2 Basics**. Context sensitive help is available throughout the program any time by pressing the F1 key, for further information on how to use help see **Help in Chapter 9**.

Using MIGRATE-GUI the user can:

- create data sets,
- execute these data sets in MIGRATEv9,
- display and print the results as graphs, • store the datasets and output in files,
- display and print the concentrations in colour,
- create batch files for multiple datasets,
- import POLLUTE files,
- use an on-screen calculator,
- customize the program interface.

The program supports a wide variety of graphic screens; including Super VGA, VGA, EGA, CGA, and Hercules. Most of the popular printers are supported; these include Epson/IBM 9 and 24 pin, HP LaserJet/DeskJet, HP Pen Plotters, HP PaintJet, HP ThinkJet, Cannon Bubblejet, and Postscript. In addition the graphic displays can be saved to a file in Desktop Publishing PCX format.

Assumptions

It is assumed that for the hydrogeology being considered contaminant transport is governed by the two-dimensional advection-dispersion equation within the porous medium (see Rowe and Booker, 1985, 1986, 1987, 1991; Rowe et al., 1994). The **MIGRATEv9** program implements a finite layer analysis which allows for multiple layers having different properties, however within any one layer, it is assumed that the parameters do not vary with lateral position.

It is also assumed that the Darcy velocities do not vary with lateral position within any layer of the deposit. The effects of velocities

varying with lateral position can be estimated by performing the analysis for a range of velocities corresponding to the expected variability with lateral position. This program may not be suitable for problems where there is a significant variation in Darcy velocities with lateral position. Judgment will be required in assessing the program's applicability for these situations.

Integration

The theoretical development involves the use of a Fourier transform and hence the synthesis of the final solution involves some numerical integration. These integrations are performed numerically by decomposing the integration range into a number of subintervals and then using Gauss quadrature within each subinterval.

The width and number of integration subintervals which are required to achieve a reasonable accuracy (say 0.1%) depends somewhat on the geometry and properties of the problem under consideration. These parameters can be determined from a few trial calculations for a representative point and time of interest. The program is capable of performing four levels of Gauss quadrature, FINE, NORMAL, WIDE, and SELECT. When SELECT is specified the user must input the integration parameters. The computation time is directly proportional to the amount of integration.

The error involved in the integration depends upon the problem and the depth of the point of interest beneath the source. This error tends to be greatest at the surface outside the landfill, and decreases with depth below the landfill (see Example 5).

Transport Mechanisms

The migration of dissolved contaminants through the subsurface involves different transport mechanisms depending upon the type of soil, presence of fractures, degree of saturation, and the soil - contaminant interaction. For a saturated clay or silt the primary mechanisms may be advection and diffusion, whereas for a saturated sand the primary mechanisms may be advection and dispersion. In a fractured soil the primary mechanisms are advection and dispersion along the fractures and diffusion from the fractures in the matrix. Soils with clay particles or organic matter may also act to retard the migration of contaminants by adsorbing the contaminant. These transport mechanisms are described in detail below.

Advection

When water flows through the soil it will carry contaminants along with it in solution, this process is called advection. The amount of

contaminant mass transported in the x or z direction by advection is proportional to the groundwater (seepage) velocity in the x or z direction, v_x or v_z , and the concentration, c , of the contaminant. Where x is the lateral offset and z is the vertical depth. This mass can be measured in the x and z directions, during a unit of time, this is called the flux, f_x and f_z . The flux is then the mass of contaminant transported per unit area per unit time and is given by:

$$f_x = n v_x c = v_{ax} c$$

$$f_z = n v_z c = v_{az} c$$

where,

- n = effective porosity of the soil,
- v_x and v_z = groundwater (seepage) velocity, in the x and z directions,
- v_{ax} and v_{az} = Darcy velocities,
- c = concentration of the contaminant at the point and time of interest.

It should be noted that the velocity that the contaminant moves through the soil is the groundwater velocity and not the Darcy velocity. If the groundwater velocity is zero (i.e., there is no flow) then there would be no advection.

Diffusion

Diffusion is the process where contaminants in the soil will migrate from areas of high chemical concentration (potential) to areas of low chemical concentration (potential). The mass flux transported by diffusion is proportional to the concentration gradient and is given by:

$$f_x = -n D_{ex} \frac{\partial c}{\partial x}$$

$$f_z = -n D_{ez} \frac{\partial c}{\partial z}$$

where,

- n = effective porosity of the soil,
- D_{ex} and D_{ez} = effective diffusion coefficients,
- $\partial c/\partial x$ and $\partial c/\partial z$ = concentration gradient.

The negative sign in the above equation arises from the fact that contaminants move from areas of high concentration to areas of low concentrations. By convention if the velocity in the z direction is positive the flow is out of the landfill, and if the velocity is negative the flow is into the landfill. The direction of transport for diffusion and advection can be in the same direction or in opposite directions. If the direction of diffusive transport is in the same direction as that

of advective transport, then diffusion will increase the amount of contaminant transported and decrease the time taken for the contaminant to move to a given point. Diffusion can also occur in the opposite direction to advection. For example, even if groundwater is flowing into a landfill, the high concentration of contaminant in the leachate can cause diffusive transport out from the landfill. Thus, even though the groundwater flow is into a landfill contaminants can still escape from the landfill by diffusion.

Dispersion

In a granular layer (eg. an aquifer) or a fractured layer there can be significant localized variations in the groundwater flow. These variations will cause mechanical mixing within the layer, this process is called dispersion [Freeze and Cherry, 1979]. Although the process is very different to diffusion it can be modelled mathematically in the same manner, and the two processes can be grouped together as the "coefficient of hydrodynamic dispersion", D_{xx} and D_{zz} , viz.:

$$D_{xx} = D_{ex} + D_{mdx}$$

$$D_{zz} = D_{ez} + D_{mdz}$$

where, D_{ex} and D_{ez} = effective diffusion coefficient,

D_{mdx} and D_{mdz} = coefficient of mechanical dispersion.

In unfractured clayey soils the coefficient of hydrodynamic dispersion is often controlled by the diffusion coefficient, and the coefficient of mechanical dispersion is negligible. In sandy soils and fractured layers the opposite is generally true and dispersion dominates [Gillham and Cherry, 1982; Rowe, 1987].

Advective-Dispersive Transport

The mass flux for advective-dispersive transport (including diffusion) is given by:

$$f_x = n v_x c - n D_{xx} \frac{\partial c}{\partial x}$$

$$f_z = n v_z c - n D_{zz} \frac{\partial c}{\partial z}$$

where the parameters are the same as those defined previously and D is the coefficient of hydrodynamic dispersion.

Other Mechanisms

In addition to the transport mechanisms mentioned above, the migration of contaminants may also be controlled by sorption and radioactive or biological decay, that can be modelled in **MIGRATEv9**. These mechanisms both serve to reduce the mass of

dissolved contaminant available for transport. Both mechanisms are discussed below and in detail in Chapter 4.

Sorption

Sorption is the process whereby contaminants are removed from solution by interaction in the soil. Typical interactions are cation exchange in clays and the attraction of organic contaminants to organic matter in the soil. Sorption can be modelled as a function of the concentration of the contaminants in the soil. **MIGRATEv9** can model linear sorption as described in detail in Chapter 4.

Radioactive or Biological Decay

Some contaminant species experience radioactive decay or biological degradation and the concentration of these contaminants may decay as a function of time. The rate of radioactive decay is very predictable and is controlled by the half-life of the contaminant species. Whereas, the rate of biological decay is a function of several factors, including the presence of the appropriate bacteria, the presence of a suitable substrate, and the temperature. Both types of decay are often modelled by first order decay, with the controlling parameter being the half-life. This process is described in greater detail in Chapter 4.

Two Dimensional Contaminant Migration

The theory implemented by the **MIGRATEv9** program, in its basic mode of operation, is described in detail by Rowe and Booker [1985, 1987, 1991] and Rowe et al [1994]. According to this theory contaminant migration in two-dimensions, for an intact material, is governed by:

$$n \frac{\partial c}{\partial t} = nD_{xx} \frac{\partial^2 c}{\partial x^2} + nD_{zz} \frac{\partial^2 c}{\partial z^2} - n v_x \frac{\partial c}{\partial x} - n v_z \frac{\partial c}{\partial z} - \rho K_d \frac{\partial c}{\partial t} - n \lambda c$$

where,

- c = concentration of contaminant at depth z at time t,
- D = coefficient of hydrodynamic dispersion at depth z,
- v = groundwater (seepage) velocity at depth z,
- n = porosity of the soil at depth z,
- ρ = dry density of the soil at depth z,
- K_d = distribution/partitioning (sorption) coefficient at depth z,
- v_a = nv = Darcy velocity,
- λ = decay constant of the contaminant species.

Boundary Conditions

The **MIGRATEv9** program solves the two-dimensional contaminant migration equation subject to boundary conditions at the top and bottom of the soil deposit being modelled. There are four

possible top boundary (i.e., the usually the point of contact between the contaminant source and the soil deposit), these are constant surface concentration, finite mass surface concentration, impermeable surface with a buried source, or a zero concentration surface with a buried source. The bottom boundary (i.e., the point of contact between the soil deposit and either a much more or much less permeable strata) may be either zero flux, zero concentration, or an aquifer. These boundary conditions are discussed in detail below and in the **Data Entry section of Chapter 4**.

**Constant
Concentration
Top Boundary**

In this boundary condition the top boundary is assumed to maintain a constant concentration. The concentration at the boundary is given by:

$$c(z=0) = c_s \quad \text{for all } t$$

where c_s is the constant concentration at the top boundary.

**Finite Mass
Top Boundary**

The top boundary may be assumed to have a finite mass, in which case the source concentration starts at an initial value c_o , increases linearly with time at a rate c_r , and then decreases with time as contaminant is transported into the soil and collected by a leachate collection system, if present. The concentration at the top boundary is given by:

$$c(t, z=0) = c_o + c_r t - \frac{1}{H_r} \int_0^t f(c, \tau, z=0) d\tau - \frac{q_c}{H_r} \int_0^t c(\tau, z=0) d\tau$$

where,

- c_o = initial source concentration at the start time,
- c_r = the rate of increase in concentration with time due to the addition of mass to the landfill,
- H_r = the reference height of leachate, and represents the volume of leachate (per unit area of landfill) which would contain the total mass of contaminant at a concentration c_o , $f(c, \tau, z=0)$ = the surface flux (mass per unit area per unit time) passing into the soil at the top boundary,
- q_c = the volume of leachate collected per unit area of the landfill per unit time, if there is no leachate collection system $q_c=0$.

If the rate of increase in concentration c_r is zero and the reference height of leachate H_r is very large, this boundary condition reduces

to a constant concentration boundary condition. The calculation of the reference height of leachate is discussed in the **Data Entry section of Chapter 4** and in **Example 3**.

**Impermeable
Top Boundary**

This boundary condition is used to represent a buried contaminant source, with the surface being, impermeable (i.e. zero flux). The flux across the surface is given by:

$$f(z=0)=0 \text{ for all } t.$$

**Zero Concentration
Top Boundary**

In this boundary condition the source of the contaminants is buried, and the surface is assumed to maintain a constant zero concentration; i.e.

$$c(z=0)=0 \text{ for all } t.$$

Buried contaminants can move upward and be removed at the surface.

**Impermeable
Bottom Boundary**

The bottom boundary may be assumed to not allow any transmission of contaminant, which corresponds to an impermeable base strata. The flux across the boundary is given by:

$$f(z=H_b) = 0 \text{ for all } t$$

where H_b is the depth of the base strata.

**Zero
Concentration
Bottom Boundary**

In this boundary condition the bottom boundary is assumed to maintain a zero concentration. The concentration at the boundary is given by:

$$c(z = H_b) = 0 \text{ for all } t \text{ where } H_b \text{ is the depth of the bottom boundary.}$$

**Aquifer Bottom
Boundary**

The bottom boundary may be specified as an aquifer, where the concentration varies with time as mass is transported into the aquifer from the landfill and transported out from beneath the landfill by the base velocity v_b . Consideration of the conservation of mass gives the base concentration as a function of distance x as:

$$c(x, H_b, t) = \int_0^t \frac{f(x, H_b, \tau)}{n_b h_b} + \frac{v_b \int_0^t [c(x, H_b, \tau) - n_b \int_0^x c(x, H_b, \tau)] dtau}{D_H} \quad \text{where,}$$

- $c(x, z=H_b, \tau)$ = the concentration in the base aquifer,
 $f(x, H_b, \tau)$ = the mass flux into the aquifer from the deposit,
 n_b = porosity of the base aquifer,
 h_b = thickness of the base aquifer,
 v_b = Darcy velocity in the aquifer and down-gradient edge of the landfill,
 D_H = coefficient of hydrodynamic dispersion in the horizontal (x) direction in the aquifer.

Note that the use of a very large base velocity will give the same results at using a constant base concentration of zero. If the base velocity is zero and the porosity is zero the bottom boundary is effectively a zero flux boundary.

Program Use

Before proceeding to use this program the user should read the following information describing the interaction between the user and the program.

This program is designed to be used with either a keyboard or preferably a mouse and a keyboard. When using a mouse, items are selected by clicking the left button, or by double-clicking the left button. The movement of the mouse and use of the buttons can be adjusted using the **Mouse** option in the **Preferences** submenu. Colours on the screen can also be adjusted using the **Colours** option in the **Preferences** submenu. The screen is divided into three sections these are:

- **Menu Line,**
- **Status Line,**
- **Desk Top.**

Menu Line

This is the top line of the screen, and displays the major command areas the user can enter. A particular command area can be selected by clicking on the command with the mouse, or by pressing its short-cut key (the Alt key and its highlighted letter simultaneously).

Each command area has a submenu of commands that are displayed when that command area is selected. The commands on these submenus are selected by clicking on them with the mouse or by using the sliding highlight bar to highlight the selection and then pressing enter. The highlight bar can be moved using the arrow keys or the mouse. Some commands in the submenus also have function keys that can be used to select them; for example, the F2 key can be used to select the Open Data command. These function keys can be used while in the submenu or the main menu areas.

Status Line

The Status line is the bottom line of the screen, and is used to display the currently active short-cut keys. These keys will change

depending on what area of the program the user is currently using. The F1 (Help), F8 (Calculator), and ESC (Cancel) keys are always active anywhere in the program.

Desktop

The desktop is the area of the screen between the menu line and the status line. This is the area where data is passed between the program and the user. Data is exchanged using dialog boxes, which are displayed when commands on the submenus are selected.

Dialog boxes are divided into several data fields, each field has a label and a data element. The user can move between data fields in the dialog boxes by pressing the Tab key, Up and Down Arrow Keys, Enter key or by clicking on the field with the mouse. In addition the dialog boxes contain buttons, typically the OK (F9) and Cancel buttons (ESC). The OK button (F9) will save the current data in the data fields and close the dialog box. Whereas, the Cancel button (ESC) will close the dialog box without saving any changes in the data fields.

There are several types of data fields:

- **Input Lines,**
- **Multiple Choice Boxes,**
- **Single Choice Boxes,**
- **List Boxes,**
- **History List Boxes.**

Input Lines

These data fields consist of a label specifying the data name, and a highlighted box where the data can be entered and displayed. The data is entered and changed using the keyboard in a similar manner to a typewriter, where:

Backspace » deletes the previous letter or digit.

Insert » toggles between insert and overstrike modes.

Delete » deletes the current letter or digit.

Home » puts the cursor at the beginning of the line.

End » puts the cursor at the end of the line. Arrow Keys » move the cursor in the indicated direction.

Enter Data: Text Data

Multiple Choice Boxes

These data fields consist of a data label, and two or more boxes with labels from which the user can choose any number of options. The user selects the options by clicking on them with the mouse, or

by highlighting them with the Tab key and pressing Enter. When selected the option boxes will have an X in them.

-
- Choice 1
 - Choice 2**
 - Choice 3
 - Choice 4**
-

Single Choice Boxes

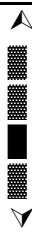
These data fields are similar to multiple choice boxes, except only one option can be selected. If an option is selected and a previous option was selected, the previous option will be deselected. The selected option will have a dot inside the brackets.

-
- Choice 1
 - Choice 2
 - Choice 3**
 - Choice 4
-

List Boxes

These data fields display several values of the data in a box, from which the user can choose one. The desired data value can be selected by clicking on it with the mouse or by highlighting it with the Tab key and pressing the Enter key.

A vertical scroll bar will appear if there are more items to be displayed than can fit in the box. The scroll bar can be used by clicking on it with the mouse, or by using the up and down arrow keys.

-
- Choice 1
 - Choice 2
 - Choice 3
 - Choice 4**
 - Choice 5
 - Choice 6
- 
-

History List Boxes

History List boxes are similar to List Boxes, except they are associated with an **input line**. The history box will be indicated by an arrow icon at the end of an input line. By clicking on the icon the user can display the history list box and select a value to be used in the input line.

Enter Data: **Text Data** ↓

Cancel If you choose Cancel, any changes to the data in the dialog box are not saved and the dialog box is closed.

Cancel

Short-cut keys Short-cut keys are highlighted in red or yellow and are usually the first letter of the command. These commands can be accessed by pressing the ALT key and the highlighted letter simultaneously. When in data entry screens the short-cut keys will take you to that section of data entry.

Some commands also have function keys defined, these are the F1 to F12 keys at the top of the keyboard. When defined these keys are shown to the right of the command and will execute the commands when pressed.

Customizing the Program

MIGRATE-GUI can be completely customized to the user's preferences using the **Environment** option of the **Preferences** submenu. In this option the directories, file extensions, printers, sound, and other options may be specified. In addition, the screen colours and mouse operation may also be customized using the **Colours** and **Mouse** options of the **Preferences** submenu.

The program environment containing the customized features mentioned above is written to a file called MIGRATE.INI in the program directory, this file is invoked when the program is started. It is very important that this file not be destroyed or corrupted, since the MIGRATE-GUI program may not start correctly without the MIGRATE.INI file.

Alt-F

The File Submenu provides access to data and output files, and offers choices to:

- create **New Data** files,
- **Open Data** files,
- **Open Output** files,
- **Save Data** files,
- **Save Data files As** a new name,
- **Save Output As a file**,
- **Change Directories**,
- **Shell to DOS**,
- **Exit** the program.

Data files are used to store the hydrogeologic and engineering information to be used in the MIGRATEv9 program. These files usually have the DOS extension .IN and are in a separate directory from the program files. Output files contain the results of the MIGRATEv9 program. Output files usually have the extension .OU and are in the same directory as the data files.

The directory containing the data and output files is called the **data directory**, this directory is specified in the **Environment** submenu of the **Preferences** menu described later in this chapter.

New Data

The New Data command resets all the defaults for the data, clearing any existing data. If there is any unsaved data the user will be given the opportunity to save the data. When the program is started initially the data is set to the defaults, and it is not necessary to select the New Data command.

Open Data

F2

The Open Data command opens an existing data file. A **File Dialog Box** is displayed, from which the user can specify the name of the data file. Any files with the default input data extension (usually .IN) contained in the **data directory** will be displayed in the file list.

Open Output

F4

The Open Output command opens an existing output file. A **File Dialog Box** is displayed, in which the user can specify the name of the output file. Any files with the default output **data extension** (usually .OU) contained in the **data directory** will be displayed in the **file list box**.

Save Data

F3

The Save Data command saves the hydrogeologic and engineering data in the current data file. If there is no current data file the user is asked to specify a name for the file, the file is then created in the **data directory**.

Save Data As

The Save Data As command saves the current input data in a file specified by the user. A **File Dialog Box** is displayed, in which the user can specify the name of a new file or select the name of an existing file. If an existing file is selected, the data in the file will be overwritten by the new data. Any files with the default **data extension** (usually .IN) contained in the **data directory** will be displayed in the **file list box**.

Save Output As

The Save Output As command saves the output data generated by the MIGRATEv9 program in a file specified by the user. A **File Dialog Box** is displayed, in which the user can specify the name of the output file. Any files with the default **output data extension** (usually .OU) contained in the **data directory** will be displayed in the **file list box**.

Change Dir

The Change Directory command allows the user to change the current **data directory**. A dialog box is displayed consisting of:

- an **input box** for the directory name,
- a **list box** showing the directory tree,
- the standard **(OK)** and **(Help)** buttons,
- a change directory **(Chdir)** button,
- and a **(Revert)** button.

Input Box

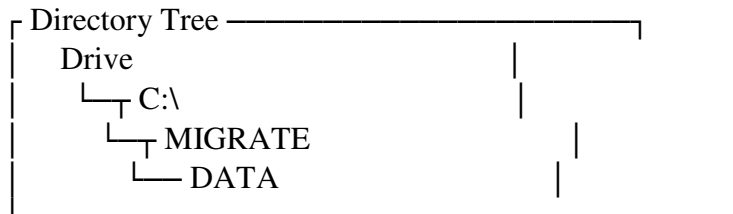
In the directory name input box the user can specify the path of the new **data directory**.

Directory Name _____
c:\MIGRATE\data |

List Box

The directory tree list box enables the user to navigate between directories. If using the keyboard, press Enter to make the selected

directory the current directory, or if using a mouse double-click on the desired directory.



Chdir After a directory has been entered or selected the (Chdir) button changes the **data directory** to this directory.

Chdir

Revert The Revert button goes back to the previous directory, as long as the user has not exited the dialog box.

Revert

OK When the desired directory is selected, pressing the OK button (F9) will switch to that directory and exit the dialog box. Pressing escape will cancel the selection and exit the dialog box.

DOS Shell Using the DOS Shell command, the user can leave the program temporarily to perform DOS commands or run other programs. To return to this program, type EXIT at the DOS prompt.

Exit The Exit command terminates this program. If there is any unsaved input data the user is given the chance to save the data.

Alt - X

File Dialog The File Dialog Box is used to specify or select the names of input or output files. It is composed of:

- a file name **input box** with a **history list**,
- a **files list box** to browse directories,
- a **(Cancel)** button,
- a **(Open)** button,
- and a **information panel**.

Input Box The file name input box is where the name of the input or output file is entered, or a filename mask to use as a filter for the list box (ex. *.*).

File Name: TEST.IN ↓

History List By clicking on the history icon (⇓) next to the input line the user brings up a history list box of previous file names selected. The user can select one of these names by double-clicking on the name, or can exit by pressing escape or clicking on the top left square.

TEST1.IN
TEST2.IN
TEST3.IN
TEST4.IN
TEST5.IN

Files List Box The files list box displays' the names of the files in the current directory that match the filename mask in the file name input box, and the names of the parent directory and all subdirectories. A file can be selected by using the arrow keys to highlight the file and then pressing enter, or by double-clicking on the file name. Files in the parent or subdirectories can be listed by selecting the parent directory (..) or subdirectory.

.. **..MIGRATE**
FILE3.IN | **FILE4.IN**
FILE5.IN | **FILE6.IN**
FILE7.IN | **FILE8.IN**



Information Panel The file information panel shows the path name, file name, date, time, and size of the selected file. None of the items on this information panel is selectable.

C:\MIGRATE\DATA*.IN

Open The Open button reads the selected file and then exits the file dialog box.

OPEN

DATA ENTRY MENU

Alt - D

The Data Entry Menu allows the user to enter hydrogeologic data for the MIGRATEv9 program. There are seven submenus that are used to enter the data:

- **Constant Properties,**
- **Time Varying Properties,**
- **Primary Liner (Subtitle D) Landfill,**
- **Leakage Rate (Subtitle D) Landfill,**
- **Primary and Secondary Liner Landfill**

Constant Properties

This submenu is used to enter models whose properties remain constant with respect to time. In these types of models the only variable that changes with time is the concentration. There are three options that are used to enter and edit model data:

- **General Data,**
- **Boundary Conditions,**
- **Layer Data.**

General Data

The General Data option is used to enter general hydrogeologic data that applies to the entire model. Data entered consists of:

- **Title,**
- **Number of Landfills,**
- **Number of Layers,**
- **Laplace Transform** parameters,
- **Gauss Integration** parameters.

Title

The title of the model is used to describe the model, and may be up to 76 characters long.

Number of Landfills

One or more landfills or other contaminant sources may be modelled. Up to 10 landfills or landfill cells may be modelled.

Number of Layers

The model or deposit is divided into layers where each layer has different parameters. The total number of layers is entered with this parameter. Data for each layer must be subsequently entered with the **Layer Data** command. There is no limit on the number of layers that can be specified in the MIGRATE-GUI program; however, the MIGRATEv9 program limits the total number of sublayers for all layers to 50.

Laplace Transform Parameters

The solution of the contaminant migration equations implemented by the MIGRATEv9 program involves the inversion of a Laplace Transform. In this inversion the accuracy depends upon four parameters: TAU, N, SIG, and RNU.

The user may adopt the default values (TAU=7, SIG=0, N=11, and RNU=1) or specify other values. It has been found that a value of TAU between 7 and 10, and a value of SIG = 0 is satisfactory in most cases. The more critical parameters, RNU and N, typically yield accurate results when:

$$RNU = \frac{0.1 \times \text{total thickness} \times \text{Darcy velocity}}{\text{minimum diffusion coefficient}}$$

and

$$N = 10 \times RNU \quad \text{if } RNU \text{ is greater than } 1.0.$$

These values of RNU and N will generally work extremely well but will often require more integration than is necessary. If the computation times seem excessive smaller values of RNU and N should be tried, and the accuracy of the results compared.

Gauss Integration Parameters

The theoretical development involves the use of a Fourier transform and hence the synthesis of the final solution involves some numerical integration. These integrations are performed numerically by decomposing the integration range into a number of subintervals and then using Gauss quadrature within each subinterval. The width and number of integration subintervals which are required to achieve a reasonable accuracy (say 0.1%) depends somewhat on the geometry and properties of the problem under consideration. These parameters can be determined from a few trial calculations for a representative point and time of interest.

The program is capable of performing four levels of Gauss quadrature, FINE, NORMAL, WIDE, and SELECT. The computation time is directly proportional to the amount of integration.

When SELECT is specified the user must specify the integration

step size, number of points, and number of steps (Figure 4.1). The user will be prompted for these parameters after the OK button is pressed.

Boundary Conditions

For every model there are two boundary conditions, one at the top and one at the bottom. The top boundary condition is usually the point of contact between the contaminant source (eg. a landfill) and the subsurface layers (model), and can be either at the surface or buried. There are four possible top boundary conditions:

- **Constant Concentration at the surface,**
- **Finite Mass at the surface,**
- **Zero concentration at the surface with buried mass,**
- **Impermeable surface with buried mass,**

The bottom boundary condition is usually the point of contact between the layered deposit and either a much more or much less permeable strata (eg. an aquifer or intact bedrock) and can be either:

- **Impermeable,**
- **Zero Concentration, or**
- **An Aquifer,**

If more than one landfill has been specified, the data for the top boundary condition must be specified for each landfill (or other contaminant source). After the data for each landfill has been entered the user can go to back to the previous landfill by selecting the

Although more than one landfill may be specified as the source for the contaminants, it should be noted that the type of boundary condition must be the same for all the landfills. This means that if the top boundary condition is specified as finite mass, then all the landfills must be finite mass. In addition, it should be noted that since the liner of a landfill is modelled as a layer, all of the landfills must have the same type of liner (i.e., it is not possible to model a landfill with a geomembrane and a landfill without a geomembrane together).

Constant Concentration at Surface

The constant concentration at the surface top boundary condition represents the case where the concentration of contaminant in the landfill remains constant throughout time, and is equivalent to the assumption of an infinite mass of contaminant in the landfill. The user will be asked to specify:

- **Offset Distance,**

- **Landfill Base Width,**
- **Landfill Surface Width,**
- **Surface Concentration.**

Offset Distance	This is the distance from the co-ordinate origin on the x-axis (horizontal) to the centre of the landfill (see Figure 1.3).
Landfill Base Width:	The landfill may have a greater width at the surface than at the base, to take into consideration the effect of side-slopes. This is the width of the landfill at the base, in the direction of groundwater flow.
Landfill Surface Width:	This is the width of the ground surface or top of the landfill, in the direction of groundwater flow.
Surface Concentration	This is the concentration of the contaminant species in the landfill.

Finite Mass at Surface

The finite mass at the surface top boundary condition is most representative boundary condition for most landfills. The concentration of contaminant may start at an initial value, increases linearly with time, and then declines as contaminant is transported into the deposit and is removed by leachate collection systems. The user will be asked to specify:

- **Offset Distance,**
- **Landfill Base Width,**
- **Landfill Surface Width,**
- **Source Half-Life,**
- **Initial Concentration,**
- **Waste Thickness,**
- **Waste Density,**
- **Percentage of Mass,**
- **Volume of Leachate Collected.**

Source Half-Life If the contaminant species experiences radioactive or biological

decay while in the landfill, the first-order decay half-life may be specified. If there is no decay specify a half-life of zero (which is taken to be infinite).

Initial Source
Concentration

This is the initial concentration of the source of contaminants, usually at time zero.

Waste Thickness

This is the average thickness of waste in the landfill (or other contaminant source). If the effects of the side-slopes of the landfill are being considered by using different base and surface widths, then the waste thickness specified here should not consider these effects.

Waste Density

This is the average density of the waste.

Percentage of Mass

The contaminant species may represent only a small portion of the total mass of the waste. This percentage is specified here, and should represent only the leachable portion of the contaminant species.

Volume of Leachate

This is the volume of leachate collected per unit area of landfill per unit time, usually by the leachate collection system. Thus, after the landfill reaches field capacity the average volume of leachate collected is equal to the average infiltration through the landfill cover less the average exfiltration through the base of the landfill (assuming the waste is at field capacity).

For example, if the average infiltration is 0.3 m/a and the average exfiltration is 0.03 m/a, then the average volume of leachate collected is $0.3 - 0.03 = 0.27$ m/a.

**Zero Concentration
and Buried Mass**

This boundary condition can be used to model a buried contaminant source. Using this boundary condition, contaminants can move upwards and downwards from the source. At the surface the concentration is assumed to be zero, which would simulate the removal of contaminants at the surface by runoff, collection systems, or other means. This boundary condition assumes that the dispersion coefficient in the soil adjacent to the buried waste, is the same as within the buried waste.

The user is required to specify:

- **Offset Distance,**
- **Landfill Base Width,**
- **Landfill Surface Width.**

**Impermeable
Surface and Buried
Mass**

This boundary conditions is similar to the zero surface concentration and buried mass condition, except in this case the surface is assumed to represent an impermeable layer, (i.e. zero-flux). The user is required to specify:

- **Offset Distance,**
- **Landfill Base Width,**
- **Landfill Surface Width.**

**Impermeable
Bottom**

The impermeable bottom boundary condition represents the case where no mass is transported into or out of the bottom of the deposit. This condition can be used to represent the case of a deposit underlain by an impermeable base stratum (e.g., intact bedrock that is impermeable relative to the overlying layer or deposit). The use is required to specify:

- **Base Integration Width**

**Base Integration
Width**

The base integration width is used to determine the flux of contaminants into and out of the base. This integration width is measured in the same direction as the landfill width, which should be in the direction of groundwater flow. Ideally the base integration width should be infinite; however, to limit computing time typical values equal to six times the landfill base width may be used.

**Zero Concentration
Bottom**

The zero concentration bottom boundary condition represents the case where the concentration of contaminant remains zero in the base strata. The user will be prompted to specify the:

- **Base Integration Width.**

**Aquifer
Bottom**

The aquifer boundary condition represents the case where the deposit is underlain by a permeable base strata. The concentration in the base strata (aquifer) varies with time as mass is transported into the aquifer from the deposit, and then transported away by the horizontal velocity and dispersion in the base strata.

The base aquifer is modelled as a boundary condition (not a separate layer) and the concentration at the bottom of the deposit is the concentration at the top of the base aquifer. This boundary condition assumes that there is sufficient dispersion/mixing (i.e. a very high vertical dispersion coefficient) such that the concentration is uniform across the thickness of the aquifer being

considered. Thus the concentration at the bottom of the aquifer thickness modelled is the same as reported at the top of the aquifer. If the actual aquifer is very thick, normally only the upper portion (3 - 6 m depending on conditions) should be considered in modelling. Alternatively, the aquifer could be modelled as an additional layer. This approach allows the consideration of vertical dispersion\diffusion, and may give a vertical diffusion profile through the aquifer's thickness. With this boundary condition the user will be asked to specify:

- **Base Integration Width,**
- **Base Thickness,**
- **Base Half-Life,**
- **Sink Removal,**
- **Base Porosity,**
- **Outflow Velocity,**
- **Dispersion Coefficient.**

Base Thickness	This is the vertical thickness of the base strata that is being modelled as a boundary condition.
Base Half-Life	This is the first-order decay half-life of the contaminant species in the aquifer. If the contaminant species does not experience radioactive or biological decay, a half-life of zero should be specified to imply infinite half-life.
Sink Removal	This is used to specify the rate of removal of contaminants from the aquifer by a passive sink (ex. a purge well system). For a more detailed discussion see Sink Removal Rate under Layer Properties.
Base Porosity	This is the porosity of the base strata, between 0 and 1.
Outflow Velocity	This is the horizontal Darcy outflow velocity within the base strata at the down-gradient edge of the landfill. If the outflow velocity is set very high the results will be equivalent to setting a constant base concentration of zero.
Dispersion Coefficient	This is the horizontal Coefficient of Hydrodynamic Dispersion in the aquifer.

Layer Data

The data for each individual layer is entered in this option, the data for all the **number of layers** that was specified in the **General Data** command should be input.

Continuity of concentration and flux is assumed at the boundary

between layers. If a fractured layer is in contact with an unfractured layer, it is assumed that all fluid flow is transported along the fractures that intersect the unfractured layers (i.e., it is equivalent to having a very thin sand layer between unfractured and fractured layers).

After the data for the layer is entered, the user can go to the next layer by selecting the Next button (F5), or go back to the previous layer by selecting the Previous button (F6). When the data for all the layers has been entered, the user can save the information by selecting the OK (F9) button.

The data required for each layer consists of:

- the number of **sublayers**,
- **thickness**,
- **dry density**,
- **porosity**,
- **Vertical & Horizontal Diffusion Coefficients**,
- **Vertical & Horizontal Darcy Velocities**,
- **distribution coefficient**,
- **Half-Life**,
- **Sink Removal Rate**,
- presence and type of **fractures**.

If the top boundary condition is buried mass the user can also specify an initial concentration for the layer, to represent this mass.

Sublayers The number of sublayers in each layer is primarily used in the output of the calculated concentrations with depth; a concentration will be calculated at each sublayer interface.

The maximum thickness of each sublayer is 5 units. The number of sublayers is automatically increased if required to keep their thickness to less than 5. For example, if the layer thickness was 50 m and it had 5 sublayers (giving a sublayer thickness of 10), the program will automatically adjust the number of sublayers to 10.

If the time-varying properties is selected, the accuracy of the results will depend on the number of sublayers.

Thickness The thickness of the layer must be specified, this is the total thickness of all the **sublayers** in the layer.

Dry Density The dry density of the layer, expressed in consistent units with those used throughout the model.

Porosity This is the porosity of the layer, which must be greater than 0 and less than or equal to 1. If the layer is being used to represent a

geomembrane the porosity should be set to 1.

Horizontal & Vertical Diffusion/ Dispersion Coefficients

These are the horizontal and vertical coefficients of hydrodynamic dispersion for the layer:

$$\begin{aligned} D_{xx} &= D_{ex} + D_{mdx} \\ D_{zz} &= D_{ez} + D_{mdz} \end{aligned}$$

where,

D_{ex} and D_{ez} = diffusion coefficients for the species,

D_{mdx} and D_{mdz} = coefficients of mechanical dispersion.

For intact clayey layers, diffusion will usually be the controlling factor and dispersion will often be negligible [Gillham and Cherry, 1982, Rowe, 1987]. In sandy layers, dispersion will tend to be the controlling factor.

Horizontal and Vertical Darcy Velocities

The horizontal and vertical velocities are defined as:

$$\begin{aligned} V_{ax} &= nv_x \\ V_{az} &= nv_z \end{aligned}$$

where,

n = effective porosity,

v_x and v_z = seepage or groundwater velocities.

Distribution Coefficient

This is the distribution coefficient for the layer. The sorption-desorption of a non-conservative species of contaminant is assumed to be linear such that:

$$S = K_d * c$$

where,

S = solute sorbed per unit weight of soil,

K_d = distribution (partitioning) coefficient,

c = concentration of contaminant.

This is a reasonable approximation for low concentrations of contaminant, however at high concentrations sorption may not be linear and more complex relationships should be used. If there is no sorption (i.e., a conservative species) the distribution coefficient is zero.

Half-Life

If the contaminant species experiences radioactive or biological decay in a layer, this option can be used to specify the half-life of the species in the layer. First order (exponential) decay is assumed for both radioactive and biological decay, eg.

$$c(t) = c(0) e^{-\lambda t}$$

where,

$c(t)$ = concentration at time t ,
 $c(0)$ = initial concentration,
 λ = decay constant = 0.693147/half-life,

If there is no decay of the species in the layer, the half-life should be specified as zero which is the same as an infinite half-life (i.e. $\lambda = 0$).

Sink Removal Rate

This is used to specify the rate of removal of contaminants from a layer by a passive sink, (for example, a secondary leachate collection system or purge wells). According to the principle of continuity of flow the rate of removal should be equal to:

$$R_r = (v_{az1} - v_{az2}) \frac{L}{h} - v_{ax}$$

where,

R_r = rate of removal (flow per unit area per unit time)
 v_{az1} = vertical Darcy velocity in the layer above,
 v_{az2} = vertical Darcy velocity in the layer below,
 v_{ax} = horizontal Darcy velocity in the layer with the passive sink,
 L = Width of the base of the landfill,
 h = thickness of the layer with the passive sink.

An example would be a 600 m long landfill with a 0.3 m thick secondary leachate, collection system. Assuming the vertical Darcy velocity above the layer is 0.01 m/a and below the layer is 0.003 m/a, then the rate of removal would be:

$$R_r = (0.01 - 0.003) \times 600/0.3 = 14 \text{ m/a.}$$

Fractures

Any or all of the layers may be fractured. These fractures may be 1, 2, or 3 dimensional. Where the first dimension is for one set of vertical fractures, the second is for a second set of (orthogonal) vertical fractures, and the third is for horizontal fractures (i.e. for a 3D block, dimension 1 is length, dimension 2 is width, and dimension 3 is depth). The user will be asked for the fracture parameters after selecting the OK (**F9**) or the Next (**F5**) buttons in layer entry.



In a fractured deposit the program can consider advective-dispersive transport along the fractures coupled with diffusion into the matrix on either side of the fracture. However, if the **Darcy velocity** is zero, or small, then the transport mechanism will be essentially diffusive through the matrix, the fractures will have no effect and should not be considered in modelling the migration of contaminants. Users planning to model migration in fractured media are warned that they should first see Rowe and

Booker, 1990, 1991a, 1991b, and Rowe et al, 1994 for a discussion of modelling of fractured systems.

The user can specify the following information about the fracture in each dimension:

- fracture **Spacing**,
- fracture **Opening Size**,
- and **number to sum**.

In addition the user can specify the fracture's:

- **Dispersion coefficient**,
- **Distribution coefficient**.

Spacing	The spacing of fractures is the distance between fractures in each dimension.
Opening Size	The fracture opening size is the typical width (gap) between the fracture walls.
Number to Sum	Is the number of terms to sum in the evaluation of the advective-dispersive equation for contaminant migration [Rowe and Booker, 1990, 1991a, 1991b]. For blocks where the fracture spacing is of the same order in all directions, 8 to 10 terms is usually adequate. As the aspect ratio (horizontal spacing/vertical spacing or vertical spacing/vertical spacing) increases more terms are required in the summation. When the aspect ratio is large, the problem can usually be reduced to a problem of lower order (eg. from 3D to 2D or 2D to 1D). For example, if the spacing between fractures in one vertical direction is 50 units, and in the other vertical and horizontal directions is 2 units. The widely spaced fractures can be ignored and the problem reduced to a 2D problem [Rowe and Booker, 1990].
Dispersion Coefficient	This is the dispersion coefficient along the fracture. For a more complete description of dispersion coefficient see the Diffusion/Dispersion Coefficient for a layer.
Distribution Coefficient	This is the distribution coefficient along the fracture as defined by Freeze and Cherry (1979). This is often assumed to be zero.

Time Varying Properties

This submenu is used for models whose properties, other than concentration, vary with time. It is used to model situations where there may be a complex source history (eg. the expansion of an existing landfill), or changes in the properties and velocities of the layers with time (eg. the progressive failure of a leachate collection system).



The Time-Varying Properties option implements a "time-marching" scheme, where the program stops and restarts the solution every time parameters are changed. In the Constant Properties option the accuracy of the solution is independent of the number of sublayers. However, in the Time-Varying Properties option the accuracy is dependent on the number of **sublayers** used. The user is warned that they should experiment with the number of sublayers to ensure that the results obtained are sufficiently accurate.

There are four options that are used to enter and edit data for time-varying properties models:

- **General Data,**
- **Boundary Conditions,**
- **Layer Geometry,**
- **Time Period Data.**

General Data

This option is used to enter general hydrogeologic data for the model, and is similar to the General Data option of the Constant Properties submenu. The user can specify:

- **the Title,**
- **Number of Time Groups,**
- **Number of Layers,**
- **Start Time,**
- **Laplace Transform Parameters,**
- **Gauss Integration Parameters.**

Except for the Number of Time Groups and Start Time, all of these parameters are described in the General Data option of the Constant Properties submenu. The number of landfills is not specified here, since it can change between Time Groups.

Number of Time Groups

This is the number of time periods that have different properties.

Start Time This is the start time of the first time period, and is the time that calculations begin (usually zero).

Boundary Conditions

As in the Constant Properties model, a top and bottom boundary condition must be specified. These boundary conditions will apply for all the time groups. Although the type of boundary condition must be the same for all time groups (eg. whether it is constant concentration or finite mass), the parameters of the boundary condition can change for each time group.

The types of boundary conditions that can be selected are identical to those in the Constant Properties submenu. All of the parameters, except for the base integration width, and aquifer thickness are specified separately for each time group in the Time Period Data option.

Layer Geometry

Except for the number of sublayers and the layer thickness, all of the properties of the layers can be varied with time. The time varying properties are specified in the Time Period Data option, and the user is required to specify the number of sublayers and thickness in this option. After these two parameters have been specified for the layers, the next layer can be selected by pressing the Next Button. Once all of the layers have been entered, the OK Button should be selected.

Time Period Data

All of the properties that vary with time can be specified in this option. These properties include data for the time group, layer data for the group, and boundary condition data for the group. Data for the time group consists of:

- **End Time,**
- **Number of Increments,**
- **Number of Landfills.**

These parameters are discussed below, after they have been entered the layer data for the time period should be entered.

This data consists of:

- **Dry Density,**
- **Porosity,**
- **Vertical and Horizontal Diffusion Coefficients,**
- **Vertical and Horizontal Darcy Velocities,**
- **Distribution Coefficient,**
- **Half-Life,**
- **Sink Removal Rate.**

And if the top boundary is buried mass the initial concentration for this mass can be specified.

After the properties for a layer have been entered, the next layer can be selected by pressing the Next button beside the layer data. Properties for all of the layers present should be specified for each time group. When a model is initially created the properties of the layers in the previous time group will be used as defaults for the second and higher time groups.

When all of the data for the layers has been entered, the data for the top boundary should be specified. This top boundary may consist of one or more landfills (contaminant sources), as specified earlier using the Number of Landfills parameter. The data that can be entered will depend on the type of top boundary that was selected, and will consist of:

- **Landfill Offset,**
- **Landfill Base Width,**
- **Landfill Surface Width.**

And if the top boundary is a constant concentration surface source:

- **Surface Concentration.**

Or if the top boundary is finite mass:

- **Source Half-Life,**
- **Initial Concentration,**
- **Waste Thickness,**
- **Waste Density,**
- **Percentage of Mass,**
- **Volume of Leachate Collected.**

When the data for a landfill has been entered, the next landfill can be selected by pressing the Next button beside the landfill data. When there is more than one landfill, the Number of Landfills must be specified before the data for the landfills is entered.

After the data for the top boundary of the time period is entered, the parameters for the bottom boundary should be specified. If the bottom boundary is impermeable or zero concentration, the only parameter to be specified is the base integration width which is specified in the Boundary Conditions option. If the bottom boundary condition is an aquifer the user will need to specify:

- **Base Half-Life,**

- **Sink Removal Rate,**
- **Base Porosity,**
- **Outflow Velocity,**
- **Dispersion Coefficient.**

These parameters are described in the Constant Properties Boundary Condition section.

When all the data for a time group has been entered, the next time group can be selected by pressing the large Next button in the top right of the screen. After all of the time groups have been selected. In this option the data will be saved whenever a Next, Previous, or OK button is selected.



WARNING: This option should only be used by someone with the hydrogeologic and engineering background necessary to appreciate the subtleties associated with the physical situation and the steps necessary for appropriate modelling of the physical situation.

THIS OPTION SHOULD NOT BE USED FOR A PROJECT OF IMPORTANCE WITHOUT THE GUIDANCE OF THE PROGRAM DEVELOPERS.

End Time	This is the end time of the group, and will be the start time of the next group.
Number of Increments	This is the number of increments to use in sub-dividing the time period. The concentrations will be calculated for the times at the end of each increment, if there is only one increment specified for the group the time will be calculated at the end time. For example, if the group started at 0 years and ended at 20 years and the number of increments was 4, concentrations would be calculated at 5, 10, 15, and 20 years.

**Primary Liner
(Subtitle D)
Landfill**

The Primary Liner (Subtitle D) Landfill submenu is used to quickly enter a landfill that may contain a leachate collection system, primary composite liner, aquitard, and aquifer. In this option the primary composite liner can be composed of a geomembrane and a primary liner. If the geomembrane is present an equivalent hydraulic conductivity must be specified. Another method of specifying this type of landfill uses a calculated leakage rate through defects in the geomembrane, the user should see **Leakage Rate (Subtitle D) Landfill** to use this option.

The user will be asked to select which layers are present from the following;

- **Primary Geomembrane,**
- **Primary Liner ,**
- **Aquitard,**
- **Aquifer.**

After the above data has been entered the user will be required to enter the **source data, head data,** and **layer data** for the layers selected (eg. geomembranes, liners, etc.).



Note: When this submenu is used data is converted to suitable input to MIGRATEv9, this conversion process is one way. The user should be careful not to corrupt the data entered in this submenu, by entering any subsequent data in **the Deposit Data, Layer Data,** and **Boundary Conditions** submenus. If data is entered in these other submenus the user should check for consistency.

Leakage Rate (Subtitle D) Landfill

The Leakage Rate (Subtitle D) Landfill submenu is used to quickly enter data for a landfill that contains a leachate collection system, primary composite liner, and aquifer. This submenu uses a leakage rate calculated according to the method proposed by Giroud et. al. (1992), to determine the Darcy velocity through the composite liner. In this method the leakage rate through a composite liner will depend on the type and number of defects in the geomembrane, the leachate head on the liner, and the thickness and hydraulic conductivity of the compacted clay or geosynthetic clay liners (see Example 1). In addition the leakage rate due to permeation can also be considered, and is calculated according to the method proposed by Giroud and Bonaparte (1989).

The user will be asked for the following information;

- Contact (Good or Poor) between the Geomembrane and Compacted Clay or GCL,
- Type of holes in the Geomembrane (circular or long and narrow).
- Whether to consider Permeation.

After pressing the OK button if the holes are circular the user will be required to enter the frequency and area of the holes, or if the

holes are long and narrow the user will be required to specify the frequency, length, and width of the holes.

When the above data has been entered the user will be required to enter the **source data**, **head data**, **geomembrane data**, **primary liner data**, and **aquifer data**.

Permeation through the geomembrane can be considered by selecting Yes. The leakage rate due to permeation will be added to the leakage rate due to defects.

According to Giroud and Bonaparte (1989) the leakage rate due to permeation (Q_g) is equal to the coefficient of migration (M_g) divided by the geomembrane thickness (T_g), i.e. $Q_g = M_g / T_g$. The coefficient of migration (M_g) may be calculated using the following equations:

$$M_g = C1 \times \underline{P}^n \quad \text{if } \underline{P} < \underline{P} \text{ plateau}$$

$$M_g = M_{gmax} \quad \text{if } \underline{P} > \underline{P} \text{ plateau}$$

where, C1 is a coefficient (in these calculations assumed to be $1 \times 10^{-22} \text{ m}^4 \text{Kg}^{-2} \text{S}^3$), \underline{P} is the pressure difference (with \underline{P} plateau assumed to be 55 kPa), n is a dimensionless coefficient (in these calculations assumed to be 2), and M_{gmax} is assumed to be $3 \times 10^{-13} \text{ m}^2/\text{s}$.

The pressure difference (\underline{P}) can be obtained using the relationship:

$$\underline{P} = \underline{h} \times \underline{C} \times g$$

where \underline{h} is the head difference across the geomembrane (which in these calculations is assumed to be the depth of water or leachate on top of the geomembrane), \underline{C} is the density of the leachate or water (assumed to be water in these calculations), and g is the acceleration due to gravity.



Note: When this submenu is used data is converted to suitable input to MIGRATEv9, this conversion process is one way. The user should be careful not to corrupt the data entered in this submenu, by entering any subsequent data in the **Deposit Data**, **Layer Data**, and **Boundary Conditions** submenus. If data is entered in these other submenus the user should check for

consistency.

CAUTION: The user is cautioned to need the above reference and examine the underlying assumptions and limitations; before using this option.

Primary and Secondary Liner Landfill

The Primary and Secondary Liner Landfill submenu is used to quickly enter a landfill that may contain a primary leachate collection system, primary composite liner, secondary leachate collection system, secondary composite liner, aquitard, and aquifer. In this option the composite liners can be composed of a geomembrane and/or compacted clay or GCL. If a geomembrane is present an equivalent hydraulic conductivity must be specified.

The user will be asked to select which layers are present from the following;

- Primary **Geomembrane**,
- Primary **Liner** ,
- Secondary **Geomembrane**,
- Secondary **Liner** ,
- **Aquitard**,
- **Aquifer**.

A secondary leachate collection system is always assumed to exist in this landfill model. In addition the units and title need to be specified. The units can either metric (SI) or US, these units are used for defaults and when converting the data for the MIGRATEv9 program.

After the above data has been entered the user will be required to enter the **source data**, **head data**, and data on the layers selected.



Note: When this submenu is used data is converted to suitable input to MIGRATEv9, this conversion process is one way. The user should be careful not to corrupt the data entered in this submenu, by entering any subsequent data in **the Deposit Data**,

Layer Data, and **Boundary Conditions** submenus. If data is entered in these other submenus the user should check for consistency.

Source Data

This dialog box is used to specify the source data for the quick entry landfills. The user is required to enter the following data:

- Initial Concentration,
- Landfill Base Width,
- Landfill Surface Width,
- Source type (Constant Concentration or Finite Mass).

If the source type is **Finite Mass** (see Finite Mass Top Boundary under Boundary Conditions for more information) the user will be required to enter the following data:

- Waste Thickness,
- Infiltration (percolation) through the cover ,
- Waste Density,
- Percentage of Waste Mass that the contaminant represents.

Head Data

To calculate the Darcy velocity through the liners the hydraulic heads must be specified. In the **Primary Liner (Subtitle D) Landfill** the user must specify:

- the leachate head above the primary liner,
- the groundwater level in the aquifer relative to the top of the aquifer, or if no aquifer is present the hydraulic gradient in the liner.

For the **Leakage Rate (Subtitle D) Landfill** the head in the aquifer is assumed to be below the base of the liner. The user must specify:

- the leachate head above the primary liner.

And in the **Primary and Secondary Liner Landfill** the user must specify:

- the leachate head above the primary liner,
- the groundwater level in the aquifer relative to the top of the aquifer, or if no aquifer is present the hydraulic gradient through the liner,
- the leachate head on the secondary liner.

Geomembrane Data

The data required for a geomembrane depends on the type of landfill being modeled. For **Primary Liner (Subtitle D) Landfills** and **Primary and Secondary Liner Landfills** the user is required to specify:

- Thickness,
- Diffusion coefficient,
- Equivalent hydraulic conductivity.

In the **Leakage Rate (Subtitle D) Landfill** the user is required to specify:

- Thickness,
- Diffusion coefficient.

A more through description of the diffusion coefficient can be found in the **Layer Data** section of this chapter. The equivalent hydraulic conductivity of a geomembrane is used to take into account the effects of defects in the geomembrane.

Liner Data

The liner can be either compacted clay or a geosynthetic clay liner (GCL). For each liner present the user must specify:

- Thickness,
- Density,
- Hydraulic Conductivity,
- Vertical Diffusion coefficient,
- Horizontal Diffusion coefficient,
- Distribution coefficient,
- Porosity.

For a description of the thickness, density, diffusion coefficient, distribution coefficient and porosity see the **Layer Data** section of this chapter.

Secondary Leachate Collection System Data

In the **Primary and Secondary Liner Landfill** a secondary leachate collection system must be present. This layer is usually a granular layer, and the user must specify the following:

- Thickness,
- Density,
- Vertical Dispersion coefficient,
- Horizontal Dispersion coefficient
- Distribution coefficient,
- Porosity.

For a description of the thickness, density, dispersion coefficient,

distribution coefficient and porosity see the **Layer Data** section of this chapter.

Aquitard Data

If a aquitard is present below the liners the user must specify the following:

- Thickness,
- Density,
- Hydraulic Conductivity,
- Vertical Diffusion coefficient,
- Horizontal Diffusion Coefficient
- Distribution coefficient,
- Porosity.

For a description of the thickness, density, diffusion coefficient, distribution coefficient and porosity see the **Layer Data** section of this chapter.

Aquifer Data

If a aquifer is present the user must specify the following:

- Thickness,
- Porosity.

After this information has been entered the minimum horizontal outflow velocity in the aquifer will be calculated and the user will have the option of using this velocity or a higher velocity. A more complete description of the outflow velocity can be found in the **Aquifer** bottom boundary section of this chapter.

EXECUTE MENU

Alt - E

The Execute submenu allows the user to execute the MIGRATEv9 program using the data file that has been created. Prior to execution the data file should have been created using **Data Entry** or read using **Open Data**.

Concentrations

This option allows the user to calculate contaminant concentrations at selected distances and times, using the previously created input data. In addition, the total mass/unit area into the soil and into the base will also be calculated at the selected times. The user will be asked to specify:

- **Number of Distances** and the distances
- **Number of Times** and the times,
- **Graphic File** generation,
- **Execution Mode**.

After entry of the above parameters and options, the program will:

- (1) save the input data to a file and if no previous file name has been specified the user is asked for a data file name,
- (2) ask the user to specify an output file name,
- (3) if a graphic file has been requested, it will ask the user for a graphic file name,
- (4) if batch mode has been requested, it will ask the user for a batch file name,
- (5) if interactive mode has been requested, it will run the MIGRATEv9 program,
- (6) it will then return to the main menu.

Number of Distances

This is the number of distances at which to calculate the concentration of contaminant. The distances should be entered in ascending order. The user can use the Next (F5) and Previous (F6) buttons to switch between distances.

Number of Times This option allows the user to specify the number of times at which to calculate the concentration of contaminant. The times should be entered in ascending order. The user can use the Next (F5) and Previous (F6) buttons to switch between times.

Graphic File This option allows the user to choose whether to have the MIGRATEv9 program generate a graphic file, it is provided primarily for computability with previous versions. This file can be used to create plots of the concentration in other programs, it is NOT required by the **Output** submenu options. The user will be asked to specify the Graphic File Name.

The Graphic File Name is the name for the graphic file to be generated by the MIGRATEv9 program. A **file dialog box** is displayed, in which the user can specify the name of the file. Any files with the default **graphics data extension** contained in the **data directory** will be displayed in the list box.

Execution Mode This is the mode of execution of the MIGRATEv9 program that the user wishes to use. The user can specify either interactive or batch mode. If the user selects interactive mode, the MIGRATEv9 program will be executed after all the necessary parameters are entered. Upon completion of the MIGRATEv9 program the output file will be loaded.

If the user selects batch mode, the user will be asked to specify the **Batch File** Name. This file can later be used to run the MIGRATEv9 program by typing the batch file name at the DOS prompt. If the batch file already exists, the user will have the option of appending this MIGRATEv9 run to the file. By appending the run, the user can build a batch file that will run the MIGRATEv9 program several times with different input and output files.

Batch File This is the name for the batch file to be used later to run the MIGRATEv9 program. A **file dialog box** is displayed, in which the user can specify the name of the file. Any files with the default **batch data extension** contained in the **data directory** will be displayed in the list box.

CHAPTER 6

OUTPUT MENU

Alt - O

The Output submenu gives the user the option of displaying and then **printing** the results of the MIGRATEv9 program. The user can choose between the following types of output:

- **Concentration versus Time,**
- **Concentration versus Depth,**
- **Concentration versus Distance,**
- **Colour Concentrations versus Time and Depth,**
- **Colour Concentrations versus Distance and Depth,**
- **Listing,**
- **Lotus 123 file.**

Printing

After the graph has been displayed on the screen the user can choose to continue by pressing 'c' or to print the graph by pressing 'p'. When selected the user will be asked to specify:

- **Titles,**
- **Title Character Size,**
- **Title Font,**
- **Title Justification,**
- **Title Position,**
- **Axes Fonts, • Axes Character Sizes,**
- **Axes Lengths.**

The user can also specify new **printer setup** parameters as described under **Printer Setup** of the **Preferences** menu. Any printer parameters changed will only be active until the MIGRATE-GUI program is terminated, and will not be saved for the next session. To have the printer setup parameters saved the parameters should be changed in the Preferences menu.

After the OK button is pressed if the printer is Postscript and the printer port is file or if the printer is in Desktop Publishing PCX format, the user will be asked for the name of the file to store the output graph.

When all of the parameters have been entered a status box will be displayed indicating the progress of the plot. The printing can be aborted at any time by pressing the ESC button.

Titles	The plot can have one or two titles, each title can be up to 90 characters long. The default titles are taken from the output file.
Title Font	This is the font used to print the titles. The user has a choice between small, sans serif, or triplex.
Title Character Size	This is the character size for the titles, it is recommended that it be as large or larger than that for the axes. The characters can be small, medium, or large.
Title Justification	This controls the justification of the titles, they can be either left, center, or right justified.
Title Position	The titles can be printed on the top or bottom of the graph.
Axis Fonts	This is the font to use for the axes' titles and labels. The user has a choice between small, sans serif, or triplex.
Axis Character Sizes	This is the character size for the axes' titles and labels. The characters can be small, medium, or large.
Axis Lengths	This is the length of the axes in inches, it is recommended the lengths be between 2 and 6 inches.
Concentration versus Time	This option will generate a Concentration versus Time graph using the MIGRATEv9 output file. The graph will have time on the horizontal axis and concentration on the vertical axis. The user will be asked to specify the following;

-
- **Minimum Values,**

- **Maximum Values,**
- **Label Intervals,**
- **Axis Titles,**
- **Show Maximum Value,**
- **Show Grid,**
- **Depths,**
- **Distances.**

Minimum Value	This is the minimum value to be used for the axis in generating the graph, and will appear on the bottom left side of the graph. The program will automatically calculate a default for this value
Maximum Value	This is the maximum value to be used for the axis in generating the graph, and will appear on the top or right side of the graph. The program will automatically calculate a default for this value.
Label Interval	This is the label interval to be used for the axis in generating the graph, if a grid is displayed it is also the grid interval. The program will automatically calculate a default for this value.
Axis Titles	This is the axis title to be used in generating the graph, and will appear on the bottom or left side of the graph. The program will automatically calculate a default for this value. The title can have a maximum of 30 characters.
Show Maximum Value	This option allows the user to display the Maximum value of the concentration appearing in the graph. The program will display the maximum and the time or depth of the maximum at the top of the graph.
Show Grid	This option allows the user to display a grid on the graph, with the grid interval being the same as the label interval.
Depths	A list box is displayed containing the depths for which the concentration has been calculated. The user can choose to display all the depths on the graph or only one depth.

Distance

A list box is displayed contains the distances for which the concentration has been calculated. The user can choose to display all the distance on the graph or only one distance.

Concentration versus Depth

This option will generate a Concentration versus Depth graph using the MIGRATEv9 output file. The graph will have Concentration on the horizontal axis and Depth on the vertical axis. The user will be asked to specify the following;

- **Minimum Values,**
- **Maximum Values,**
- **Label Intervals,**
- **Axis Titles,**
- **Show Maximum Value,**
- **Show Grid,**
- **Times,**
- **Distances.**

All of the above are described under **Concentration versus Time**, except for **Times** which is described below.

Times

A list box is displayed containing the times for which the concentration has been calculated. The user can select to display all the times on the graph, or only one time.

Concentration versus Distance

This option will generate a Concentration versus Distance graph using the MIGRATEv9 output file. The graph will have Distance on the horizontal axis and concentration on the vertical axis. The user will be asked to specify the following;

- **Minimum Values,**
- **Maximum Values, • Label Intervals,**
- **Axis Titles,**
- **Show Grid,**
- **Times.**

All of the above are described previously.

Colour Concentrations versus Depth and Time

This option will generate a Colour Concentration graph using the MIGRATEv9 output file. The graph will have Time on the horizontal axis and Depth on the vertical axis. Colours will be used to represent the values of Concentration. The user will be asked to specify the following (which are described under **Concentration versus Time**);

- **Minimum Values,**
- **Maximum Values,**
- **Label Intervals,**
- **Axis Titles,**
- **Distances.**

In addition, the user will need to specify the following:

- **Home Weight,**
- **Outpost Weight,**
- **Number of Interpolations,**
- **Colours.**

Home Weight

This is the weight to apply to the home grid cell during the smoothing process. The home grid cell is the cell which is currently being interpolated, with all neighboring grid cells being referred to as Outposts. In the smoothing process the home grid cell is multiplied by the home weight and the outpost grid cells are multiplied by the outpost weight. The results are summed and divided by the total weight applied to get the new smoothed value of the grid cell.

Outpost Weight

This is the weight to apply to outpost grid cells during the smoothing process. The outpost grid cells are those adjoining the home grid cell, there is a maximum of 8 outpost grid cells for each home grid cell.

Number of Interpolations

This is the number of interpolations of the data to perform. Each interpolation will quadruple the amount of data. Each interpolation pass divides the horizontal and vertical grid size by 2, and applies a smoothing operation using the **Home** and **Outpost** weights. Typically four interpolation passes are adequate to give a reasonably smoothed and gridded display.

Colours

This option provides an **Input Line** and a **History Box** for selecting the colour palette to use in displaying the concentrations. Several colour palettes have been created. The first colour in the palette is used for the minimum concentration value, and the second colour is used for the maximum concentration (e.g., the red-blue palette uses red for the minimum and blue for the maximum, with intermediate values represented by colours between red and blue).

Colour Concentration versus Depth & Distance

This option is used to generate a graph of concentration versus depth and distance. The graph will have Distance on the horizontal axis and Depth on the vertical axis. Colours will be used to represent the values of Concentration. The user will be asked to specify;

- **Minimum Values,**
- **Maximum Values,**
- **Label Intervals,**
- **Axis Titles,**
- **Times,**
- **Home Weight,**
- **Outpost Weight,**
- **Number of Interpolations,**
- **Colours.**

These parameters are described earlier in this chapter.

List Output to Screen

Lists the contents of an output file on the screen. The user can use the following keys to move around in the file:

- | | | |
|-----------|---|------------------------|
| Up | » | Move up one line |
| Down | » | Move down one line |
| Right | » | Move right one column |
| Left | » | Move left one column |
| PgUp | » | Move one page up |
| PgDn | » | Move one page down |
| Home | » | View right most column |
| End | » | View left most column |
| Ctrl-PgUp | » | View top of file |
| Ctrl-PgDn | » | View bottom of file. |

The file will be displayed in a box on the left side of the screen. On the right side of the screen will be Print, Zoom, and Done buttons. These buttons will print the file, enlarge the file display to the full screen, and return to the main menu.

Lotus 123

This option allows the user to generate a Lotus 123 worksheet file using the output from the MIGRATEv9 program. A **file dialog box** is displayed, in which the user can specify the name of the worksheet file. Any files with the default .wk3 extension contained in the **data directory** will be displayed in the list box.

TOOLS MENU

Alt - T

The Tools submenu provides access to various tools that can be used to simplify data entry. These tools include:

- **Import POLLUTE file,**
- **Calculator,**
- **Advise,**
- **Add-ns.**

Import POLLUTE File

This option is used to import and convert POLLUTE data files. The user will be prompted for the name of the POLLUTE data file. After the file name has been specified, the data will be imported and converted to MIGRATE data. This converted data should then be saved as a MIGRATE data file using the Save Data As option of the File submenu.

When converting the POLLUTE data the layers are assumed to be isotropic, such that the horizontal diffusion/dispersion coefficient is assumed to equal the vertical diffusion/, dispersion coefficient. Any features in the POLLUTE data that are not available in MIGRATE are ignored (ex. initial concentration profiles and non-linear sorption). All units of the converted data are set to NONE, since no units are available in the POLLUTE data. However, if the POLLUTE data was entered using the quick landfill option, the units are preserved.

F8

Calculator

This option provides the user with an on-screen calculator to perform simple mathematical calculations. The calculator can also be used to calculate **Darcy Velocity** and **Leachate Reference Height**. The calculator can be activated any time by pressing the F8 key. The keyboard or the mouse can be used to press the

buttons on the calculator. These buttons are defined as:

- 0 - 9 » enters the digit
- . » enters a decimal
- e » enters scientific notation mode
- * » multiplies
- / » divides
- + » adds
- » subtracts
- C » clears the numeric display
- M » clears the memory
- U » changes the **units**
- H » **reference height of leachate**
- % » percent of contaminant by mass
- d » density of the waste
- t » thickness of the waste
- c » contaminant concentration
- V » **Darcy Velocity**
- i » hydraulic gradient
- k » permeability

For example, to add 2 and 13 together, press the following buttons in sequence: 2, +, 1, 3, =. The result (15) will be displayed in the upper right.

Units

The calculator can convert between several types of units for use in the calculation of **Darcy velocity** and **Leachate Reference Height**. These units are displayed to the right of the variable in memory. To convert between units press the Units button (U) and then the button of the variable (i.e., H, t, d, c, V, k). Repeat this step (i.e., U and variable button) until the desired units are displayed to the right of the variable.

Darcy Velocity

The calculator can calculate the Darcy velocity given the hydraulic gradient (i) and permeability (k). First the hydraulic gradient and permeability are entered in to memory, then the Darcy velocity button (V) is pressed.

For example, if $i = 1.2$ and $k = 2.30e-8$ cm/s, the user would perform the following operations to get the Darcy velocity:

1. Enter 1.2 and press the hydraulic gradient button (i),
2. Adjust the **units** on the permeability to cm/s, and enter 2.30e-8, then press the permeability button (k),
3. Press the Darcy velocity button (V) to get the result (0.0087), and adjust the **units** to the desired units (m/a).

Reference Height of Leachate

The calculator can calculate the Reference Height of Leachate given the percentage of contaminant (%), density of waste (d), thickness of waste (t), and concentration of contaminant (c). First the parameters (% , d, t, c) are entered in to memory, then the Reference Height of Leachate button (H) is pressed.

For example, if % = 0.1%, d = 600 kg/m³, t = 12.5 m, and c = 1000 mg/L (i.e., 1 kg/m³) the user would perform the following operations to get the Leachate Reference Height:

1. Enter 0.1 and press the percentage by mass button (%),
2. Adjust the **units** on the density to kg/m³, and enter 600, then
3. Adjust the **units** on the thickness to m, and enter 12.5, then
4. Adjust the **units** on the concentration to mg/L, and enter

Add-Ons

This submenu provides access to additional products that may be used in conjunction with MIGRATEv9, and that are purchased separately. A list is displayed showing the additional products available, the user can access a product by selecting it from the list and then pressing the OK button.

CHAPTER 8

PREFERENCES MENU

Alt - P

The Preferences submenu allows the user to control how the program interacts with the data files and devices, as well as the use of the mouse and screen colours. It has three submenus:

- **Preferences,**
- **Mouse Control,**
- **Colour Control.**

Environment

Upon startup the program reads an environment file (MIGRATE.INI) which contains user customized options for directories, file extensions, printers, colours, etc. Initially these options are set during the installation of the program. This option allows the user to customize these options, after which they are saved to the environment file by clicking on the OK (F9) button.

The user can specify the following options:

- **Program Directory,**
- **MIGRATEv9 Program Name,**
- **Data Directory,**
- **Data Extension,**
- **Output Extension,**
- **Graphic Extension,**
- **Batch Extension,**
- **Screen, • Printer,**
- **Introduction Screen,**
- **Sound,**
- **Default Colours,**
- **Screen Saver.**

Program Directory

This is the DOS directory that contains the MIGRATE-GUI program (i.e., this program) and files. The full path name should be given (e.g. c:\migrate), and can be up to 64 characters long.

MIGRATEv9 Program Name

This is the name of the MIGRATEv9 program, including the full path, up to 64 characters long (e.g. c:\migrate\mig900.exe).

Data

This is the name of the directory where the data files for the

Directory	MIGRATEv9 program are to be stored. The name should include the full path name (eg., c:\migrate\data), and can be up to 64 characters.
Data Extension	This is the default extension to be used for data files, and will be used to automatically list data files when opening a data file. The extension must start with a period and then can have up to 3 characters (e.g., .in).
Output Extension	This is the default extension to be used for output files, and will be used to automatically list output files when opening an output file. The extension must start with a period and then can have up to 3 characters (e.g., .ou).
Graphic Extension	This is the default extension to be used for graphic files , and will be used to automatically list graphic files. The extension must start with a period and then can have up to 3 characters (e.g., .gr).
Batch Extension	This is the default extension to be used for batch files , and will be used to automatically list batch files. The extension must start with a period and then can have up to 3 characters (eg. .bat).
Screen	This option lets the user select the type of graphics screen to use for displaying graphs in the Output menu. The current screen type is displayed on the input line, the user can select a different screen type by clicking on the history arrow icon to the right of the input line. When the history arrow icon is clicked a list is displayed showing all the screen types supported. The user can select a screen type or select DETECT. If DETECT is chosen the program will automatically select the screen type.
Printer	The type of printer can be selected with this option. An input line shows the current printer, a different printer can be selected by clicking on the history arrow icon to the right of the input line. When the history arrow icon is selected a list box is displayed with the printers currently supported by the program, from which the user can select a printer by clicking on it in the list box. During the installation process the user is asked for the printer to be used initially. The parameters of the printer can be specified by pressing the Printer Setup button.
Introduction Screen	This option allows the user to control whether an introduction screen is displayed when the program starts. The introduction screen is the About screen displayed under Help .
Default Colours	This option allows the user to control whether default colours or customized colours are used the next time the program starts. If the user selects default colours after customizing colours, the

customized colours will be lost.

- Sound Beeps and buzzes that occur during the use of this program can be turned on and off with this option.
- Screen Saver This option is used to turn on and off the screen saver and to specify the number of minutes without activity until the screen is saved.

Printer Setup

If the printer type is changed or if the printer setup button is pressed the printer setup dialog box is displayed. This allows the user to select:

- Paper Size,
- Resolution,
- Color Type,
- Orientation,
- Port.

If the port selected is either COM1 or COM2 the parameters for serial communication will be requested when the OK button is pressed. The serial communication parameters requested are:

- Baud rate,
- Data Bits,
- Stop Bits,
- Parity.

If the printer type is Postscript the user will also have the option of specifying the port as an output file. And if the printer type is Desktop Publishing PCX format output, the port is assumed to be an output file.

Mouse Control

This submenu is used to set various options that control how the mouse works, such as double-click speed and right and left button usage.

Mouse Double Click

Slow Medium Fast



A mouse Double Click slider bar adjusts the double-click speed of your mouse.

The Reverse Mouse Buttons option makes the right mouse button take on the normal functions of the left mouse button and vice versa.

Reverse Mouse Buttons

Colour Control

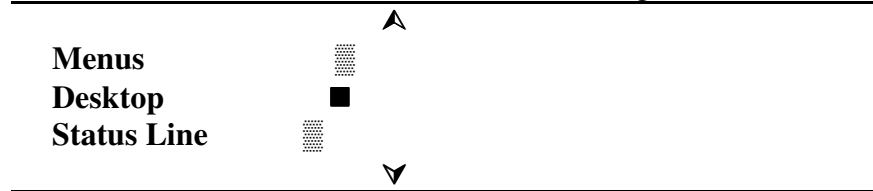
The Colours submenu is used to customize the colours of the screen. This option is composed of:

- **Group List Box,**
- **Item List Box,**
- **Colour Modification,**
- **Black and White Modification,**
- **Display Text.**

Changes do not take effect on the screen until the OK (F9) button is selected. These changes are permanently saved in the program environment file. The user can return the colour settings to their defaults by selecting the **default colours** option in the **Environment** option.

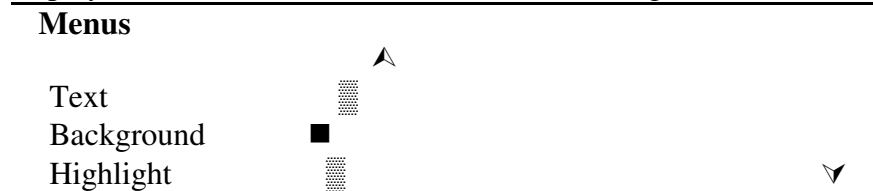
Group List Box

The Group List box contains the names of the different regions of the screen that for which the colours can be changed.



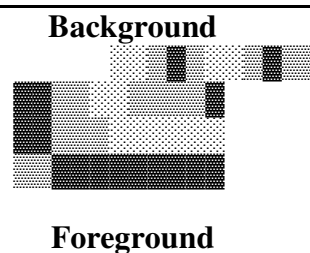
Item List Box

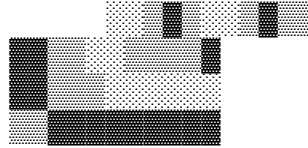
When the user selects a group from the Group list, the Item list box displays the names of the different areas in that region



Colour Modification

The screen area consists of a foreground and a background colour. The foreground colour is the colour of the text, and the background colour is the colour surrounding the text. These colours are selected by clicking on them in the colour palette boxes with the mouse.





Black & White
Modification

On monochrome systems, use the Attributes list box to select the character attributes.

- Mono High**
 - Mono Low**
 - Mono Underline**
 - Mono Inverse**
-

Display Text

The display text (above the Help button) shows the current colour or character attribute settings.

F1

The Help submenu accesses several types of information:

- **Help Contents,**
- **Help Index,**
- **Help Usage,**
- **Program Revisions,**
- **About the Program.**

Help screens can also be accessed by pressing the Help (F1) button, the help shown will be in context with the current area of the program. The help screen can be scrolled vertically and horizontally by clicking on the scroll bar. To exit the Help window press the Done or Cancel buttons (ESC or F9).

Some help topics contain highlighted (normally in green) cross-references these can be accessed by clicking on the highlighted area or on the Cross-Reference button. The user can also click on the **Contents**, **Index**, or **Usage** buttons to display help information on these topics.

**Help
Contents**

This option lists help on the main areas of the program, from which additional help information can be displayed.

Help Index

In this option all of the help topics are listed alphabetically. The user is then able to select a topic for further information.

Help Usage

This option describes how to use the help screens.

**Revision
History**

This provides a brief description of the changes that have occurred to the program in each revision.

About

This option gives a brief description of the program and the version number.

EXAMPLES

Conventions

In the examples that follow several text conventions are used to clarify the program interaction. These are:

- *italics* - instructions and comments on the use of the program,
- normal - information that appears on the screen,
- **bold** - information to be entered by the user,
- **[bold]** - menus or buttons to be selected.

Example Directory

All the examples in this appendix have been stored in a subdirectory of the MIGRATE-GUI program directory. This subdirectory is called EXAMPLES, and was created by the installation program.

The default data directory should be switched to this directory when using the examples. To change the default data directory either change the directory using the CHDIR submenu of the File menu, or change the data directory in the Environment submenu of the Advanced menu. If the data directory is changed in the Environment submenu, the program will have to be exited and restarted for the changes to be in effect.

All the input files used in the examples have a slightly different name than their already stored counterpart in this directory. The difference is that there is no CASE prefix in the files stored previously. For instance, in example 1 the input file described is called CASE1.IN and the file stored previously on disk is 1.IN. The reason for the different names is to prevent the overwriting of the supplied example files while doing the examples and saving the input data.

Example Summary

- Example 1** Shows how to create and run MIGRATEv9 for a **Subtitle D landfill** with a composite liner and constant concentration source. The flow through the composite liner is calculated using a leakage rate calculation as proposed by Giroud et. al. (1992).
- Example 2** This example demonstrates how to create a landfill with a **composite primary liner, primary and secondary leachate collection systems, and a compacted clay secondary liner**. The flow through the composite primary liner and the equivalent effective hydraulic conductivity of the geomembrane are calculated using the leakage rate calculation proposed by Giroud et al (1992).
- Example 3** Shows how to create, run, and display the output for an input dataset. Data is for the case of **pure diffusion** with constant source and zero base concentrations.
- Example 4** Edits the previously entered data in example 3 to include **advective transport and fixed outflow** in the base stratum. Shows how to open, save, execute, and display the output for the edited information.
- Example 5** This example shows how to calculate the concentration of a contaminant that originates from a **buried landfill**. Several lateral positions are investigated to gain a better understanding of the movement of the contaminant.
- Example 6** The effects of **insufficient integration** are demonstrated in this example. A solution will need more than the default integration parameters if the concentrations at a point will be small, if negative concentrations are produced, or if negative flux is calculated.
- Example 7** In this example the dataset from example 6 is modified to allow user selected **Talbot integration** parameters. Talbot parameters $N=20$ and $RNU=2$, are used with satisfactory results. All flux values become positive and some negative values at larger depths become approximately zero.
- Example 8** This example attempts to eliminate the negative concentrations in the top 5.6 m of example 7 by using user selected **Gauss integration** parameters for the Fourier integral. Ten trial solutions using 48, 99, 120, 200, 201, 202, 203, 204, 205, 250 integration steps are shown to illustrate the effect of wider Fourier integration on this problem.

- Example 9** The results of the MIGRATEv9 program are compared with an **analytical solution** given by TDAST. TDAST is a computer program used to solve 2-D plane dispersion problems for an infinitely deep porous media.
- Example 10** This case has a 1 m thick compacted clay liner underlain by a 3 m thick **fractured till layer**. The source is finite mass with a leachate collection system, and the base is an aquifer with fixed outflow. Sorption in the liner and the fractured till is also considered.
- Example 11** In this example contaminant migration from two adjacent landfills is modelled. Both of these landfills are assumed to begin operation at the same time and reach closure at the same time. The first landfill has a surface width of 300 m and the second landfill has a surface width of 600 m.
- Example 12** The model of two landfills used in example 11 is modified for the case where the landfills have different source histories. In this example the second, larger landfill is assumed to be constructed and filled 20 years later than the closure of the first landfill,
- Example 13** The termination of the operation of a primary leachate collection system in a U.S. RCRA Subtitle D landfill is modelled in this example. This landfill is the same as was used in example 1.

**GAEA Environmental Engineering Ltd.
P.O. Box 334
Waterloo, Ontario
Canada
1-519-571-8121**